G-ZERO
WinLauncher

A 32-bit Windows program that communicates with your CNCs and interfaces with other programs already installed on your computer or network.

GØ

G·ZERO

CAM Software

RAPID OUTPUT

(888) 656-1945

**www.g-zero.com**

Contents

Interface

Main Screen

This is the interface you are going to see when you first install WinLauncher. Each block of text is "hypertext" that starts its respective subroutine when you click on it.
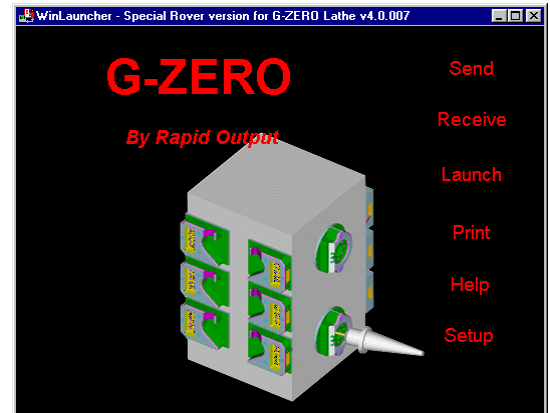
The upper right corner of the window has Windows standard Minimize, Maximize and Close buttons. The Close button is used to **exit** WinLauncher.

Additional Features
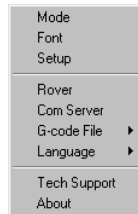
Click your right mouse button to access the *additional features* of WinLauncher.

Through this menu, you can switch to a different interface (mode), change the Font used on the main screen, access the setup features, display the technical support information, access Rover and Com Server, Resequence G-code files (insert or remove line numbers), Backplot G-code files (wireframe simulation of the G-code tool motion), and print Toolsheet.

Toolbar

The Toolbar interface has almost the same options as the main screen. The advantage of the Toolbar interface is that it frees up more space on your desktop.

The upper right corner of the window has Windows standard Minimize, Maximize and Close buttons. The Close button is used to **exit** WinLauncher.

When you click on the **Mode** button, it will change your interface back to the Main Screen.

Launcher

Click on the "Launch" hypertext to launch to any program to which your computer has access. Use the "Look in" section of the "Launch any File" window to select the drive and directory where your target file is located. If you want the program to display only the mill, lathe and tape files, select the appropriate option in the "Files of type:" section.

If the file you selected to open is a program, WinLauncher will start that program for you. Otherwise, WinLauncher will open it with its associated program. The standard "Open with" file association is used for those files without direct association.
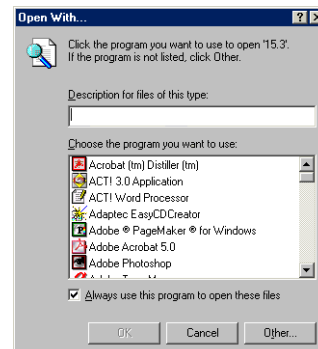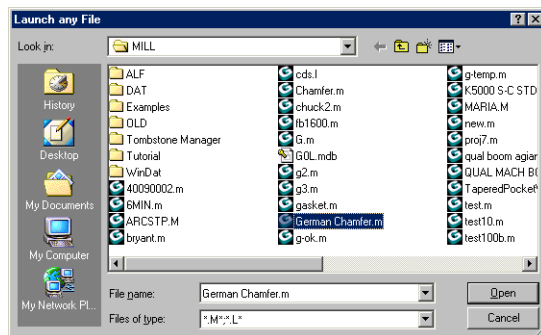
Print

Click on the "Print" hypertext to get a hardcopy of any G-ZERO or ASCII file. Use the "Look in" section of the "Print a File" window to select the drive and directory where the file you want to print is located.

WinLauncher prints by association; in other words, it will open the associated program file and allow you to use its printing features.
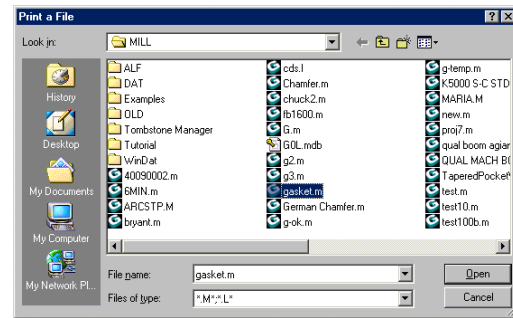
Communication

Send G-code to your CNC

Once the right connections and setup are done, you can send G-code files (tape files with file extension .T, .NC, or any other extension) directly to CNC machines:

1. From the main screen, click the "Send" hypertext to open the "Send a G-code File" window.

2. From the "Send a G-code File" window, select the file you want to send and click the "Open" button to start transmission. Note that you can use the "Look in" section on the top of the window to search for your file in other directories.
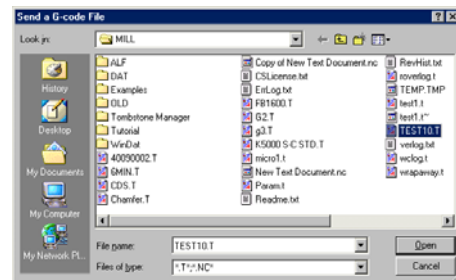
3. The "Shop Map" will be displayed. On the lower left corner of the window you will see a "Machine #" section. Use the arrow keys or click on the scroll buttons to locate the machine to which you want to send the file. Click the "Setup" button to verify or modify the machine communication setup. Click on the "Send" button when you are ready to start transmission. A safety warning sign will pop up on your screen. Press any key after reading the warning.
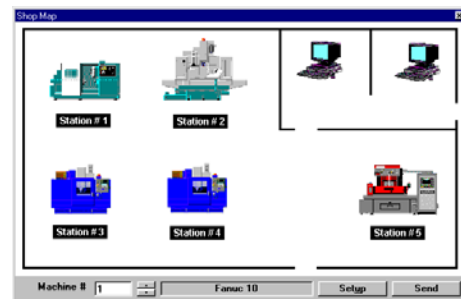
4. During file transfer, you will see a "Sending" window that shows the progress of transmission. Click on the "Cancel" button if you wish to stop transmission, or wait until the "Transfer Complete" message pops up on your screen to continue. Click on the OK button to go back to the main screen.

Customize your own program with "beforesend" and "aftersend"

WinLauncher "beforesend" and "aftersend" features allow you to automatically execute some commands or even run a program before and after a file is sent to a CNC. For example, if you want to add a line of code to all files sent to machine #12, you can do it by writing your own procedures in a file named **beforesend12.exe** and let WinLauncher do the rest.

Right before a file is sent to a CNC, WinLauncher searches for the file **beforesend#.exe** (where **#** is the CNC number) and executes its commands. If WinLauncher can't find beforesend#.exe, it will look for

beforesend#.bat (where # is the CNC number) and executes its commands. The files beforesend#.exe or beforesend#.bat need to be located in the same directory with WinLauncher.

Similarly, after a file is sent to a CNC, WinLauncher searches for the file **aftesend#.exe** and executes its commands. If WinLauncher can't find aftersend#.exe, it will look for **aftersend#.bat** and executes its commands. The files aftersend#.exe or aftersend#.bat need to be located in the same directory with WinLauncher.

Note: You can use any current programming language such as Qbasic, Visual Basic, C++, or Delphi to write the exe files. Batch files can be written with any text editor.

Receive G-code from your CNC

If G-code was altered at the machine, the updated info can be transmitted back to the computer:

1. From the main screen, click the "Receive" hypertext to open the "Receive a G-code File" window.

2. From the "Receive a G-code File" window, select the file into which you want to save the updated G-code (or you can type a new name) and click the "Open" button to start transmission. Be sure to include the extension (.T or .O or .NC, for example). Note that you can use the "Look in" section on the top of the window to use other directories. If the file name you selected already exists in that directory, WinLauncher will need to know if you want to Overwrite or Absorb the existing file. When you "Absorb" a file, you can see all the changes that were made at the CNC enclosed by the symbols < and > or << and >>; in this way you may allow or disallow the changes. Example:

   N11G0H0Z0M5
   N12M1
   <>   ← this line was deleted at CNC
   N13G90T2M6
   N14G0S2673M3
   N15E1X0Y0
   N16H2Z.1M8
   N17G83G99Z0R00Q0F<54.4><<36.>>   ← feedrate was changed from 54.4 to 36
   N18X2.Y2.
   N19G0G80   (The next time this file is sent to a CNC, the blank line, 54.4, as well as
                     the symbols < > << >> are not going to be sent)
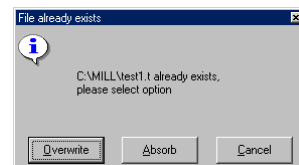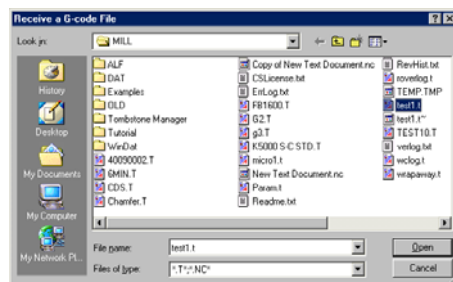
3. The "Shop Map" will be displayed. On the lower left corner of the window, you will see a "Machine #" section. Use the arrow keys or click on the scroll buttons to locate the machine from which you want to receive a file. Click the "Setup" button to verify or modify the machine communication setup. Click the "Receive" button when you are ready to start transmission.

4. During file transmission, you will see a "Receiving" window with a "Waiting" sign. Click on the "Cancel" button if you wish to stop transmission. After transmission is complete, you will be able to return to the main screen.
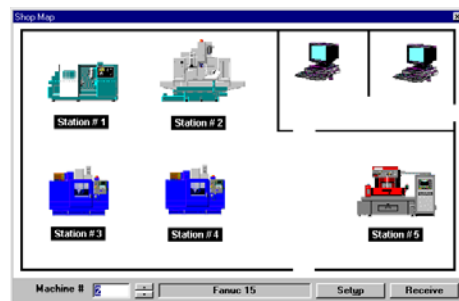
Customize your own program with "beforereceive" and "afterreceive"

WinLauncher "beforereceive" and "afterreceive" procedures allow you to automatically execute some commands or even run a program before and after a file is received from a CNC. For example, if you want all the files that come from machine #3 to be copied to the drive F:\CNC3, you can now do it by simply writing a batch file named **afterreceive3.bat** with the command **copy %1 f:\cnc3**.

Right before a file is received from a CNC, WinLauncher searches for the file **beforereceive#.exe** (where **#** is the CNC number) and executes its commands. If WinLauncher can't find beforereceive#.exe, it will look for **beforereceive#.bat** (where **#** is the CNC number) and executes its commands. The files beforereceive#.exe or beforereceive#.bat need to be located in the same directory with WinLauncher.

Similarly, after a file is received from a CNC, WinLauncher searches for the file **afterreceive#.exe** and executes its commands. If WinLauncher can't find afterreceive#.exe, it will look for **afterreceive#.bat** and executes its commands. The files afterreceive#.exe or afterreceive#.bat need to be located in the same directory with WinLauncher.

Note: You can use any current programming language such as Qbasic, Visual Basic, C++, or Delphi to write the exe files. Batch files can be written with any text editor.

Network CNC Communication

With WinLauncher *Network CNC Communication,* you can assign any computer on your network as a CNC communication server (Com Server). In this way, any computer on your network (Com Client) can automatically send and receive programs to/from your CNCs.

Here is how it works: a Com Server (computer "C" in diagram below) has a WinLauncher subroutine running in the background which is constantly checking if the file WINCOM.DAT has been changed. Whenever this change happens, Com Server can identify which Com Client (any computer on your network) is requesting a file transfer to/from a CNC. At this point, Com Server will locate the requested file to send to CNC (Com Client sends to CNC) or prepare the file to receive a transfer from a CNC (Com Client receives from CNC).
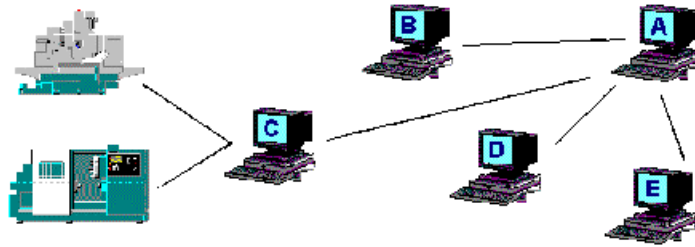

Basic requirement:

- ◆ Operating computer network.
- ◆ All CNCs connected to Com Server.
- ◆ Com Server installed with WinLauncher.
- ◆ Com Clients installed with one of these programs: WinLauncher, G-ZERO CAM for Mills, or G-ZERO CAM for Lathes.

To setup a Network CNC Communication:

1. Assign a computer on the network to be the Com Server.

2. Place WINCOM.DAT and G-ZERO G-code files in directories accessible to all computers on the network.

3. Setup Com Server in the "Setup | Options | Com Server" window.
   - Com Server **Off**: Select the Com Server OFF option if you are not using the WinLauncher Network CNC Communication.
   - Com Server **On**: Select the Com Server ON option if you are using the WinLauncher Network CNC Communication, and your computer is the communication server (computer connected to all CNCs).
   - Com Server **On Client**: Select the Com Server ON CLIENT if you are using the WinLauncher Network CNC Communication, and your computer is a communication client

4. Setup the WINCOM.DAT path. Use the Browse button to locate the WINCOM.DAT file in your computer network as set up in step 2.

Rover (option)

Rover is a WinLauncher option that allows remote and/or background file transfers between CNCs and a computer. This is a very handy feature for machinists to get files from an unattended remote computer.
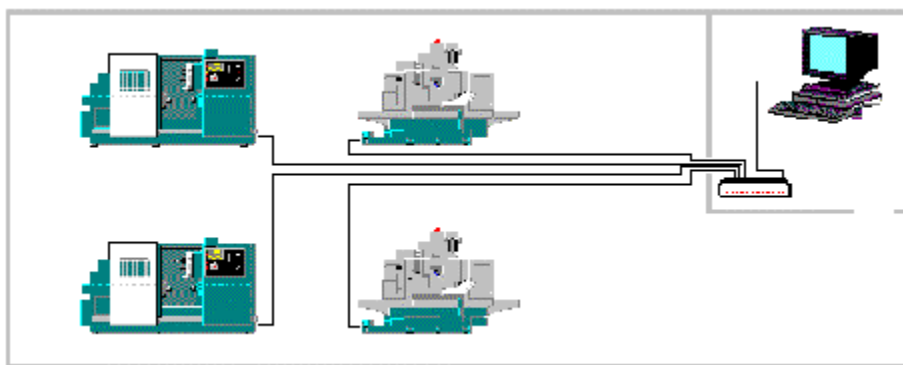
The procedure is simple:

1. Machinist sends a "fetch" file containing the name of the program file needed, its location in the computer, and the CNC's machine number.

2. Immediately after the "fetch" file is sent, machinist sets CNC to Receive or Input Mode and waits for the computer to respond.

3. Once the computer receives the "fetch" file, it automatically locates the requested file and sends it back to the CNC using its identification to make the appropriate switch on the switchbox.

Note: Baud Rate of all CNCs should be set to the LOWEST baud rate of all the machines connected to Rover.

Basic Requirement

- WinLauncher with Rover.
- WTI 16-way electronic switchbox (expandable to 64-way) or WTI 8-way electronic switchbox.
- CNC control with RS-232 both IN and OUT, and must be able to key in (MDI) the program and subdirectory name into the control's memory.

NOTE: Rover and switchboxes are sold separately.

Setup Rover

Click on the Rover Setup button located at the bottom of the WinLauncher setup window and make any changes to the default values:

Setting at Start Up. Select ON if you want to activate Rover every time you start WinLauncher. If you are running WinLauncher from a dedicated machine used only to send and receive G-code files, you may want to set it ON. (This will generally be the case.)

Prefix characters in fetch file. Some machines require a prefix character before each different component in the fetch file name. This format needs to be used on all machines connected to Rover even if only one machine requires it. If this is your case, select the ON switch.

Send characters. These are the characters used by Rover to identify the different components of the fetch file:
◆ 1st character: character(s) preceding the name of the file to be fetched/sent (default value is one slash: /)
◆ 2nd character: character(s) preceding the name of the subdirectory where the file to be fetched/sent is located (default values are two slashes: //)
◆ 3rd character: character(s) preceding the machine number from where the fetch is taking place (default values are three slashes: ///)
◆ 4th character: character(s) that denote the end of the machine number (default values are four slashes: ////)

Machine cutoff #. Your machine numbers should start with milling machines first, and followed by lathe machines. Enter here the machine number that correspond to the last milling machine on your list. Rover needs to know when to start processing for lathe machines.

Delay time in seconds. Enter here the number of seconds the computer will wait before sending the file requested by a CNC. This is to allow time for the machinist to setup the CNC to "receive" mode.

Input file extension. File extension of the file used by the fetch file.

Mill data directory. Location of the G-code files from where Rover is going to send to milling machines. The default directory is C:\MILL; use the browse button if you need to change the directory.

Lathe data directory. Location of the G-code files from where Rover is going to send to lathe machines. The default directory is C:\LATHE; use the browse button if you need to change the directory.

OK. Click the OK button when you are done with the Rover settings to save any change made.
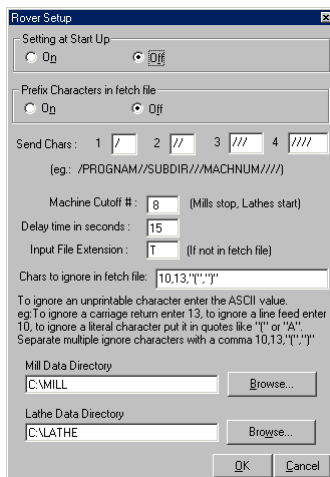
Sample fetch file:

O0099
(/PROG1234//HP-COLO///3////)
M30

Backplotter

Backplotter is a wireframe simulation of the G-code tool motion.

- ◆ Select the Backplot option from the Additional features pop up menu (right mouse button).
- ◆ From the Backplot a G-code file window, select the G-code file you want to backplot and click on the Open button.
- ◆ At this point, Backplotter will display the wireframe representation of the G-code file selected.

Top Information
Maximum and minimum values of the X, Y, and Z dimensions of the area used by the tools.

Blue box
Three dimensional representation of the X, Y and Z dimensions of the area used by the tools.

Green solid lines
Path followed by tools while feeding.

Esc - Exit
Exit Backplotter and return to G-code Editor.

F2 - Away
Rotate wireframe up and away from the user.

F3 - Toward
Rotate wireframe down and toward the user.

F4 - Left
Rotate wireframe to the left.

Dbl mouse - Zoom in/out (*)

F7 - DXF Export
Save wireframe representation as a DXF file in the current directory using the same file name.

F6 - Single
Read through G-code and draw wireframe one block at a time.

F5 - Right
Rotate wireframe to the right.

(*) Note: **Zoom in:** (a) Position cursor at a corner of the area to be zoomed. (b) Press BOTH mouse buttons and move cursor to the opposite diagonal corner of the area to be zoomed. (c) Release mouse buttons to redraw the zoomed area.

Zoom out: Press and release BOTH mouse buttons simultaneously without moving the mouse.

Configure Backplotter

BP.UNI is the Backplotter initialization file located in the same directory as your WinLauncher. BP.UNI is a text file and it can be edited with any text editor. This is a copy of the BP.UNI file:

Colors are:

1) Blue	5) Magenta	9) Bright Blue	13) Bright Magenta
2) Green	6) Brown	10) Bright Green	14) Yellow
3) Cyan	7) Gray	11) Bright Blue	15) White
4) Red	8) Dark Gray	12) Bright Red	0) Black (invisible)

1. What color should tool 1 be? 2_
2. What color should tool 2 be? 3_
3. What color should tool 3 be? 4_
4. What color should tool 4 be? 5_
5. What color should tool 5 be? 6_
6. What color should tool 6 be? 9_
7. What color should tool 7 be? 10
8. What color should tool 8 be? 11
9. What color should tool 9 be? 12
10. What color should tool 10 be? 13

11. What color should the text be? 15
12. What color should the box be? 7_

13. What signifies the end of the main program? M02___
14. What else for end of main? M30___

15. What letter comes just before a sub number? O
16. What signifies a sub call? M98___
17. What letter comes just before the sub jump? P
18. What signifies the end of a sub? M99___

19. What signifies a clear screen? [CLEAR__
20. What signifies a pause? M00_____
21. What comes just before a new color callout? [COLOR__

22. What signifies a tool change? M6__
23. What letter comes just before a tool number? T
24. If T preselect, what must NOT be on the tool change line? _____

25. What signifies a tool dia callout? [TDIA____

26. Would you like all Lathe plots flipped in X (Y,N) ? Y
27. Are IJs (center point of arcs) Absolute (Y,N) [usually N]? N
28. Should mill plots start in iso view (Y,N)? N

Note: for line numbers 1-10, 13-14, and 25, you may embed tool diameter information so that drill holes look more realistic.

Setup

Click on the "Setup" button from the main screen to display the Setup window. All settings are grouped and controlled by five tabs and one button:

- Com Setup
- Options
- Directories
- Map Settings
- 1 Time Options
- Test
- Rover Setup

Since this section contains the setup of the main features of WinLauncher, an Administrator button has been included for your protection from unwanted changes. You can assign a password to prevent unauthorized users from changing various setups that may affect the performance of WinLauncher.

Note: Make sure to click the "OK" button after customizing your settings.

Com Setup

Set the communication settings for all CNC machines.

- **Number.** Machine number you need to define when you Send G-code files to CNC machines or Receive files from CNC machines. This number is automatically set by WinLauncher; you can have up to 100 machines.
- **Name.** Description or name to identify the CNC machine. The first one is reserved for the "switchbox" setup.
- **Port.** COM port (serial port) number through which CNCs are connected with the computer. This number can be any number from 1 to 25.
- **Baud.** Communication speed. Choose from any standard baud rate between 110 and 28800. Default baud rate: 1200.
- **Data.** Data bit should be either 7 (seven) or 8 (eight). Default data bit: 7.
- **Parity.** Parity should be either EVEN or ODD. Default parity: EVEN.

- **Stop.** Stop bit should be either 1 (one) or 2 (two). Default stop bit: 1.
- **Flow.** Flow control (communication protocol) can be XON/XOFF, Hardware controlled, or Both. Default flow control: XON/XOFF.
- **Format.** Code format should be either ASCII/ISO or EIA. Default code format: ASCII/ISO.
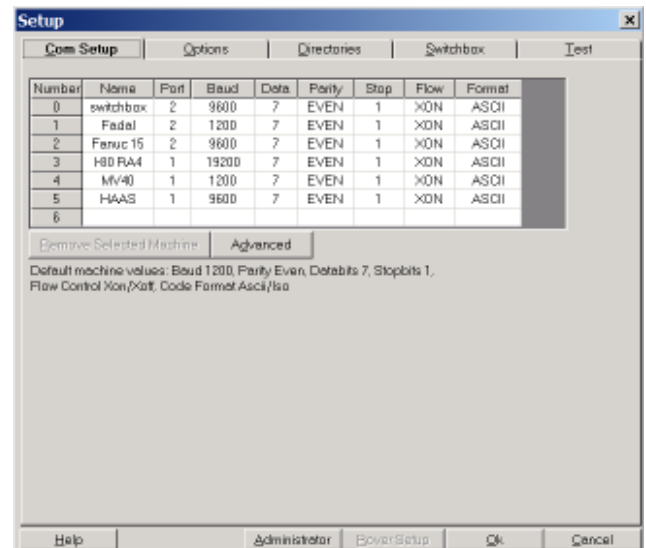
To add a new machine:

The last line is always empty to allow you to enter a new machine.

1. Click on the corresponding box for Name, and type in the name or description of the new machine.
2. Key in or choose the rest of the communication setup values for the new machine. If you leave any box empty, WinLauncher will assign the default value.
3. Click on the OK button when done.

To remove a machine:

1. Use your mouse to select the name of the machine you want to remove. If a machine name is not selected, WinLauncher will remove the machine that corresponds to the selected communication setup value.
2. Click on the "Remove Selected Machine" button. WinLauncher will automatically resequence the machine numbers.
3. Click on the OK button when done.

To change a machine name:

1. Click on the machine name you want to change.
2. Use the standard editing keys (backspace, delete, arrow keys) and enter the new name.
3. Click on the OK button when done.

Note: You cannot edit the "switchbox" name.

To change communication setup values:

1. Click on the setup value you want to change.
2. Different options are going to appear in a pull-down menu.
3. Select the option that best suits your arrangement.
4. Click on the OK button when done.

To change a machine number:

Machine numbers are automatically setup by WinLauncher so you cannot edit them. However, you move the whole row to the machine number you wish.

- To move a machine one row up:
    1. Click on the machine name you want to move.
    2. Click on the Arrow-Up button located on the right side of the window. WinLauncher will automatically resequence the rest of the machines.
    3. Click on the OK button when done.
- To move a machine one row down:
    1. Click on the machine name you want to move.
    2. Click on the Arrow-Down button located on the right side of the window. WinLauncher will automatically resequence the rest of the machines.
    3. Click on the OK button when done.

Note: Machine numbers have to be unique values; no duplicate machine number is allowed.

Advanced button

The Advanced button adds three columns of communication settings:

- **ICD.** Inter-character delay: number of milliseconds of delay between characters when sending files to CNCs (required by some machines). Default value: 0 (zero)
- **ILD.** Inter-line delay: number of milliseconds of delay between lines when sending files to CNCs (required by some machines). Default value: 0 (zero)
- **EOL.** End of Line: Default value: NONE

Note for Heidenhein machines: WinLauncher automatically adds the Heidenhein special numbering sequence to the tape file before sending it to the machine that requires it.

Options

Com Server (Network-CNC Communication)

- ◆ **Off:** You are NOT using WinLauncher Network-CNC Communication.
- ◆ **On:** You are using WinLauncher Network-CNC Communication <u>and</u> your computer is the Com Server (Network-CNC communication server). In other words, your computer is connected to all CNC machines.
- ◆ **ON Client:** You are using the WinLauncher Network-CNC Communication <u>and</u> your computer is the Com Client (Network-CNC communication client).
- ◆ **File:** Use the Browse button to locate the file WINCOM.DAT in your network.
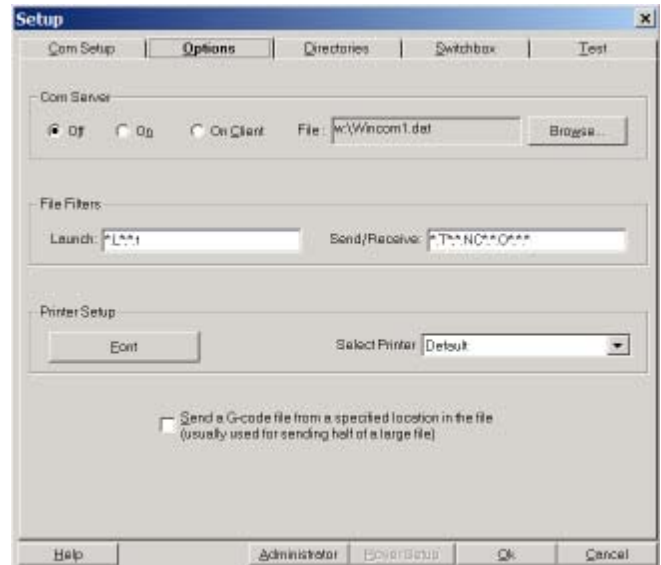
File Filters

Use this section to customize the file filters in your launching, sending and receiving windows. Use the asterisk as a wildcard to replace all or part of a file name or extension to be displayed. Examples:

*.m   to display all files with a .m extension

.t   to display all files with extension beginning with the letter T

*.nc   to display all files with extension .nc

ab*.*   to display all files beginning with the letters ab, regardless of their extension

Printer Setup

When printing source or G-code files, you can select the printer you want to use, and the font (font, font style, size) of the text to be printed.

Send from the Middle

Sometimes, you may encounter difficulties sending very large G-code files to CNCs. Your computer may not be able to send the complete G-code file. In this case, use the Otions tab of the WinLauncher Setup to set the mid-tape location in your G-code file from where you want to start the transfer.

1. Click on the box to select "Send a G-code file from a specified location in the file."
2. Enter a search string from where you want to start sending the G-code. Example: the string could be your line number (eg.: N255).
3. Click the OK button at the bottom of the window.
4. Proceed with the normal steps for sending a G-code file to CNC.

Directories

File Dialog

Use this section to select the type of dialog interface you want to use: Standard or Incremental Search. Once you selected your preferred method, that is going to be the one used by the Send, Receive, Launch, and Print sections of WinLauncher.

- **Standard:** This is the default dialog interface. See the Send, Receive, Launch, and Print sections of this document for more details.

- **Incremental Search:** The Incremental Search dialog interface is very similar to the Standard dialog interface. Its main advantage is that as you type characters in the first box, the display file box will narrow down the search.
  Example: the d:\mill directory has twenty files (with extensions ".T" or ".NC"). As you type the character "T" in the first box, the display file box lists only the four files that start with the character "T". As you add more characters to the text box, the display file box will continue narrowing down the list to match your file search.
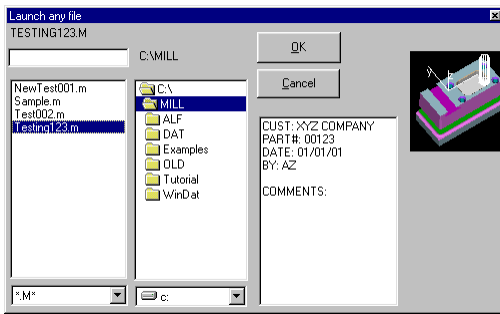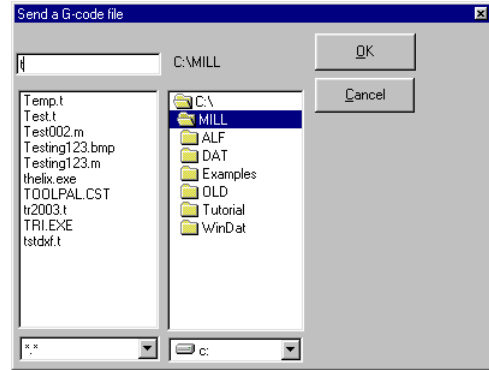
Another benefit of incremental search is that as you click on a file name, information about the file shows in the lower right-hand corner of the dialog. In fact, if you save a bitmap (BMP) of your source files, then a small picture also displays. If you own G-ZERO Mill or Lathe, you can create bitmaps by clicking on the camera icon (top of viewport), or by right-clicking on the viewport and selecting Solid View Settings|Save Snapshot on Exit. This automatically creates a bitmap every time you view (and exit from) a source program.

Same Start Up Directory

This check box allow you to set up the default drive and directory to display when you use the Send, Receive, Launch, and Print options. Some versions of Windows have their own mechanism for 'remembering' your last folder, so this feature may not work for you.

- Use the Browse button to select the default drive and directory, *OR*
- Key in the path you want. Note that no changes will be made if the path does not exist.

Mill Program File

This section sets up the path and file name that correspond to the executable file for the program G-ZERO CAM Software for Mills.

- Use the Browse button to select the default drive and directory, *OR*
- Key in the path you want.

Lathe Program File

This section sets up the path and file name that correspond to the executable file for the program G-ZERO CAM Software for Lathes.

- Use the Browse button to select the default drive and directory, *OR*
- Key in the path you want.

Switchbox (and Map)

If your computer is connected to only one CNC, or you are using a manual switchbox rather than an electronic switchbox, WinLauncher does not need to ask you to identify the machine to which you are communicating. If this is your case, click on the box provided on the Switchbox tab.
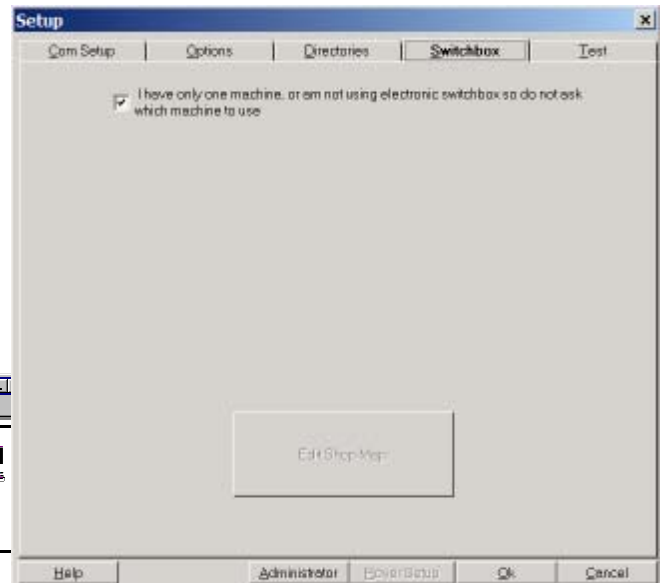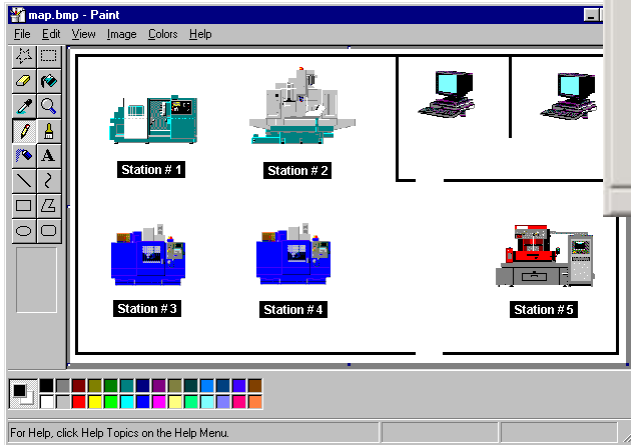
If you are using an electronic switchbox, WinLauncher will display a floor plan of your shop. This is a bitmap file that can be modified with a Paint program by clicking on the Edit Shop Map button.

Electronic Switchbox Setup

If you uncheck the box called "I have only one machine...", then use this section to setup the electronic (or manual) switchbox, if you have one.

1. Select the appropriate type of switchbox: None/mechanical (manual), 16-way WTI, 8-way WTI, etc.
2. Read your switchbox documentation and DIP switches for the control code and offset code, and then enter the ASCII code numbers in the corresponding area. For example, if you swtchbox requires two ^A codes (ASCII 1) followed by a 1 to switch to port 1, then enter 1,1 for 'On Code(s)' and 48 (shifts 1 up to ASCII 49, which is the '1' character). If the code to put the switch back into ready mode is an escape, followed by two nulls then enter 27,0,0 in the 'Off Code(s)' section.

Test

Electronic Switchbox Test Mode

Select the Electronic Switchbox option if you want the switchbox codes to be displayed at the top of the WinLauncher main screen when sending and receiving G-code files. This feature is very handy for testing and debugging the electronic switchbox operation.

Auto Detect Ports

Click on the Auto Detect Ports button to have WinLauncher automatically check your current Com ports (serial ports) settings. The test takes only a few seconds and the results will appear in a window similar to the one shown below.

- Green Light means that the Com port is not connected to any device. You can use it for a new connection.
- Yellow Light means that the Com port is currently connected to another device (usually a modem).
- Red Light means that the Com port is not available in your computer.
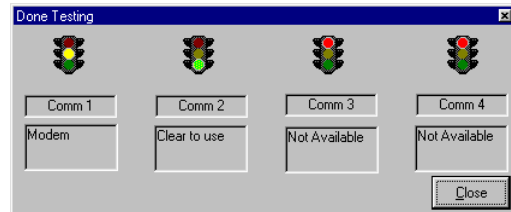
System Properties

Click on this button to open your computer System Properties to display information such as the version of your Windows system, device manager, hardware profiles, and computer performance.   The same window can be opened from your Windows: Start | Settings | Control Panel | System (on most operating systems).

Terminal Mode

This section was designed for the advanced user who wants to have access to the terminal mode to communicate with CNCs. You can use the terminal mode to test the WTI switchbox, Com ports, and cables.

- Click on the ON button to open the terminal mode screen.
- Click on the OFF button to close the terminal mode screen.

Administrator

Click the Administrator button to enter a password, so that all further changes to the setup can only be made by you.

Without the password, future visitors to the Setup Dialog can only change the printer and printer font. They can also send a G-code file from the middle. All other WinLauncher features remain available to all users.