Techno *CNC*

GCODE Interface
For DaVinci & C-Series Controllers
Windows Version

Techno-Isel
2101 Jericho Turnpike
New Hyde Park, NY 11042-5416
Phone: (516) 328-3970
Fax: (516) 358-2576
www.techno-isel.com
support@techno-isel.com

HTM00490111

Techno GCODE Interface
For DaVinci & C-Series Controllers
Windows Version

© 2001 Techno, Inc.

Software License Agreement

1. **Grant of License:** Techno, Inc. grants right of usage for one copy of the Techno GCODE Interface on a single computer.

2. **Copyright:** This software is owned by Techno, Inc. and is protected by United States copyright laws. This software may only be duplicated to produce a single copy for backup purposes or to transfer the software to a single hard disk.

3. **Modification:** Modification of this software in any way is forbidden.

4. **Transfer:** Transfer of this software is permitted only when all copies and documentation are also transferred.

Limitation of Warranties and Liability

All Techno manuals are produced by qualified people according to rigorous guidelines. However, Techno makes no warranty, expressed or implied, that this manual is free of errors or that products described herein are suitable for any specific application. Techno, Inc. assumes no responsibility for loss or damages resulting from this manual or from use of the products herein. Techno reserves the right to alter its hardware, software, and any applicable documentation without notice.

Techno makes no warranty that its products are fit for any use or purpose to which they may be put by the customer, whether or not such use or purpose has been disclosed to Techno in specifications or drawings previously or subsequently provided, and whether or not Techno's products are specifically designed and/or manufactured for such use.

THIS WARRANTY IS IN LIEU OF ALL OTHER WARRANTIES, EXPRESSED OR IMPLIED.

ALL OTHER WARRANTIES ARE HEREBY DISCLAIMED.

SAFE OPERATION OF YOUR MACHINE

Read these instructions thoroughly BEFORE operating machine

1. Keep fingers, hands, and all other objects away from machine while power is on.

2. Disconnect power to all system components when not in use, when changing accessories, and before servicing.

3. Do not loosen, remove, or adjust machine parts or cables while power is on.

4. Exercise care with machine controls and around keyboard to avoid unintentional starting.

5. Make sure voltage supplied is appropriate to specifications of components.

6. Machines must be plugged into three-pronged grounded outlets. Do not remove the grounding plug or connect into an ungrounded extension cord.

7. Keep cables and cords away from heat, oil, and sharp edges. Do not overstretch or run them under other objects or over work surfaces.

8. Use proper fixtures and clamps to secure work. Never use hands to secure work.

9. Do not attempt to exceed limits of machine.

10. Do not attempt to use machine for purposes other than what is intended.

11. Use machine only in clean, well-lit areas free from flammable liquids and excessive moisture.

12. Stay alert at all times when operating the machine.

13. Always wear safety goggles.

14. Do not wear loose-fitting clothing when operating machine. Long hair should be protected.

15. Always maintain proper balance and footing.

16. Maintain equipment with care. Keep cutting tools clean and sharp. Lubricate and change accessories when necessary. Cables and cords should be inspected regularly. Keep controls clean and dry.

17. Check for damaged parts. An authorized service center should perform all repairs. Only identical or authorized replacement parts should be used.

18. Remove any adjusting keys and wrenches before turning machine on.

Table of Contents

I. The Techno GCODE Interface

The Techno GCODE Interface translates previously generated GCODE toolpath commands and sends them to your DaVinci or C-Series controller, which moves the machine to produce your part. The GCODE Interface also:

- Makes mass production possible by writing files that can be downloaded and run directly from your controller.

- Creates separate configuration files for each of your GCODE programs.

- Previews GCODE toolpaths before running your program.

- Translates and executes your file simultaneously or translates the file and stores it to be run later.

Quick to learn, easy to use!

Download and run files from your controller

Preview toolpaths

Create custom configurations for each file

The Techno GCODE Interface disk may include a README file containing information about the software unavailable at the time this manual was printed. Please read this file.

Installing the GCODE Interface Software

System Requirements

To use the Techno GCODE Interface, you must have a PC with these minimum requirements:

- **Windows 95**
- **486 processor (we recommend Pentium)**
- **one available serial port**

Loading the Software

1. Turn on your computer and wait for Windows (95 or 98) to boot up.

2. Insert the GCODE Interface disk.

3. Click   **Start**

4. Select and click   **Run...**

5. Type **a:\setup** in the Run Dialog Box**.**
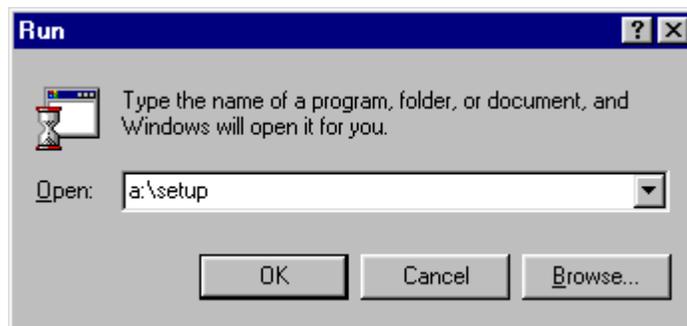
6. Click   **OK**

7. Follow the simple on-screen instructions from here to install the software.

- **Make sure all other applications are closed before attempting to install the Techno GCODE Interface.**

The Techno GCODE Interface disk may include a README file containing information about the software unavailable at the time this manual was printed. Please read this file.

GCODE INTERFACE MENU MAP

Select File

Preview your toolpath

Jog to your toolpath origin.

Setup system for your file

Select File

Select File

Select File

Jog to your toolpath origin

Jog to your toolpath origin
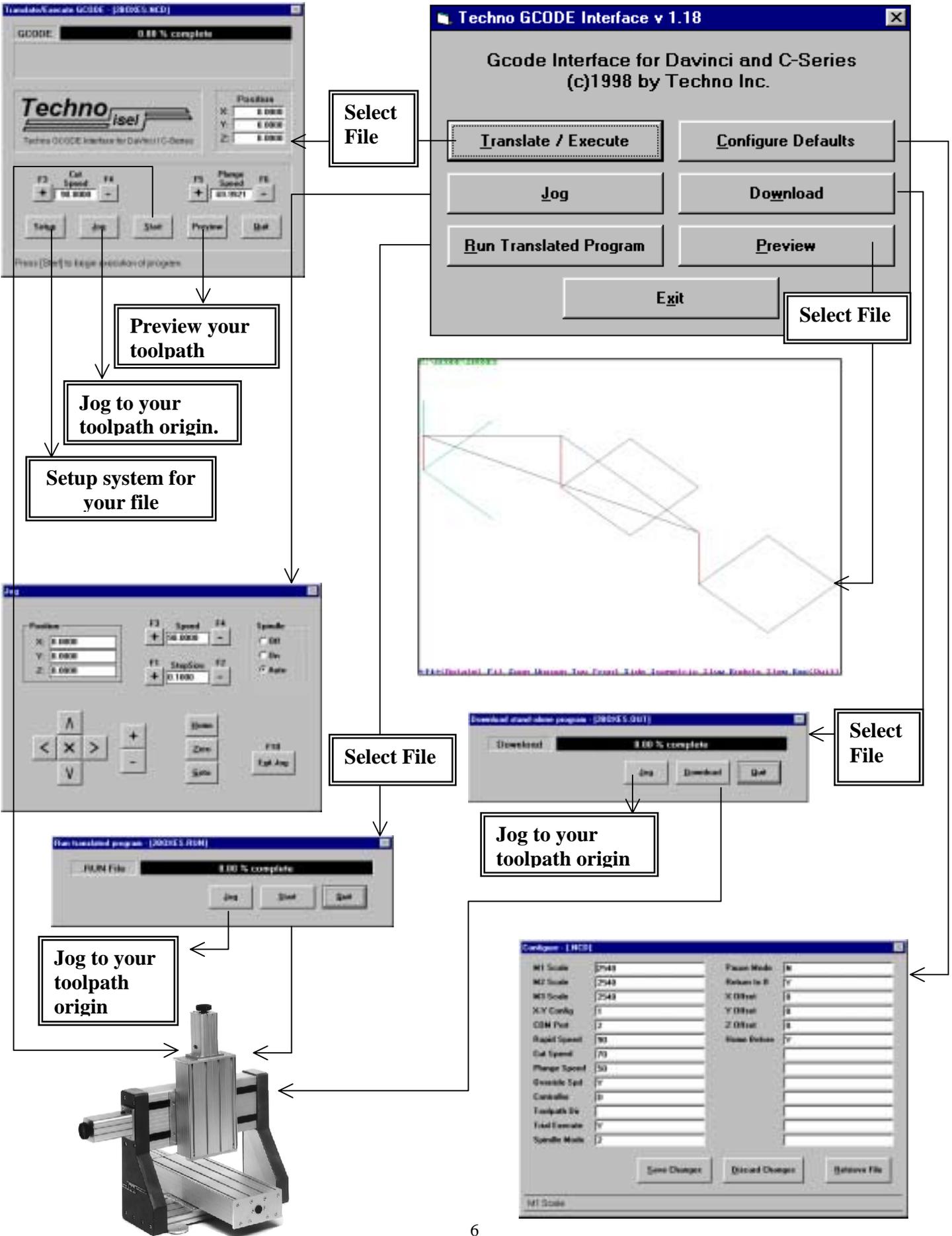
II. Quick-Start Tutorial

This simple tutorial employs and explains the main functions of the Techno GCODE Interface by cutting the sample program, **2boxes.ncd**. If you have difficulty with the tutorial or are just learning to use your machine, we recommend reading the manual completely before proceeding.

- **Before using this software, consult your *DaVinci Setup* or *C-Series Wiring and Setup* manual to ensure proper machine setup.**
- **Capitalized words will be explained in greater detail in the following section.**
- **Move machine away from HOME position when beginning by rotating knobs on the end of each motor.**
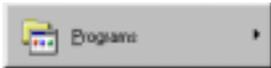
1. From the **Start** menu, select **Programs** ▸ and then **Techno GCODE Interface**
   The Interface main menu appears:
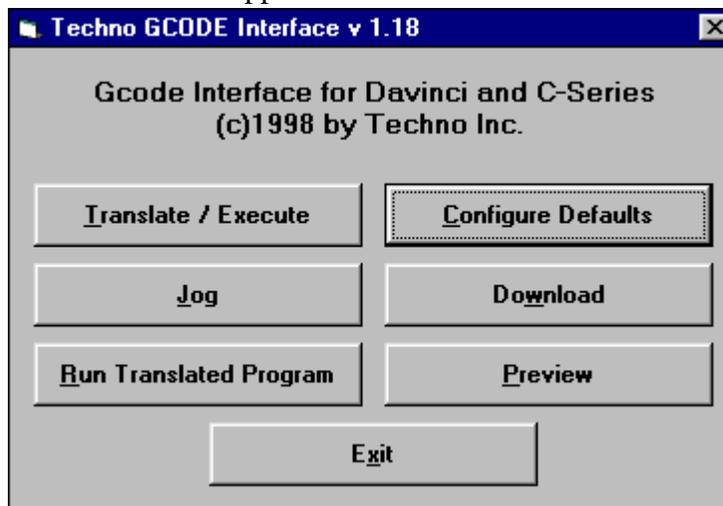
   Click **Configure Defaults**
   From here, system default parameters can be adjusted.

2. The GCODE Interface's default settings are displayed (see below). These parameters will be applied to the sample program, **2boxes.ncd**. You may have to change CONTROLLER and COM PORT settings to indicate your controller model and the communication port on your PC the controller is plugged into.

   The configuration screen is shown below with each function briefly described. If the terminology is new to you, it is a good idea to read the *Explanation of Functions and Terms* section before proceeding.

Configure - [.NCD]

a)	M1 Scale	2540	n)	Pause Mode	N
b)	M2 Scale	2540	o)	Return to 0	Y
c)	M3 Scale	2540	p)	X Offset	0
d)	X-Y Config	1	q)	Y Offset	0
e)	COM Port	2	r)	Z Offset	0
f)	Rapid Speed	90	s)	Home Before	Y
g)	Cut Speed	70			
h)	Plunge Speed	50			
i)	Override Spd	Y			
j)	Controller	D			
k)	Toolpath Dir				
l)	Trial Execute	Y			
m)	Spindle Mode	2			

Save Changes   Discard Changes   Retrieve File

M1 Scale

a) **M1 SCALE** is the **SCALE FACTOR** for Motor 1,which moves the x-axis in this (+1) X-Y Configuration.
b) **M2 SCALE** is the scale factor for Motor 2,which moves the y-axis in this (+1) X-Y Configuration.
c) **M3 SCALE** is the Motor 3, or z-axis scale factor**.**
d) **X-Y CONFIGURATION** orients the axes to your specifications
e) **COM PORT** identifies which port on your PC the controller is plugged into. Set this to **1** or **2**.
f) **RAPID SPEED** is the speed of the tool when not cutting.
g) **CUTTING SPEED** is the speed of the tool when cutting.
h) **PLUNGE SPEED** is the speed of the Z-axis when descending into your material.
i) **OVERRIDE SPEED** uses speeds set in Configure Defaults rather than those set in your GCODE program.
j) **CONTROLLER** identifies the model of your controller.

	Enter:		
	D	for	**D**aVinci
	W	for	**W**izard
	4	for	C-Series **4**.0 (e.g. C10)
	5	for	C-Series **5**.0 (e.g. C142)
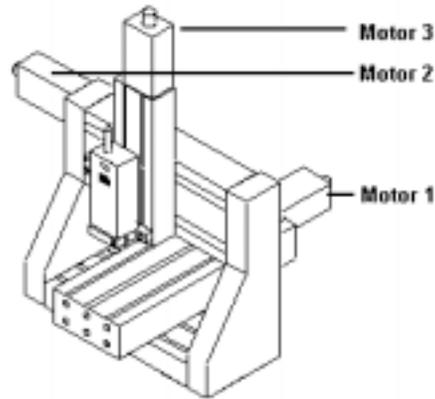
k) **TOOLPATH DIRECTORY** shows the directory where toolpath files are located.
l) **TRIAL EXECUTE** lets you translate files with or without executing them.
m) **SPINDLE MODE** turns the spindle on and off, or sets it to automatic mode.
n) **PAUSE MODE** lets you override toolchange and other programmed pauses.
o) **RETURN TO 0** returns the machine to your toolpath origin (0,0,0) after running a program.
p) **X-OFFSET** moves the machine, along the x-axis, the number of units specified away from the toolpath origin before your program is run.
q) **Y-OFFSET** serves the same function as X-Offset, except along the Y-axis.
r) **Z-OFFSET** is this same function again, this time along the Z-axis.
s) **HOME BEFORE** sends all axes to the home position before running a program.

3. Click [**Save Changes**]

    Anything you have changed will become the new default setting for the GCODE Interface and you will return to the main menu.

    Or, click [**Discard Changes**] to simply return to the main menu.

4. Click [**Translate / Execute**] from the main menu.

    An Open File dialog box appears.
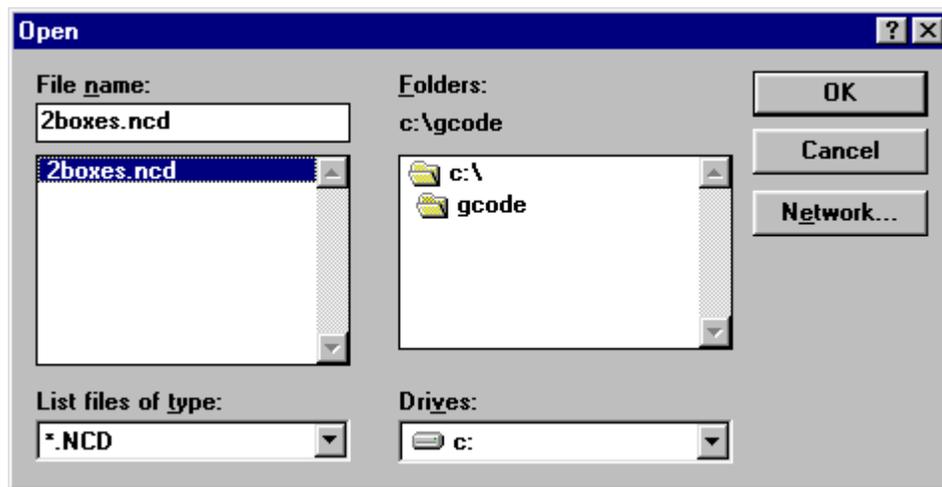
5. Select **2boxes.ncd** by double clicking it, or highlighting and clicking [OK]

The Translate/Execute window opens.


```
Translate/Execute GCODE - [2BOXES.NCD]                          ☒

  GCODE              0.00 % complete


  Techno isel                              Position
                                       X:      0.0000
                                       Y:      0.0000
  Techno GCODE Interface for DaVinci / C-Series     Z:      0.0000


        Cut                      Plunge
   F3   Speed    F4         F5   Speed    F6
   +  90.0000   –          +  69.9921   –


   Setup      Jog      Start    Preview     Quit


  Press [Start] to begin execution of program.
```
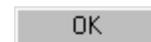
6. Click [Setup] to double-check configuration for **2boxes.ncd**.

Setup lets you adjust parameters for the **.NCD** file you want to run. The system configuration created in Steps 2 and 3 appears. You'll notice that Controller COM Port, and Toolpath Directory are locked out in Setup.

- Setup applies only to the program you are running and is independent of system configuration defaults.

7. Click [Discard Changes] to return to Translate/Execute.

8. Click [Jog]

Jogging manually positions the axes at an appropriate starting point. This starting point is called the TOOLPATH ORIGIN.

The Jog window opens.

Current axis position. Resets to (0,0,0) when you Exit Jog

Position
X: 1.5000
Y: 2.5000
Z: -0.5000

F3 Speed F4
+ 90.0000 –

F1 StepSize F2
+ 0.1000 –

Spindle
○ Off
○ On
⊙ Auto

Move z-axis up (+) and down (-)

Sends axes to machine's HOME position

Face the front of machine and these directional keys move X and Y-axes in direction indicated

Λ
< X >
V

+
–

Home
Zero
Goto

F10
Exit Jog

Resets position counters (0,0,0) and returns to previous menu
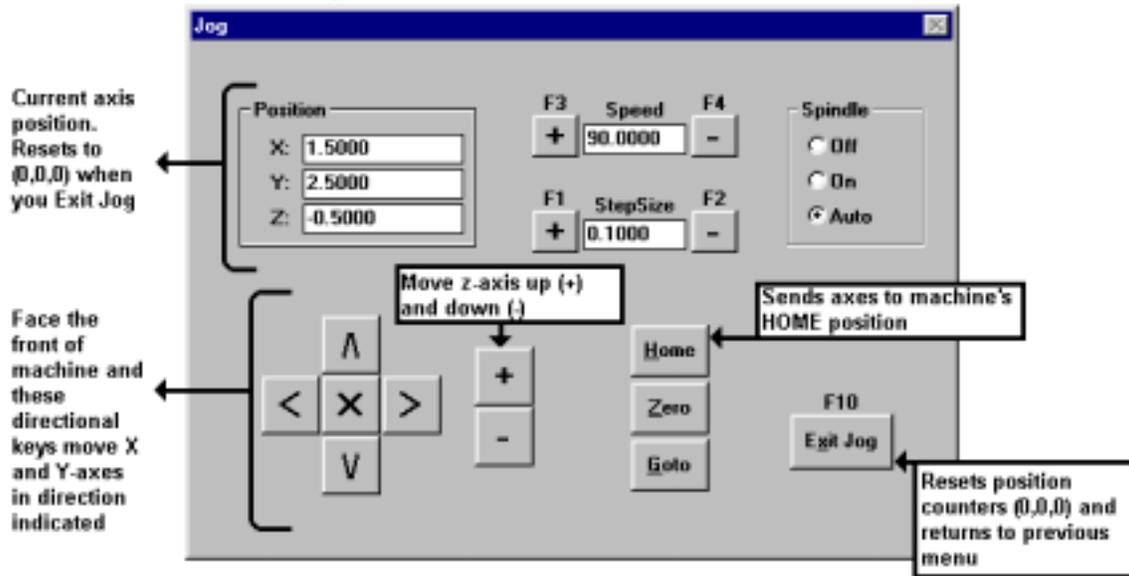
9. Click **Home**

From the pull-down menu, select Home All.

The machine is now in the Home position and counter reads (0,0,0).

cancel
Home X
Home Y
Home Z
Home All

10. Click Λ V > < + – icons to jog machine to the position displayed at upper left in the window above. (This position is an approximation. Do not worry if you are not at this precise position.) This will be the toolpath origin.

11. Click **Exit Jog** to return to Translate/Execute.

Before translating and executing **2boxes.ncd**, we will preview the toolpath.

12. Click **Preview**

- Preview allows you to view the design from several different angles, rotate the image, or look at the program step by step. These functions are shown along the bottom of the Preview screen and are activated by pressing the corresponding highlighted keys.

C:\GCODE\2BOXES

File name and directory

Your GCODE toolpath (isometric view)

Manipulate and alter perspective of the image using these keyboard commands

←↑↓→(Rotate) Fit Zoom Unzoom Top Front Side Isometric Slow Endpts Step Esc(Quit)
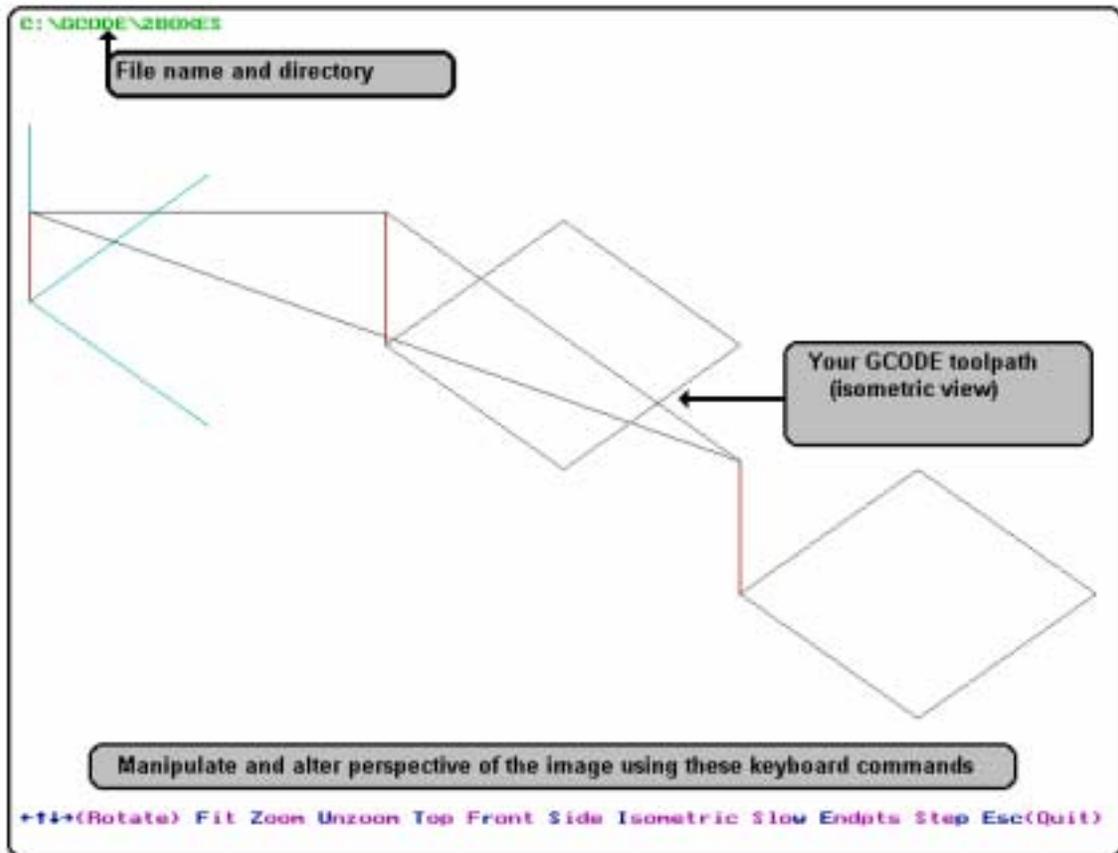
13. Press the **Esc** key to Quit Preview and return to Translate / Execute. You are now ready to run **2boxes.ncd**.

14. Click [**Start**]   from Translate/Execute.

The Interface simultaneously translates and executes **2boxes.ncd** on your machine. The window is shown below as it appears while running a program:

Percentage of program completed

GCODE   41.14 % complete

Program commands currently being executed

Y2 F40.0
X2
Y1

Techno isel

Techno GCODE Interface for DaVinci / C-Series

Position

X: 2.0000
Y: 1.0000
Z: -0.2500

Current position of axes

F3   Cut Speed   F4   F5   Plunge Speed   F6

+   69.9921   –   +   50.0079   –

Pause allows you to stop at any point and resume from where you've left off.

Pause   Jog   Start   Return   Halt

Halt allows you to abort the program at any time

Program is executing.

15. The sample program is complete. A pop-up box appears.

Techno GCODE Interface v 1.18

Program execution completed.

OK

Click   OK
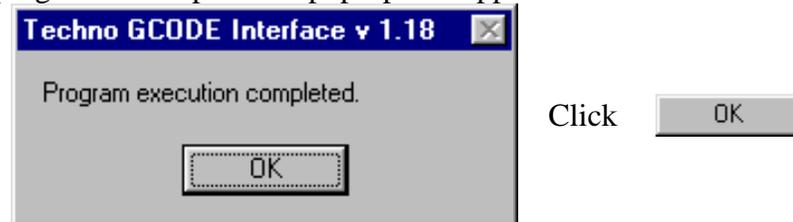
16. Click   **Quit**   to return to the main menu.

■   Clicking   **Pause**   stops the machine after it has completed the current motion.

You may   **Resume**   from this point.   **Halt**   also stops the machine after

the current motion is completed, but does not allow you to resume.

III. Explanation of Functions and Terms

This chapter explains the Techno GCODE Interface's functions. Terms encountered when using the GCODE Interface are defined when necessary. You'll also find:

- Quick, simple practice lessons to familiarize you with features of the Interface.
- Time and headache saving shortcuts.
- Warnings against misuse that might tie up production.

Your GCODE file must have an **.NCD** extension in order to be translated by the Interface. Setup parameters created by the Interface for your program will be saved in a separate file with a **.CFG** extension. This configuration file **(.CFG)** is applied to the toolpath motion command file **(.NCD)** to write two additional files, one with a **.RUN** extension (section 4 of this chapter) and the other a standalone file with an **.OUT** extension (section 5 of this chapter).

- **The interface automatically adds appropriate file extensions to your program.**

File Extensions and Description

Extension	Sample File Name	Description
.NCD	2boxes.ncd	GCODE toolpath file
.CFG	2boxes.cfg	Configuration file created by Interface and user input
.RUN	2boxes.run	Motion commands interpreted for machine by Interface
.OUT	2boxes.out	Standalone program for controller

The *Explanation of Functions and Terms* is divided into six sections which correspond to the six options (not including Exit) offered by the Main Menu. This section is ordered to first introduce unfamiliar terminology and then apply this terminology to the functions of the Techno GCODE Interface.

The **MAIN MENU** appears when you start the Techno GCODE Interface. The main menu offers six options. These options are numbered on the following page according to how they are presented in this section.
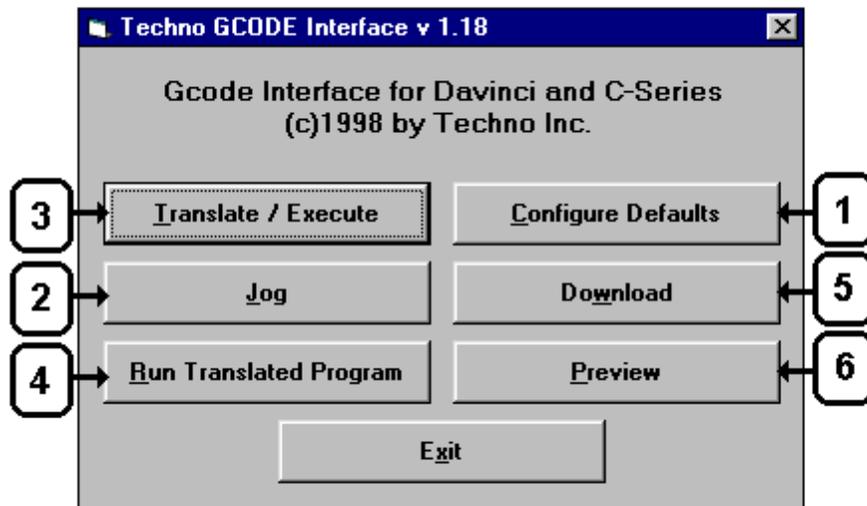
1. Configure Defaults

Configure Defaults tells the Interface:

- The model of your controller
- The communication port on your PC the controller is plugged into
- The location of the directory containing your toolpath files
- Default settings that will be used to create a new **.CFG** file when you first translate your **.NCD** file

Configure Defaults should not be confused with **Setup** which is used only to make changes in configuration for the particular **.NCD** file you have opened from the **Translate / Execute** window.

The **Configure Defaults** Title Bar will ALWAYS read:

Configure - [.NCD]

This means that all changes will be made to the system default configuration file **(default.cfg)**.

The **Setup** Title Bar will contain the name of the **.NCD** file you are running.

Configure - [C:\GCODE\2BOXES.NCD]

Any changes made in Setup are to the configuration file of the program in the Title Bar.

- When you select **Retrieve File** the file you want to view will be seen, but changes will be made to either the **default.cfg** file (if you are in Configure Defaults) or your **.NCD** file (if you are in Setup for that file).

Explanation of Configuration Parameters
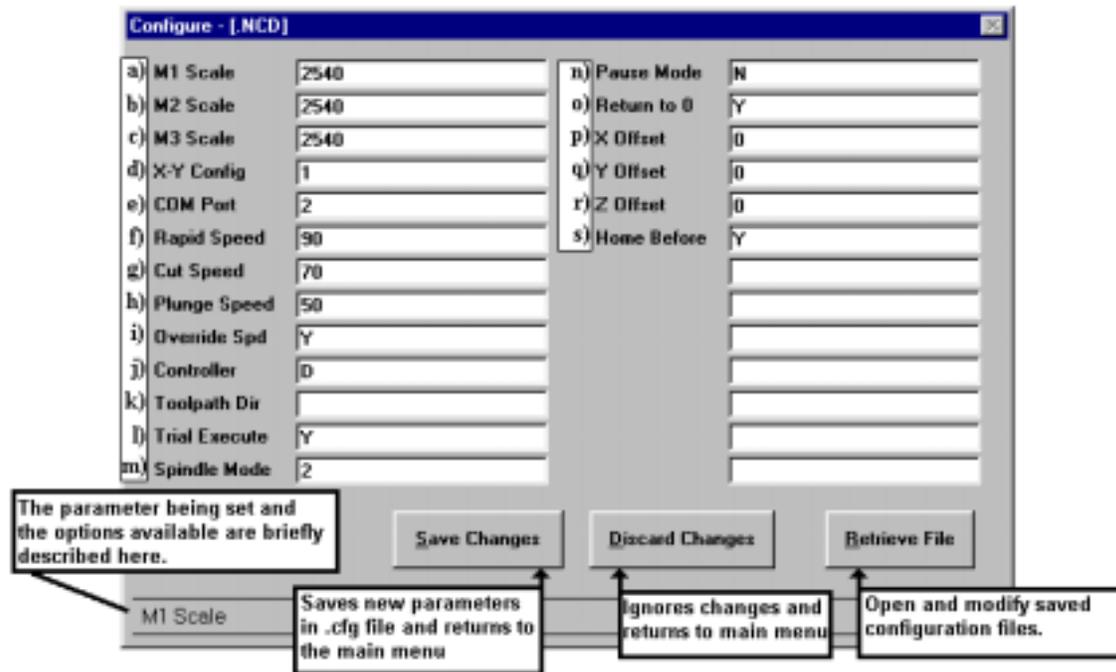
a) **M1 SCALE** is the Motor 1 scale factor.
b) **M2 SCALE** is the Motor 2 scale factor.
c) **M3 SCALE** is the Motor 3 scale factor.

SCALE FACTOR is the number of steps per unit of measurement your machine takes. Scale factors define the unit of measure you will use and are determined using the desired unit of measure, the screw pitch of the axis, and motor resolution. The table on the following page gives scale factors for common screw pitches in motors with a resolution of 400 steps/revolution.
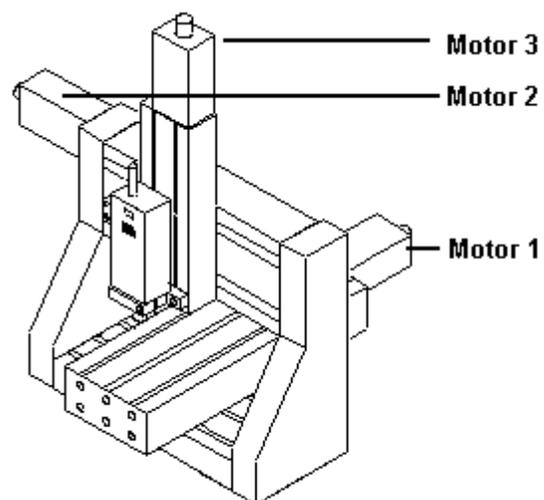
- DaVinci units are factory-equipped with 4mm screws.

Common Scale Factors

Scale Factor (steps/unit)	Screw Pitch (mm)	Measurement Unit of Drawing	
5080	2	Inch	
2540	**4**	**Inch**	**DaVinci**
2032	5	Inch	**scale**
2000	2	Centimeter	**factors**
1000	**4**	**Centimeter**	**are in**
800	5	Centimeter	**boldface.**
200	2	Millimeter	
100	**4**	**Millimeter**	
80	5	Millimeter	

- Changing the scale factor will alter speed and distance traveled by that axis. Hence, changing the Z-axis scale factor will alter its programmed depth. Unexpected machining results may occur. Make sure scale factors are properly set before attempting to produce a part.

d) **X-Y CONFIGURATION** determines orientation of the X and Y-axes. This will be set to (**+1**) or (**–1**). The following is an overhead view of both configurations.
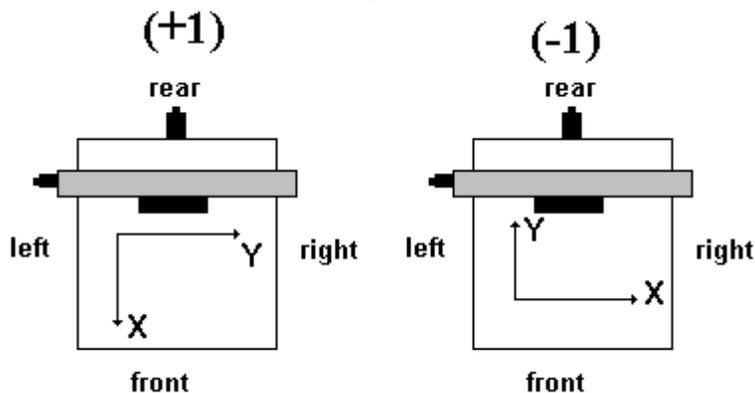
- X-Y Configuration also effects jogging. Although directional keys will move the machine in the same fashion, axes are assigned differently for each configuration.

e) **COM PORT** should be set to **1** or **2**, depending on which communications port on the back of your PC the controller is plugged into. If you are unsure about the numbering of communication ports, consult your PC manual.

f) **RAPID SPEED** is the rate (units/minute) the tool moves when not cutting. Rapid speeds should be appropriate for the screw pitch and cutting tool you are using. Maximum rapid speeds for typical screw pitches are given below.
- Rapid Speeds entered into your GCODE program will be replaced if the Interface's Override Speed feature is activated. (see following page)
- Maximum rapid speed for DaVinci machines is 140 in/min (3600 mm/min).

Screw Pitch (mm)	Max. Rapid Speed (in/min)	Max. Rapid Speed (mm/min)
2	120	3000
4	200	5000
5	200	5000

g) **CUTTING SPEED** is the rate (units/minute) the tool moves while cutting. The type of tool, the material being cut, and spindle speed determine cutting speed, and the options are too numerous to list here. Follow tool and material recommendations for appropriate cutting speeds.
- Cutting speed can also be adjusted manually during translation and execution of your program.
- Cutting speeds in your GCODE program will be replaced when the Override Speed feature is activated.

h) **PLUNGE SPEED** is the rate (units/minute) the tool moves along the z-axis (i.e. plunges into your material) when cutting. Plunge speed is also determined by the tool and material you are working with. Recommended plunge speeds are also available from your tool and materials manufacturer.
- Plunge speed may be adjusted manually during translation and execution.
- Plunge speeds entered in your GCODE program are overridden when the Interface's Override Speed function is activated.

Be sure to always use safe cutting, rapid and plunge speeds. You should not exceed speeds recommended for your controller or the speeds for which your cutting tool is rated. Consult the appropriate manuals for this information.

i) OVERRIDE SPEED tells the Interface to ignore speeds contained in your GCODE program. If the override programmed speed function is activated **(Y)**, any rapid, cutting, or plunge speeds in your program will be replaced by those contained in the configuration file created by the Interface. If this function is disabled **(N)**, your program will be executed using speeds you've entered in the GCODE file.

- Keeping the Override Speed feature activated is helpful when you are first learning operation of your system.

j) CONTROLLER asks the model of your controller.

Enter: **D**   for   **D**aVinci
           W   for   **W**izard
           4   for   C-Series **4**.0 (e.g. C10)
           5   for   C-Series **5**.0 (e.g. C142)

k) TOOLPATH DIRECTORY tells the Interface where to find your GCODE files. The format should be **drive:\folder.** e.g. **c:\gcode.**

- Double clicking in the Toolpath Directory box allows you to view and select folders.
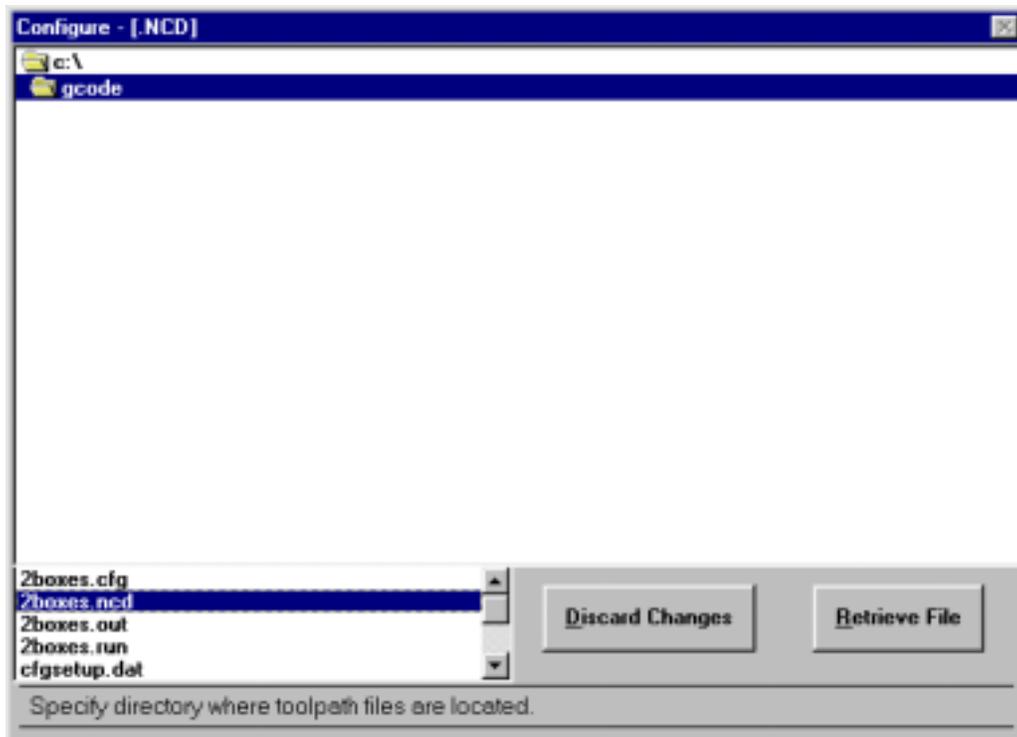
If you locate a folder you would like to designate your Toolpath Directory,

click   **Retrieve File**   and   **Save Changes**

Click   **Discard Changes**   to return to the previous window.

- All files in the selected folder can be viewed at the bottom left of this window.

l) **TRIAL EXECUTE** executes your program while it is being translated **(Y).** If Trial Execute is off **(N)**, the Interface only translates the file so that it may be easily run later.

PRACTICE LESSON: TRIAL EXECUTING A FILE

1. Click [Translate / Execute] from the main menu.

2. Select and open `2boxes.ncd`

3. Click [Setup]

- **Enter (Y)es for Trial Execute**
- **Enter (N)o for Override Speed**

4. Click [Save Changes]

   to return to Translate/Execute.

5. Click [Jog] to move the

   machine to the toolpath origin.

   and then click [Exit Jog]

6. Click [Start] from Translate/Execute. The program is simultaneously

   translated by the Interface and executed by your machine.

7. Click [OK] and [Quit] when complete.

---

1. Click [**Translate / Execute**] from the main menu.

2. **Select and open `2boxes.ncd`**

3. Click [**Setup**]

   - ■ **Enter (N)o for Trial Execute**
   - ■ **Enter (Y)es for Override Speed**
   - ■ **Set Cut Speed to 90 inches/min.**
   - ■ **Set Plunge Speed to 70 inches/min.**

4. **Click** [**Save Changes**]

5. **Click** [**Start**] **from Translate/Execute.**

   The Interface translates the program into .RUN and .OUT files (which it also does when Trial Executing) but the machine doesn't execute any movements.

6. **The translation is over in a few seconds.**

   Click [**OK**] when prompted.

7. **Click** [**Quit**] **to return to the main menu**

8. **Click** [**Run Translated Program**]

9. **Select and open `2boxes.run`.**

10. **Click** [**Jog**] **to position axes and then** [**Exit Jog**] **to return to the Run**

   Translated Program window
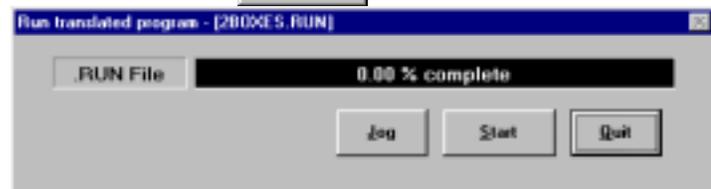
11. **Click** [**Start**] **and your machine runs the file using the new settings.**

12. **Click** [**OK**] **and** [**Quit**] **when complete.**

m) SPINDLE MODE controls when the spindle motor is activated during execution.

Enter   **0**   to leave the spindle off.

        1   to turn spindle motor on at start of program and off at the end.

        2   to activate automatic spindle mode.

- **AUTOMATIC SPINDLE MODE** turns the spindle on and off according to your GCODE program. Automatic spindle mode requires a relay option for your controller.

n) PAUSE MODE gives you the option of ignoring toolchange and other programmed pauses by entering **(N)**. Entering **(Y)** will set the controller to stop at all such pauses.

- Pause Mode should be enabled during any program that requires tool changes.

o) RETURN TO 0 returns the machine to the point last designated (0,0,0) (the toolpath origin) after it has completed running a program. **(Y)** engages this feature, **(N)** disables it.

- Return to 0 returns the machine to the toolpath origin you have assigned, not the machine's home position.

p) X-OFFSET is the distance in units (or fractions of units) which the x-axis will move away from the toolpath origin before running the program.

q) Y-OFFSET is the distance in units (or fractions of units) which the y-axis will move away from the toolpath origin before running the program.

r) Z-OFFSET, as you may have guessed, is the distance in units (or fractions of units) which the z-axis will move away from the toolpath origin before running the program.

- Offsets automate machine setup for production by sending the machine to a set position each time the program is run.
- Offsets are most useful with the Home Before Run (see bottom of page 24) function activated (enter 'Y' in Setup).
- Using Offsets without activating Home Before Run may be useful if you are running the same program several times on the same piece of material. For example, if you were cutting several parts from one block, you could enter an X-Offset large enough to have the machine move away from the cuts already completed. When you run the program again, the machine moves the distance specified away from the first part, and performs the same cuts further along the block. When attempting cuts like this, you may want to disable Return to 0 (enter 'N' in Setup).
- Offset can also be used in conjunction with the Jog function. The Practice Lesson below will show you how to employ Offset in making parts.

PRACTICE LESSON: USING OFFSETS

1. **Click** [Translate / Execute] **and open** `2boxes.ncd`**.**

2. **Click** [Setup]

   - **Activate Home Before Run by entering (Y).**
   - **Enter (+1) for the X-Y Configuration.**
   - **Enter the following Offsets:**

X Offset	1.5
Y Offset	2.5
Z Offset	-.5

3. **Click** [Save Changes] **to return to Translate/Execute.**

4. **Click** [Start] **from the** [Translate / Execute] **window.**

   - **Now, instead of jogging the machine to position axes manually, your program starts from the same position by homing the axes first and then moving the distance specified as Offsets.**

5. **Click** [Quit] **when complete to return to the main menu.**

- The following lesson illustrates a shortcut that lets you save jogged positions as X, Y, and Z Offsets in Setup, instead of having to enter them each time.

```
┌─────────────────────────────────────────────────────────────────────┐
│                                                                       │
│        PRACTICE LESSON: SAVING JOGGED POSITIONS AS OFFSETS            │
│                                                                       │
│                                                                       │
│   1.  Click   [ Translate / Execute ]    and open 2boxes.ncd.         │
│                                                                       │
│   2.  Click   [ Jog ]    from the Translate/Execute window.           │
│                                                                       │
│                               ┌──────────┐                            │
│                               │ cancel   │                            │
│                               │ Home X   │                            │
│   3.  Click   [ Home ]  and select │ Home Y   │  from the pull-down   │
│                               │ Home Z   │       menu.                │
│                               │ Home All │                            │
│                               └──────────┘                            │
│                                                                       │
│   4.  Use directional buttons to jog the machine to (1.2, 3.1, -0.7). │
│                                                                       │
│   5.  Click   [ Exit Jog ]   to return to Translate/Execute.          │
│                                                                       │
│   6.  Click   [ Setup ]                                               │
│                                                                       │
│      ▪  Enter (Y) in the Home Before Run and Return to 0 displays.    │
│   7.  Place the cursor in the X-Offset display (Y or Z will also work)│
│       and double-click.  A dialog box appears.                        │
│                                                                       │
│        ┌────────────────────────────────────────────────┐            │
│        │ Do you want to use the saved jog position?  [X] │            │
│        │                                                 │            │
│        │   1.2000 , 3.1000 , -0.7000                     │            │
│        │                                                 │            │
│        │        [   Yes   ]        [   No   ]            │            │
│        └────────────────────────────────────────────────┘            │
│                                                                       │
│   8.  Click   [ Yes ]                                                 │
│                                                                       │
│      ▪  The jogged position now appears as the XYZ Offsets.           │
│                                                                       │
│   9.  Click  [ Save Changes ]   to add these new values to the        │
│       Setup of the file.                                              │
│                                                                       │
│   10. Click   [ Start ]   from Translate/Execute.                     │
│                                                                       │
│      ▪  The program will now run as in the previous lesson, homing    │
│         the axes first, then moving the Offset distances, and         │
│         beginning the toolpath from this point.                       │
│   11. Click   [ OK ]   and   [ Quit ]   when complete.                │
│                                                                       │
└─────────────────────────────────────────────────────────────────────┘
```
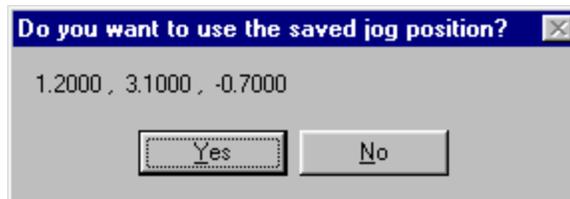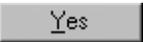
s) HOME BEFORE RUN sends the machine to the home position before starting a program. Enter **(Y)** to activate Home Before Run, **(N)** to turn it off.

   ▪ Home Before Run, in conjunction with Offsets and Return to 0, is useful in establishing a common toolpath origin. See the lesson above.

2. Jog

JOGGING manually positions the axes. The axes can be moved to any point you wish to consider your toolpath origin.

- When jogging to establish a toolpath origin, disable Home Before Run (enter 'N' in Setup).
- The previous practice lesson shows how to save jogged positions as axis offsets.

The Jog Window
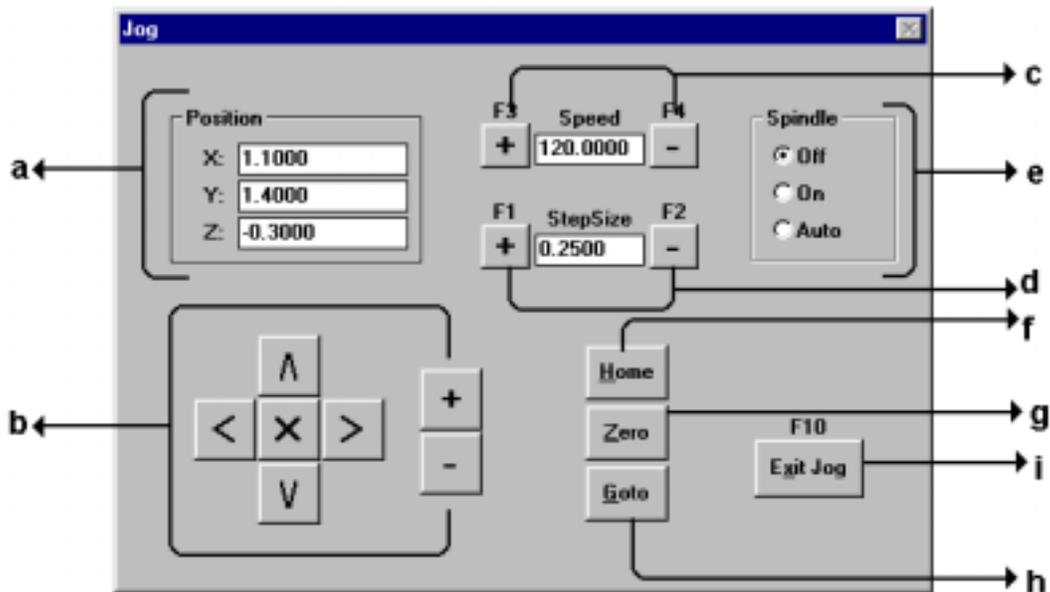
a. The **POSITION** of each axis is displayed in the upper left.

- When you   **Exit Jog**   position counters reset to (0,0,0). Thus, your toolpath origin always has these coordinates, regardless of axes' position.

b. **DIRECTIONAL KEYS** let you move axes in positive and negative directions. The axes will move the step-size and speed shown in this window. You may either click icons or use cursor directional keys (↑,↓,→,←,+,−) to move axes.

- Functions of directional keys in Jog do not change with (+1) and (-1) X-Y Configuration. Only the name of the axis that is being moved changes.
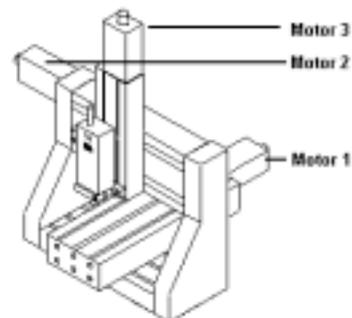
- Directional keys jog the machine as follows (face front of machine):

Using Jog Directional Keys

Icon	Motor # moved	Direction	+1 Config. Axis	-1 Config. Axis
Λ	1	Forward	X	Y
V	1	Back	X	Y
>	2	Right	Y	X
<	2	Left	Y	X
+	3	Up	Z	Z
-	3	Down	Z	Z
X	—	Stop	—	—

c. **SPEED** is the rate (units/minute) axes move during jogging. Increase speed by clicking the **(+)** icon to the left of the speed display or pressing the **F3** key. Decrease speed by clicking the **(–)** icon to the right of the display or pressing **F4**.

- You can also enter specific speeds by double clicking in the speed display bar. The dialog box below will appear:

Enter Speed

Enter Speed in units/minute

OK

Cancel

90.0000
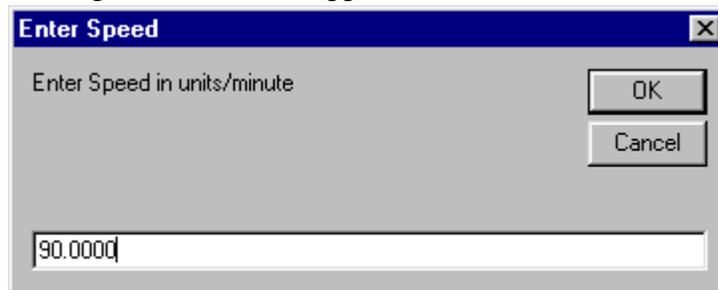
Enter the desired speed in units/minute and click [OK]

- The Interface prevents exceeding speeds allowed by the machine. When you have reached a speed limit, the display will freeze. If you attempt to enter an inappropriate specific value, the maximum (or minimum) speed is assigned.

d. **STEP-SIZE** is the distance in units (or fractions of units) the machine moves along its axes at the speed displayed each time you press one of the directional keys. Pressing **F1** or clicking **(+)** increases step-size, **F2** or **(–)** reduces step-size. Each mouse click or keystroke moves your machine one step.

- Step-size can be set to specific values in the same manner as Speed. Double click on the display:

Enter Jogging Step-Size

Enter Jogging Step-Size in units

OK

Cancel

0.1000
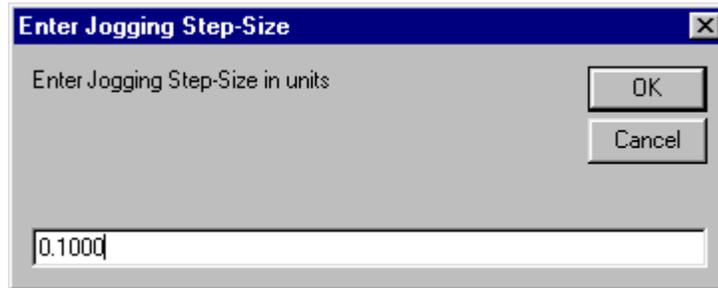
  Enter your value in units and click   OK

- Start with a small step-size since a machine error occurs each time an axis moves past its limit switch.
- The Interface locks out step-sizes inappropriate for your machine. Again, if you exceed these values the Interface will automatically assign the maximum or minimum step-size possible.

e. SPINDLE MODE can also be set during Jog mode by clicking radio buttons.

Spindle
- Off
- On
- Auto

  Functions are the same as in Configure Defaults (page 21).

- Adjusting the spindle mode from the Jog screen will not change this parameter in the configuration file of your program.

f. HOME sends specified axis to the machine's home position.

cancel
Home X
Home Y
Home Z
Home All

When you click   **Home**   a pull-down menu
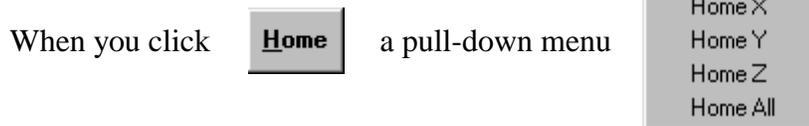
  asks which axis you want to home. You may choose any or all.

- When you have homed the axes, avoid directional keys that will push the machine past its limit switch.

g. ZERO sets position counters back to 0, but **does not** move the machine.

Cancel
Zero X
Zero Y
Zero Z
Zero All

When you click   **Zero**   a pull-down menu
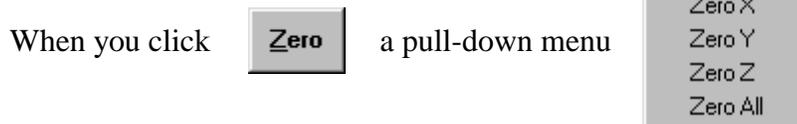
  asks which axis you want to zero. You may choose any or all.

- Zero All is automatically performed every time you exit Jog.

- Remember that this feature only resets the position counters and all subsequent movements by the machine will still be in reference to the same position, although they will be assigned new coordinates.

h. **GOTO** sends the machine to specific coordinates in reference to the machine's current position.

Clicking [**Goto**] offers the option [Cancel / Enter Position / 0, 0, 0] of going to the point last designated (0,0,0) or entering specific coordinates (remember that these will be relative to current axis position). When you select Enter Position, position counters will flash and you'll be prompted to enter coordinates:
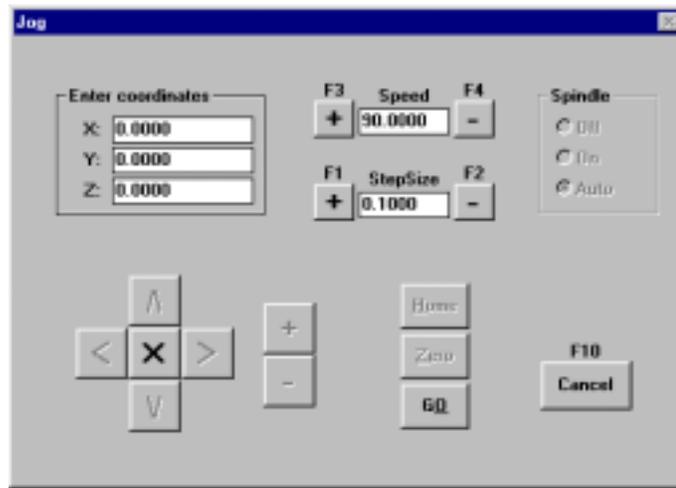
Enter coordinates and click [**GO**] to send the machine to this position.

Click [**Cancel**] to return to the Jog window.

i. **EXIT JOG** resets the position counters (i.e. Zeroes All) and returns to the previous menu.

3. Translate / Execute

Translate / Execute   converts GCODE programs into files that can be understood by Techno DaVinci and C-Series controllers, enabling your machine to move along the translated toolpath and produce a part. When you Translate/Execute a file, the Interface writes **.RUN** files that can be run later and **.OUT** files that can be downloaded and run directly from your controller.
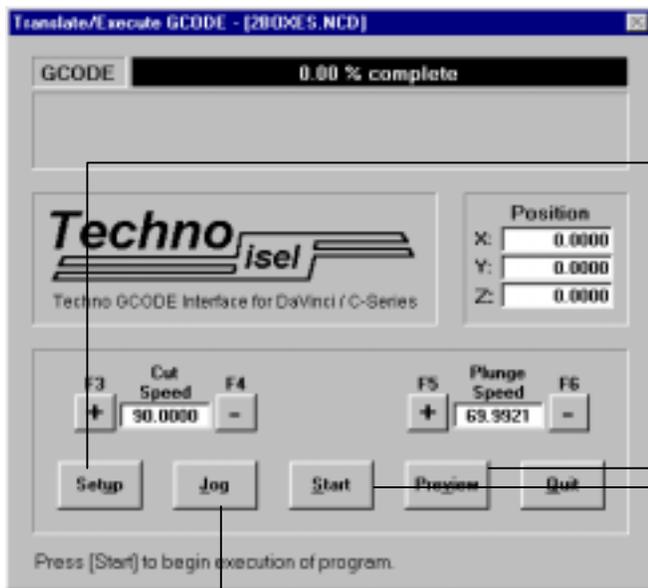
Set system parameters for your **.ncd** file.
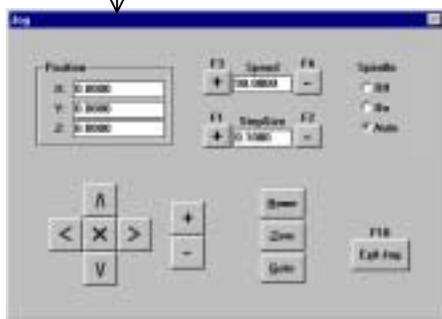
Jog the machine to your toolpath origin.
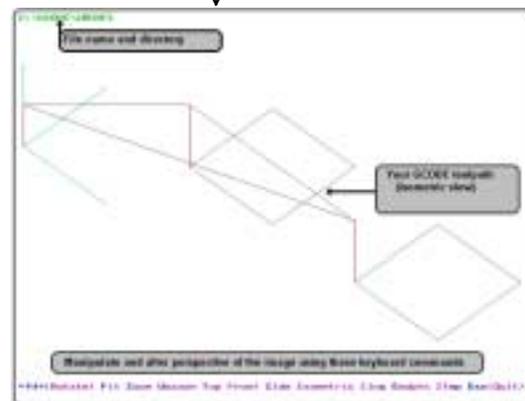
Preview the toolpath before running a program.

Translate and/or execute your program.

Translating / Executing a GCODE file:

1. Click [**Translate / Execute**]

2. An Open File dialog box appears. The toolpath directory shown
   will be the one specified in Configure Defaults.

Open   ? ✕

File name:
*.ncd

2boxes.ncd

List files of type:
*.NCD ▾

Folders:
c:\gcode

📁 c:\
📁 gcode

Drives:
💾 c: ▾

[**OK**]
[**Cancel**]
[**Network...**]
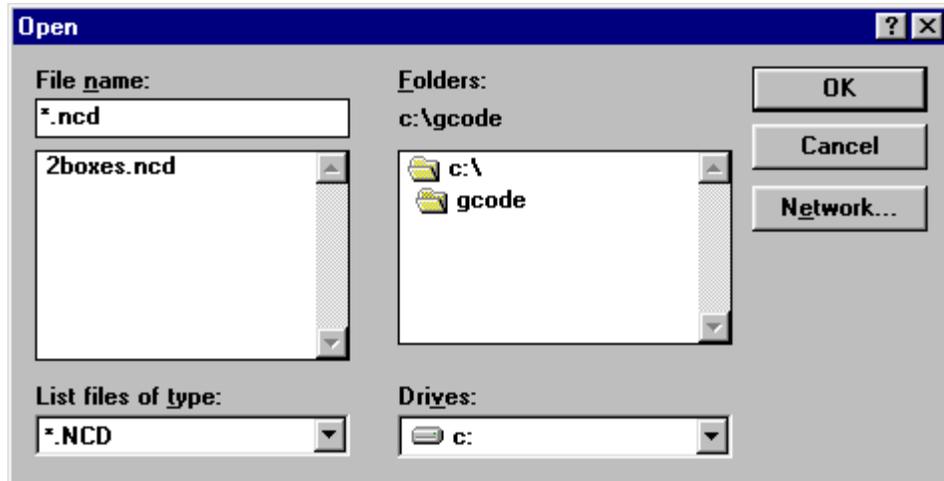
    Browse folders by double clicking on them.

3. When you've located your file, select it and click [OK]

4. Click [**Setup**] and then [**Save Changes**]
   if you've adjusted any parameters.

5. Click [**Jog**] to position the machine manually if necessary.

6. Click [**Start**] from Translate/Execute.

7. When complete, click [OK] then [**Quit**]

Translate/Execute

a. The name of your **GCODE file** appears in the Title Bar.

b. **PROGRAM COMMAND DISPLAY** displays GCODE commands currently executing.

```
G1 Z-.25 F20.0
Y2 F40.0
X2
```
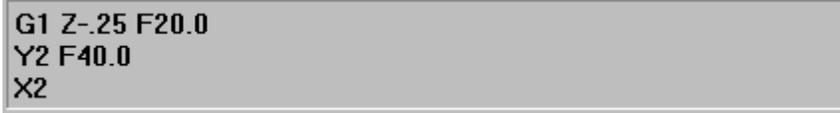
- This display helps spot errors in your GCODE program.

c. **CUTTING SPEED** can be manually adjusted while translating/executing your program, independent of cutting speeds you may have entered in Setup.
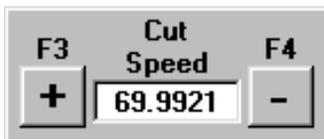
 click or **F3** to increase. or **F4** to decrease.

- Altering cutting speed while running a program may affect production of your part. This may be helpful, however, in determining ideal cutting speed.

d. SET<u>U</u>P opens the configuration file of your program. Setup functions are the same as Configure Defaults (section 1 of this chapter) **except** they are applied only to the program you are running. From Setup, you can adjust any parameter except Controller and COM Port, which must be changed from Configure Defaults.

Clicking [**S̲ave Changes**] in [**Set̲up**] will add new values to the **.CFG** file you are running.

- You may also retrieve saved configuration files in Setup. Remember though that any changes made in Setup will be to the file you are translating and/or executing.

- When you are running a file for the first time, or if a configuration (**.CFG**) file does not exist, the Interface reminds you to check the Setup and that it will create a **.CFG** file for your program using system defaults until changes are made.

> **Techno GCODE Interface v 1.18**   ✕
>
> A new 2BOXES.CFG file was created using default settings.
> Be sure to check the SETUP before running the file.
>
> [OK]

e. J̲OG accesses the Jog window described in the previous section of this chapter. All keys, icons, and functions are identical.

- Remember that speeds set in Jog only apply while jogging.

f. The **TASK COMPLETION DISPLAY BAR** displays the percentage of your program completed.

> [GCODE]   38.61 % complete

g. POSITION DISPLAY shows position of each axis during execution.

> **Position**
> X: 2.0000
> Y: 2.0000
> Z: -0.2500
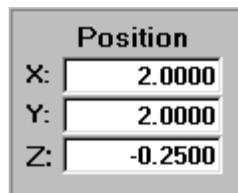
h. PLUNGE SPEED, like cutting speed, can be adjusted manually during translation/ execution.

> F5 **Plunge Speed** F6
> [+] 50.0079 [–]

click [**+**] or **F5** to increase.   [**–**] or **F6** to decrease.
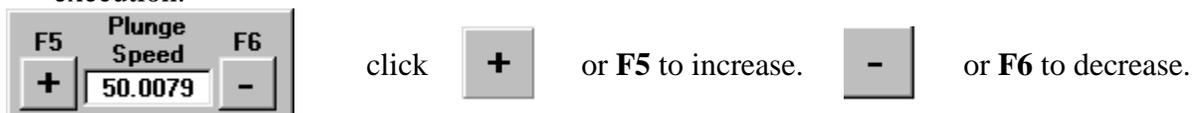
- Altering plunge speed while your program is running may effect production of your part. This may be helpful, however, in determining ideal plunge speed.

i. **<u>S</u>TART** tells the Interface to begin translating and/or executing your program.

- If Trial Execute is disabled (N) in [Set<u>u</u>p] your machine will not execute the File. The Interface will write a file with a **.RUN** extension that can be run later from the [<u>R</u>un Translated Program] window (page 36). A stand-alone program (.**OUT** extension) is also written. This file can be downloaded and run directly from your controller from [Do<u>w</u>nload] (page 38).

- If Trial Execute is enabled (Y), your program will be translated and executed simultaneously. While your program is Translating/Executing, the window appears as below:

Translate/Execute GCODE - [2BOXES.NCD] ⊠

| GCODE | 43.67 % complete |

X2
Y1
X1

Techno isel

Techno GCODE Interface for DaVinci / C-Series

Position
X: 1.0000
Y: 1.0000
Z: -0.2500

| F3 | Cut Speed | F4 | | F5 | Plunge Speed | F6 |
| + | 69.9921 | − | | + | 50.0079 | − |

[<u>P</u>ause] [<u>J</u>og] [<u>S</u>tart] [<u>R</u>eturn] [<u>H</u>alt]

Program is executing.
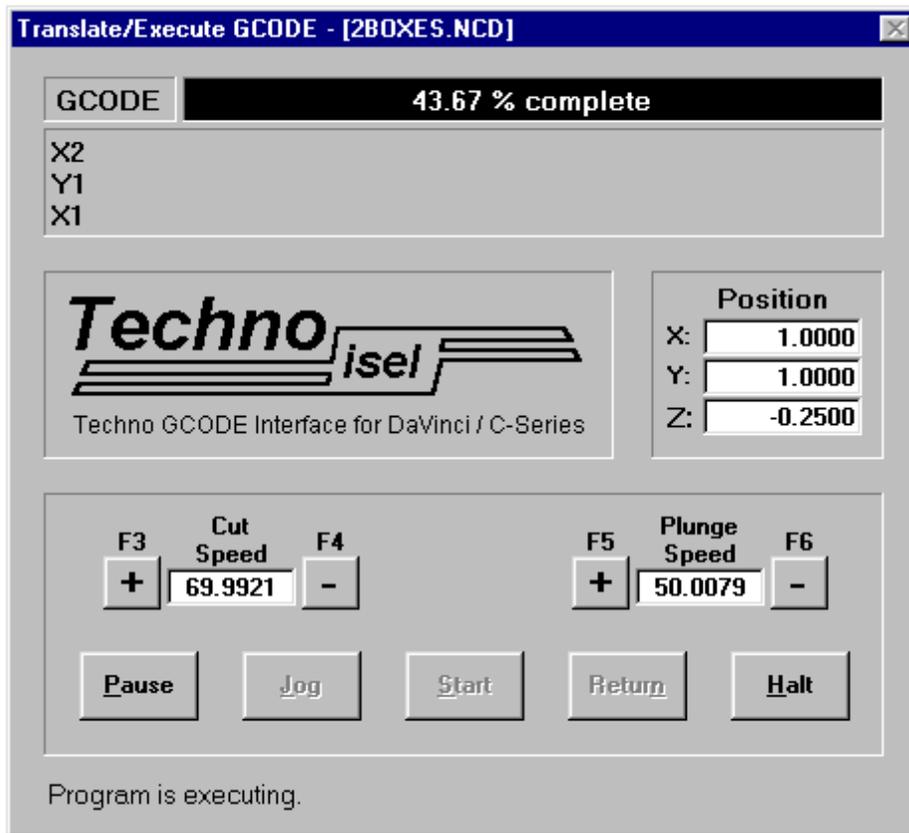
- While your program is translating/ executing, Pause and Halt options become available.

[<u>H</u>alt] Aborts the execution of your program. Click [OK] to return to [<u>T</u>ranslate / Execute]

Pause   freezes the program and offers the following options:

1 Step   Executes the next step of your program and returns to Paused status.

Resume   Continues execution of your program.

Return   Returns the machine to the toolpath origin.

Quit   Aborts execution and returns to main menu.

j. PREVIEW displays the toolpath before translation and execution of your file. The Preview feature is detailed in section 6 of this chapter.

k. QUIT exits Translate/Execute and returns to main menu.

4. Run Translated Program

Run Translated Program executes previously translated GCODE files.
These files have already been written by the Interface and have a **.RUN** extension.

  ▪ Setup changes cannot be made from the Run Translated Program window.
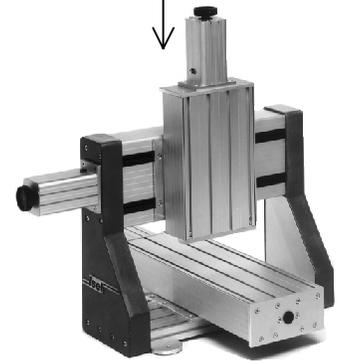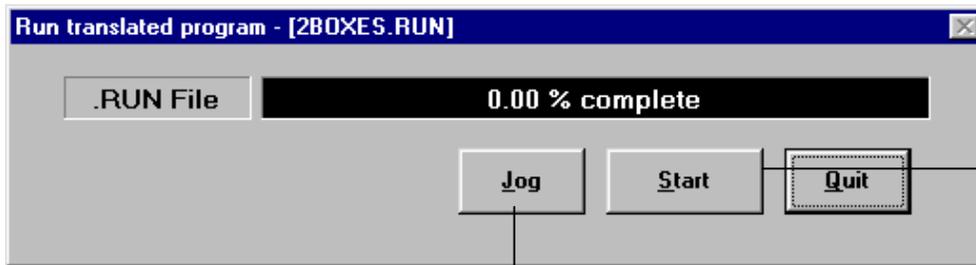
Running a Translated Program:

1. Click **Run Translated Program** from the main menu.
2. An Open File dialog box shows your toolpath directory and all **.RUN** files.
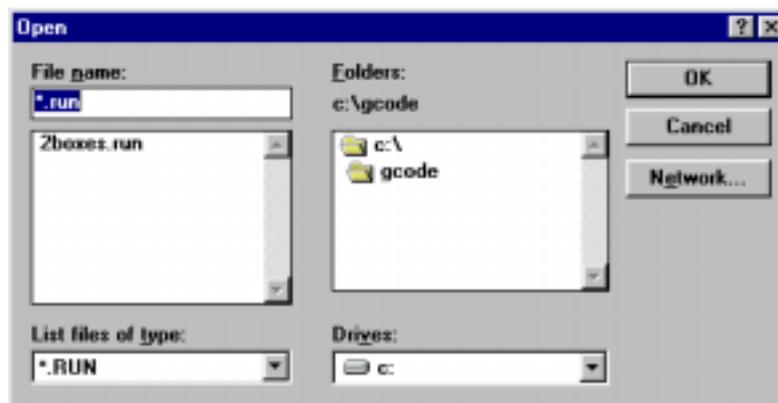
3. Select the file you want to run and click **OK**

- Folders can be browsed by double clicking on them.

4. The **Run Translated Program** window opens.

Run translated program - [2BOXES.RUN]	⊠
.RUN File	0.00 % complete

Jog   Start   Quit
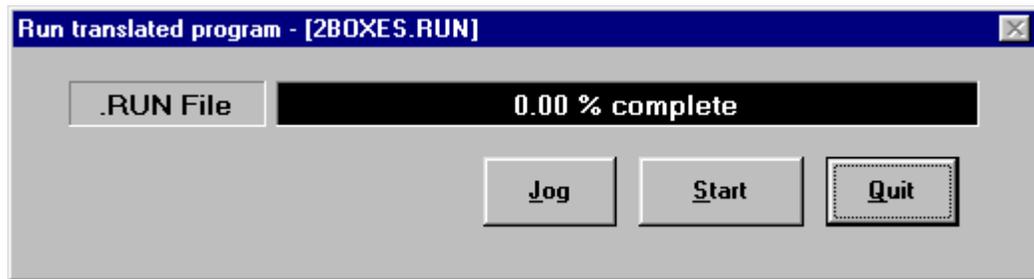
5. Click **Jog** if necessary to manually position axes at your toolpath origin.

6. Click **Exit Jog** to return to Run Translated Program.

7. Click **Start** to begin execution.

8. Click **Quit** when your program has completed.

- The **Pause** and **Halt** options available in Translate/Execute

  are also available in Run Translated Program.

- Using Offsets rather than Jogging is helpful in Run Translated Program since Setup cannot be accessed.

5. Download

 sends standalone programs written by the Interface (files with
`.OUT` extensions) to your controller's memory. Once the file is downloaded, it can be run
directly from your controller by simply pressing Start on the machine's display panel.
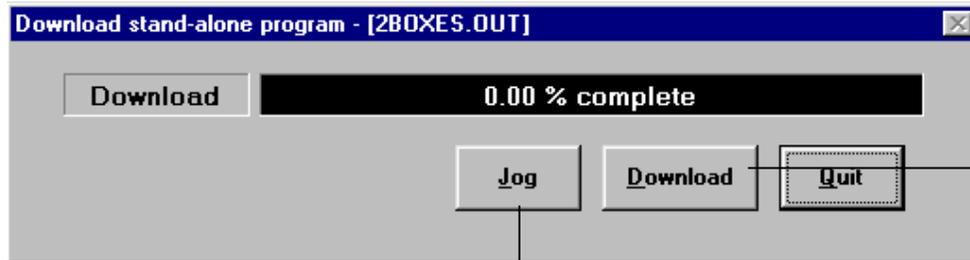
- Your controller can only run the most recently downloaded program. Once a new
  program has been downloaded, or if the controller's power has been turned off,
  any previously downloaded program is erased and must be downloaded again.
  This process takes only a few seconds.
- Standalone files are created for every program translated.

Downloading and Running a Standalone Program

1. Click from the main menu.

2. An Open File dialog box opens listing all `.OUT` files.
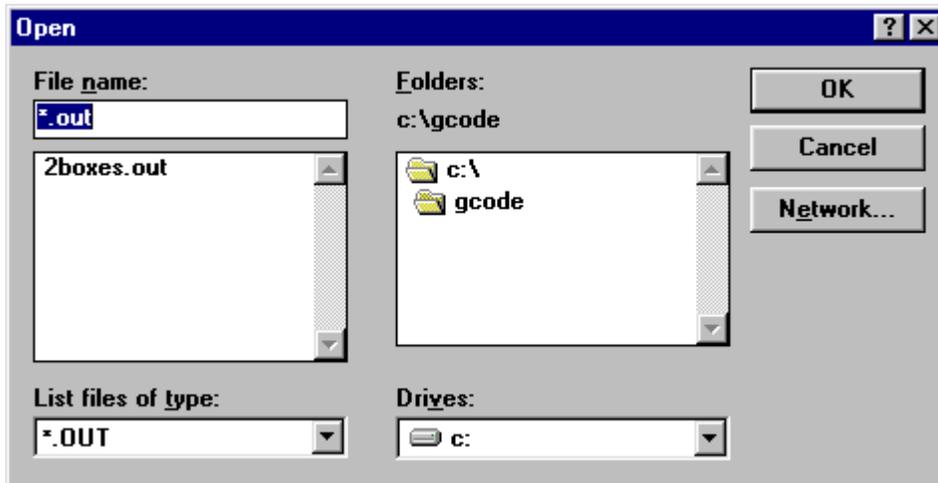
3. Select the standalone file you want to run from your controller and click [OK]

4. The [**Download**] window opens:
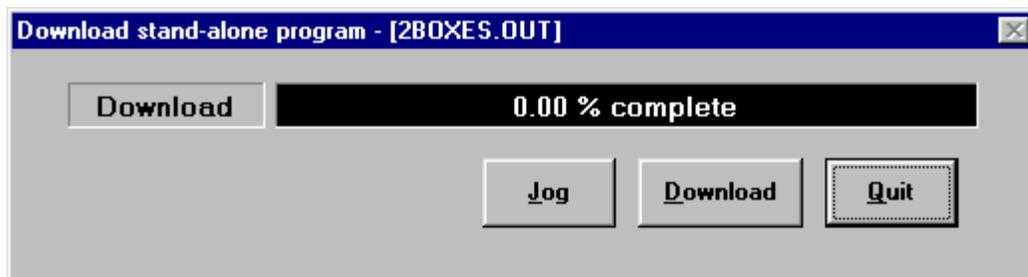
5. Click [**Jog**] if necessary to manually position to your toolpath origin.

6. Click [**Exit Jog**] to return to Download.

7. Click [**Download**] to send the file to your controller.

   - During Downloading, Pause and Halt features are available and the Task Completion Display shows the percentage of your file that has been sent.

8. Press the Start button [⎜I⎜] on your controller to execute the program once.

   - You can execute the same file as many times as you wish once it is downloaded. Just press the Start button. Turning controller power off erases the file from the controller's memory.
   - When setting up standalone files, it is helpful to use Offsets in conjunction with Return to 0 and Home Before Run. This ensures a consistent toolpath origin.

6. Preview

| Preview | displays your toolpath prior to translation and/or execution of your file.

- Preview cannot be run in a window. Your computer may issue a warning. If the warning occurs, press any key to enter full screen mode. When you exit Preview, you will automatically revert to window mode.

Previewing Your GCODE Toolpath:

1. Click | Preview | from the main menu.

2. An Open File dialog box appears.

Open ? X

File name:
*.ncd

2boxes.ncd

List files of type:
*.NCD

Folders:
c:\gcode

c:\
gcode

Drives:
c:

OK
Cancel
Network...
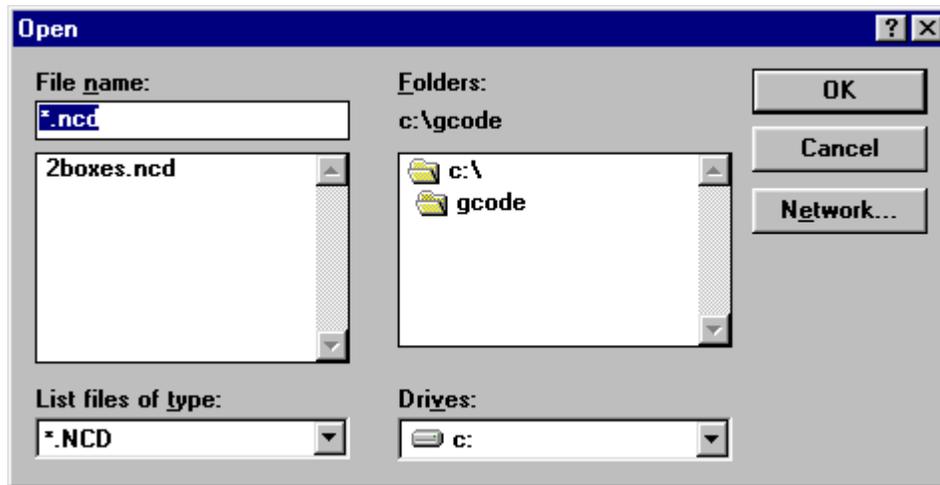
3. Select the file you'd like to Preview and click | OK |
4. The Preview of your file appears. Manipulate the image and view it from different perspectives using keyboard commands.
5. Press the **Esc** key to return to the main menu.

Preview of 2boxes.ncd
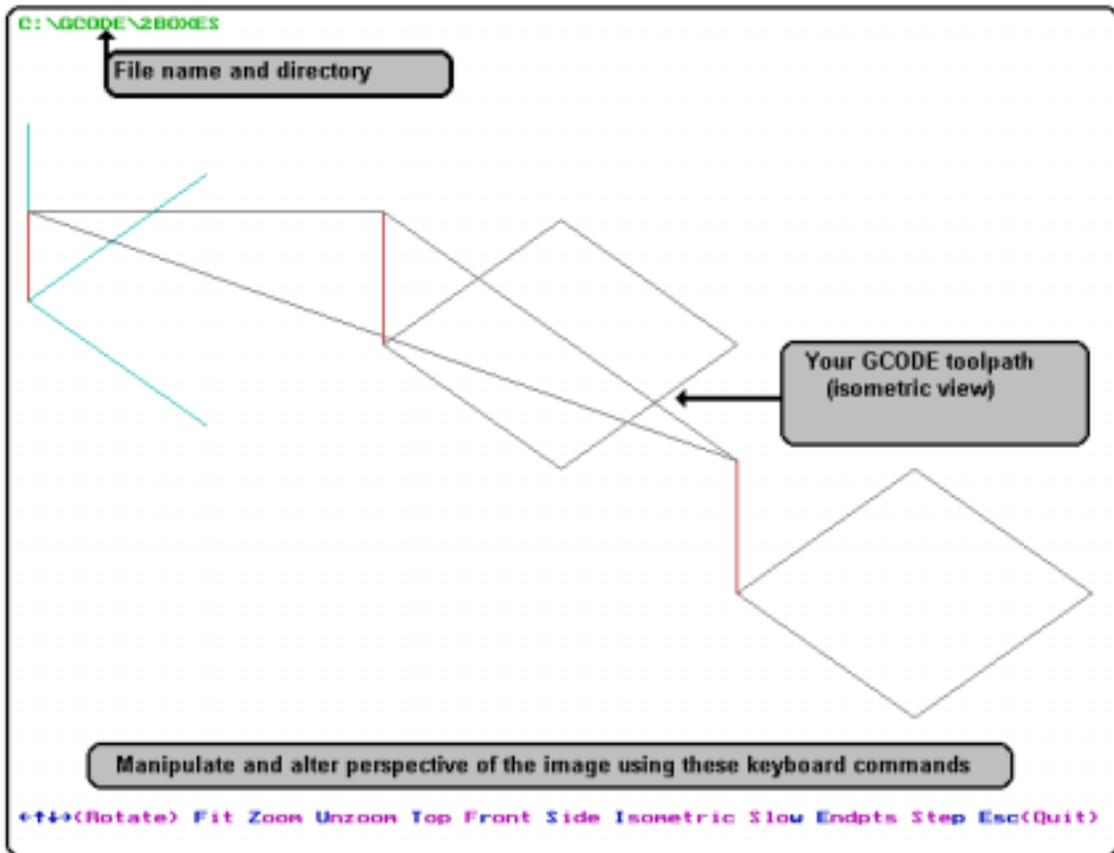
Using Preview Keyboard Commands
(Highlighted keys perform indicated function)

Function & Key	Description
←↑↓→(Rotate)	Rotates image around axis indicated by arrow.
Fit	Fits image to the size of your screen.
Zoom	Zoom in on part of your design. Press **Z,** place the cursor where you want to zoom, and click.
Unzoom	Zooms out from where you have just focused.
Top	Switches to top angle view of your design.
Front	Switches to front view of your design.
Side	Switches to side view of your design.
Isometric	Switches to isometric view (shown above) of your design.
Slow	Slows down movement of image.
Endpts	Highlights endpoints of each command.
Step	Shows program being executed line by line. Commands being shown are displayed during step viewing.
Esc(Quit)	Returns to main (or previous) menu.

IV. GCODE Command Summary by category

SET UP COMMANDS

CODE	COMMAND	FORMAT	PURPOSE	PAGE #
F	**Feed Speed**	Fn	Designates feed rate, or rate of movement, of the axes.	45
G4	**Dwell Time**	G4/d	Specifies a programmed delay during a drill cycle.	49
G90	**Absolute Coordinates**	G90	Indicates that *absolute coordinates* will be used to perform subsequent motion commands.	56
G91	**Incremental Coordinates**	G91	Indicates that *incremental coordinates* will be used to perform subsequent motion commands.	57
G92	**Set Absolute Position**	G92XxYyZzAa	Sets new absolute coordinates for the current position.	58
M0	**Program Pause**	M0	Pauses the .NC file run.	59
M30	**End Of Data**	M30	Designates the end of a block of commands in a file.	63
M3 M4	**Spindle On**	M3 *or* M4	Starts the spindle.	60
M5	**Spindle Off**	M5	Turns the spindle off.	60
M6	**Tool Change**	M6T#	Allows the tool to be changed by turning off the spindle and coolant and pausing the program.	61
M7 M8	**Coolant On**	M7 *or* M8	Turns the coolant on.	62
M9	**Coolant Off**	M9	Turns the coolant off.	62
M90	**Output Off**	M90 OUT#	Turns off the designated output.	63
M91	**Output On**	M91 OUT#	Turns on the designated output.	64

ROUTING COMMANDS

CODE	COMMAND	FORMAT	PURPOSE	PAGE#
G0	**Rapid Move**	G0XxYyZzAa	Moves one or more of the axes, at the rapid speed, to a specified location.	45
G1	**Linear Cutting Move**	G1XxYyZzAa	Moves one or more of the axes along a straight line, at the cutting speed, to a specified location.	46
G2	**Clockwise Arc**	G2XxYyIiJi *or* G2XxZzIiKi *or* G2YyZzJiKi	Moves two of the axes, at the cutting speed, along an arc in a clockwise direction to a specified location.	47
G3	**Counterclockwise Arc**	G3XxYyIiJi *or* G3XxZzIiKi *or* G3YyZzJiKi	Moves two of the axes, at the cutting speed, along an arc in a counterclockwise direction to a specified location.	48

DRILL CYCLE COMMANDS

CODE	COMMAND	FORMAT	PURPOSE	PAGE #
G80	**Drilling Cycle Off**	G80	Turns a drill cycle off.	49
G81	**Standard Drilling Cycle Without Dwell**	G81XxYyZzRrPpFf	Provides a feed-in, rapid-out sequence used for standard drilling without a dwell time.	50
G82	**Standard Drilling Cycle With Dwell**	G82XxYyZzRrPpFf	Provides a feed-in, rapid-out sequence used for standard drilling with a specified dwell time.	51
G83	**Peck Drilling Cycle**	G83XxYyZzRrQqVvPpFf	Provides a series of feed-in, peck drilling motions with full retract used for drilling holes.	53
G87	**Chip Break Drilling Cycle**	G87XxYyZzRrQqVvWwPpFf	Provides a series of feed-in, rapid-out motions that is similar to peck drilling, but the retract is a specified distance.	55

GENERAL DRILL CYCLE COMMAND FORMAT

G8n Xx Yy Zz Rr Qq Vv Ww Pp Ff

X	the x coordinate of the hole
Y	the y coordinate of the hole
Z	the z coordinate of the hole
R	the reference height for start of drill plunge
Q	the initial z peck increment
V	the subsequent z peck increment.
W	the peck clearance, z retract between pecks
P	the dwell time (seconds)
F	the drilling federate, plunge speed
n	an integer in the range of zero to nine

G-Code And The Interface Program

The Electronic Industries Association developed a standard for a code it defines as an "interchangeable variable block data format for positioning, contouring, and contouring/positioning numerically controlled machines". The standard for this general machine code, known as G/M-Code or, more commonly, G-Code, is EIA-274-D.

The Techno G-Code CNC Interface is designed to recognize some of the standard G/M codes. The codes it supports can be used to revise an existing G-Code (.NC) file or to create an original file that can be run by the Techno G-Code Interface. The codes that are recognized by the Techno Interface are listed and described in detail in this chapter. And at the end, a sample G-Code file is written out for you to look over. For more comprehensive information about General Machine Code, or for a complete listing of all G/M codes, refer to a G-Code language manual.

> ☞ *In standard G-Code command definitions, the word "program" refers to the G-Code program, also known as the .NC file. Commands such as* **Program Stop** *or* **End Of Program** *refer to stopping or ending the .NC file, not the interface program.*

Each G/M code recognized by the interface program is described in this chapter using the format shown below.

Code Number   Command Name

Format   Presents the syntax of the command for a specific code. The *format* illustrates the way the command is written in the program. In many cases, the command will involve variables, which will be indicated by symbols that are defined in the text.

Purpose   Explains how the command is used and what it is used for.

Example   A sample set of commands for the code is presented and explained.

Commands Recognized By The Techno Interface

F   Feed Speed

Format   **Ff**

f = rate in units/minute

Purpose   Designates the rate of movement of the axes. This command can be used to set the feed rate for cutting moves, the plunge rate for drilling cycles, and the rapid speed for rapid moves. It is ignored if **Override Program Speed** is selected in the Techno CNC G-Code Interface.

Example   **G1X5.00Y4.00Z-2.00F2.00**

The linear cutting motion indicated above will be performed at a rate of two units per minute when **Override Program Speed** has not been selected in the interface program. If the unit selected in the interface program is inches, then the cutting motion will be performed at a rate of two inches per minute.

G0   Rapid Move

Format   **G0XxYyZzAa**

X,Y,Z,A = axis identifiers
x, y, z, a = distance in incremental units, or absolute coordinates, to which the axes specified by the preceding upper case letter is designated to move

Purpose   Moves one or more of the axes, at the rapid speed, to a specified location or a specified distance away from the current position. Location is designated by absolute coordinates when this command is preceded by the *Absolute Coordinate* command. Distance is specified by incremental units when this command is preceded by the *Incremental Coordinate* command.

A rapid move consists of a move of the X and/or Y axis along the XY plane and a move of the Z axis up or down. If the Z-axis move is Directed from a lower point to a higher point, it will be executed before the XY move; if it is directed from a higher point to a lower point, it will be executed after the XY move.

Rapid speed is indicated in the .NC file by the *Feed Speed* command. F commands are ignored by the interface program when **Override Program Speed** is selected. When this selection is made, the rapid speed specified in Set-Up will be used instead.

Example **G90**
G0X2.00Y2.00Z2.00

The first command in this set indicates absolute motion, so the rapid move that follows (line 2) is made to the absolute coordinates (2,2,2).The axes move to a position that is two units away on each axis from the software home.

G1 Linear Cutting Move

Format **G1XxYyZzAa**

X,Y,Z,A = axis identifiers
x, y, z, a = distance in incremental units, or absolute coordinates, to which the axes specified by the preceding upper case letter is designated to move

Purpose Moves one or more of the axes, at the cutting or feed speed, along a straight line to a specified position or a specified distance away from the current position. Location is designated by absolute coordinates when this command is preceded by the *Absolute Coordinate* command. Distance is specified by incremental units when this command is preceded by the *Incremental Coordinate* command.

Example **G91**
G1X-3.50Y-4.00Z1.00
M30

The first command in this set indicates that the cutting command in line 2 will be performed in incremental motion. In line 3, the axes will move at the cutting speed as follows: the X axis will move 3.5 units away in a negative direction from its previous position, the Y axis will move 4 units away in a positive direction from its previous position, and the Z axis will move 1 unit away in a positive direction from its previous position.

G2   Clockwise Arc

Format   **G2XxYyIiJi**
                          or
                    G2XxZzIiKi
                          or
                    G2YyZzJiKi

X,Y,Z = axes on a three axis machine
x, y, z = distance in incremental units, or absolute coordinates, to which the axes specified by the preceding upper case letter is designated to move
I = X axis center point position
J = Y axis center point position
K = Z axis center point position
i = an incremental value representing the distance in units to the center position on the axis specified by the preceding letter

Purpose   Moves two of the axes, at the cutting speed, along an arc in a clockwise direction, to a specified location or a specified distance away from the current position. The arc is defined by the movement of the two axes around a designated center point, from the current position to a specified end point. The two axes used will determine the plane in which the axes will move along the arc.

If the Clockwise Arc command is preceded by the *Absolute Coordinate* command, the X,Y,Z values will be absolute. If it is preceded by the *Incremental Coordinate* command, these values will be incremental. Whether absolute or incremental, the X,Y,Z values represent axis positions marking the end point of the arc. Center point values are always incremental.

Example   **G92X3.00Y3.00Z3.00**
                    G91
                    G2X5.00Y-1.00I3.00J2.00
                    M30

The first command in this set establishes the current position as 3 units away on each axis in a positive direction from the software home. The line 2 command specifies that the axes move in incremental movement.

In line 3, the axes will move along an arc in a clockwise direction in the XY plane. The X and Y axes will move along the arc from the current position (3,3) to the end point, which is designated to be 5 units away in a positive direction along the X axis and 1 unit away in a negative direction along the Y axis from the current position. The end point is position (8,2). The X and Y axes will move clockwise around a center point that is 3 units away in a positive direction along the X axis and 2 units away in a positive direction along the Y axis from the current position. The center point position is (6,5).

47

G3   Counterclockwise Arc

Format   **G3XxYyIiJi**
                           or
                    G3XxZzIiKi
                           or
                    G3YyZzJiKi

                    X,Y,Z = axes on a three axis machine
                    x, y, z = distance in incremental units, or absolute coordinates, to which the
                           axes specified by the preceding upper case letter is designated to move

                    I = X axis center point position
                    J = Y axis center point position
                    K = Z axis center point position
                    i = an incremental value representing the distance in units to the center
                           position on the axis specified by the preceding letter

Purpose   Moves two of the axes, at the cutting speed, along an arc in a counterclockwise
                    direction, to a specified location or a specified distance away from the current
                    position. The arc is defined by the movement of the two axes around a
                    designated center point, from the current position to a specified end point. The
                    two axes used will determine the plane in which the axes will move along the arc.

                    If the Counterclockwise Arc command is preceded by the *Absolute Coordinate*
                    command, the X,Y,Z values will be absolute. If it is preceded by the
                    Incremental Coordinate command, these values will be incremental. Whether
                    absolute or incremental, the X,Y,Z values represent axis positions marking the
                    end point of the arc. Center point values are always incremental.

Example   **G92X3.00Y3.00Z3.00**
                    G90
                    G3Y9.00Z15.00J3.00K6.00
                    M30

                    The first command in this set establishes the current position as 3 units away
                    on each axis in a positive direction from the software home. The line 2
                    command specifies that the axes move in absolute movement. In line 3, the
                    axes move along an arc in a counterclockwise direction in the YZ plane. The Y
                    and Z axes move along an arc from the current position (3,3,3) to the end point.
                    The end point is designated to be 9 units away in a positive direction along the
                    Y axis and 15 units away in a positive direction along the Z axis from the
                    software home (0,0,0). The end point is position (3,9,15). Since center point
                    values are always incremental, the Y and Z axes will move counterclockwise
                    around a center point that is 3 units away in a positive direction along the Y
                    axis and 6 units away in a positive direction along the Z axis from the current
                    position. The center point position is (3,6,9).

48

G4   Dwell Time

Format   **G4/d**
       or
     G4 d

     d = duration of the timed delay in seconds

Purpose   Specifies a programmed delay, indicated in seconds. Note that a forward slash (/) or blank space must follow G4 in this command.

Example   **G0X2Y2Z2**
     G4/10
     G0X8.00 Y6.00 Z5.00

     In this example, the two rapid moves will be separated by a 10 second delay.

G80   Drilling Cycle Off

Format   **G80**

Purpose   Turns off a drill cycle. This command should follow immediately after the last drilling cycle command.

Example   **G90**
     G0X0Y0Z0
     G81X1.00Y2.00Z-1
     X4.00Y5.00
     X7.00Y8.00
     G80

     In the command set above, the first command designates subsequent motion commands to be in absolute motion. The standard drill commands in lines 3, 4 and 5 create a series of three holes. (Notice that the drill command code is written only once, in line 3, although it applies to lines 4 and 5 as well.) After the third hole is drilled, the drill cycle is cancelled.

G81   Standard Drilling Cycle Without Dwell

Format
    G81XxYyZz

    X,Y,Z =   axes on a three axis machine
    x, y, z =   distance in incremental units, or absolute coordinates, to which the axes specified by the preceding upper case letter is designated to move

Purpose
    Provides a feed-in, rapid-out sequence used for drilling a series of holes. Standard drilling involves a single plunge of the tool at the plunge speed, and a single retraction of the tool to the initial Z position at the rapid speed. The initial Z position is the position of the Z axis before the drilling cycle starts.

    The speed at which the Z axis descends into the material will either be the feed rate specified in the .NC file by the *Feed Speed* command or the plunge speed specified in Set-Up. The interface program will only recognize the *Feed Speed* command if **Override Program Speed** is not selected in Set-Up.

    In drilling, the command code usually appears on the first line of the cycle only, although there may be several holes drilled in the cycle. Also, the Z axis value is always unsigned (i.e., there is no positive or negative sign in front of it).

Example
    G90
    G81X4.00Y6.00Z-1
    X6.00
    X8.00
    G80
    G0X10.00Y10.00Z0.00

    Line 1 indicates that subsequent motion commands will be in absolute motion. In the standard drilling cycle initiated in line 2, the X axis moves 4 units in a positive direction away from the software home and the Y axis moves 6 units in a positive direction away from the software home. The X and Y axes move at the cutting speed set in the interface program. A hole is drilled at the new location (4,6). To drill the hole, the Z axis moves 1 unit in a negative direction away from its current (initial) position and then retracts to its initial position. The Z axis moves at the plunge speed set in the interface program.

    Two other holes of the same depth are drilled, one at the (6,6) position and one at the (8,6) position. The drill cycle is ended in line 5, and a rapid move is executed in line 6. This command is executed in absolute motion, according to the *Absolute Coordinate* command in line 1.

Illustration
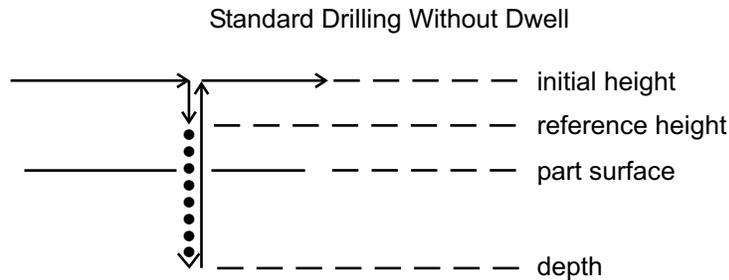
Standard Drilling Without Dwell

	initial height
	reference height
	part surface
	depth

Key

●●●●●●●●●●●	drill motion at plunge speed
――――――――	rapid motion at rapid speed
initial height	height at which rapid motions between drill holes takes place
reference height	height at which drill is moved at rapid speed before drilling takes place

G82   Standard Drilling Cycle With Dwell

Format   **G4/d**
                G82XxYyZz

X,Y,Z = axes on a three axis machine
x, y, z = distance in incremental units, or absolute coordinates, to which the axes specified by the preceding upper case letter is designated to move

Purpose   Provides a feed-in, dwell, rapid-out sequence used for standard drilling and counterboring operations. G82 Standard Drilling With Dwell involves a single plunge of the tool at the plunge speed, a timed delay when the Z axis reaches its lowest point, and a single retraction of the tool to the initial Z position at the rapid speed. The initial Z position is the position of the Z axis before the drilling cycle starts. The duration of the delay is specified by the Dwell command preceding the drill command.

The speed at which the Z axis descends into the material will either be the feed rate specified in the .NC file by the *Feed Speed* command or the plunge speed specified in the Set-Up window of the interface program. The interface program will only recognize the *Feed Speed* command if **Override Program Speed** is not selected in Set-Up.

In drilling, the command code usually appears on the first line of the cycle only, although there may be several holes drilled in the cycle. Also, the Z axis value is always unsigned (i.e., there is no positive or negative sign associated with it).

51

Example

    G90
    G82X10.00Y8.50Z-0.5P10
    G80
    G0X1.00Y1.00Z9.00
    M30

The X and Y axes move at the cutting speed set in the interface program. A hole is drilled at the new location, (10,8.5). To drill the hole, the Z axis moves 0.5 units in a negative direction away from its current (initial) position at the plunge speed, dwells at the bottom of the hole for 10 seconds (P10), and then retracts to its initial position at the rapid speed.

The drill cycle is ended in line 3, and a rapid move is executed in line 4. This rapid move is performed in incremental motion, as specified by the *Absolute Coordinates* command in line 1.

Illustration

Standard Drilling With Dwell

	initial height
	reference height
	part surface
	depth

Dwell

Key

●●●●●●●●●●●	drill motion at plunge speed
————————	rapid motion at rapid speed
initial height	height at which rapid motions between drill holes takes place
reference height	height at which drill is moved at rapid speed before drilling takes place
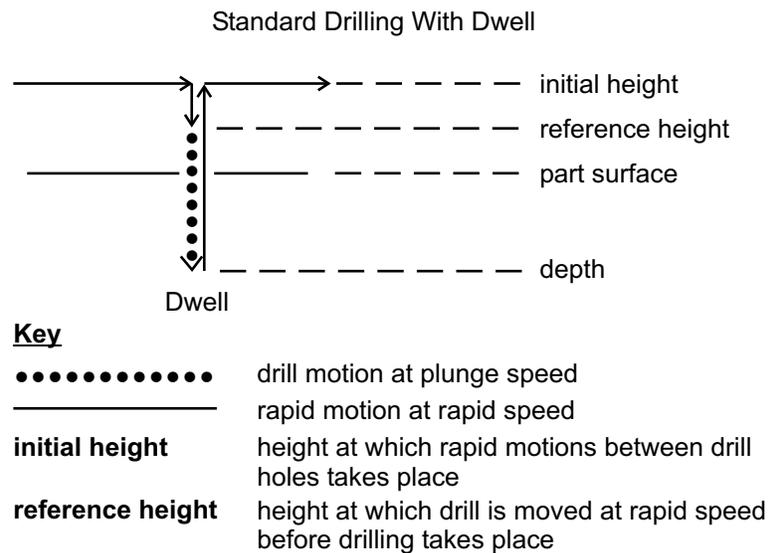
G83   Peck Drilling Cycle

Format

    G83XxYyZtZfZs
        or
    G83XxYyZtZf

X,Y,Z = axes on a three axis machine
x, y, z = distance in incremental units, or absolute coordinates, to which the axes specified by the preceding upper case letter is designated to move
t = total Z axis depth
**f** = first peck increment
s = subsequent peck increments

52

Purpose

Provides a series of feed-in, rapid-out motions used for deep hole drilling. In peck drilling, the tool performs a series of plunges, or pecks, and retractions at the same XY position.

Each peck is done at the plunge speed, and each retraction is done at the rapid speed. The speed at which the Z axis descends into the material will either be the plunge rate specified in the .NC file by the *Feed Speed* command or the plunge speed specified in Set-Up. The interface program will only recognize the *Feed Speed* command if **Override Program Speed** is not selected in Set-Up.

The total depth for the Z axis is specified in the *Peck Drilling Cycle* command. The depth of the first peck and of all other (subsequent) pecks are also specified. The total number of pecks is determined by the total Z axis depth and the depths of the peck increments designated in the G-Code command. Assuming that the total Z axis depth is 10, the first peck increment is 4, and subsequent peck increments are 2, there will be a total of 4 pecks. This figure is arrived at by subtracting the depth of the first peck from the total Z axis depth, then dividing the remainder by the value for subsequent peck increments. If the value of subsequent peck increments is omitted from the G-Code command, all peck increments will have the same value as the first peck.

The Z axis retracts to its initial position after each peck increment. The initial position of the Z axis is its position before the drilling cycle starts. Also, the Z axis value is always specified as unsigned (i.e., there is no positive or negative sign associated with it).

In drilling, the command code usually appears on the first line of the cycle only, although there may be several holes drilled in the cycle.

Example

G90
G83X-5.00Y-2.50Z4.00Z1.00Z0.5
G80
G0X10.00Y10.00Z0.00

Line 2 indicates that subsequent motion commands will be in absolute motion. The line 2 command specifies a peck drilling cycle to take place at the (-5,-2.5) position. The Z axis is specified to plunge a total of 4 units deep.

The first peck will be made at a depth that is 1 unis away from the current Z axis position (0). The tool will then retract to the initial height, the Z=0 point. There will be four additional pecks, making five pecks in total.

The second peck will be made at a position that is 2 units away from

53

the bottom of the first peck. The bottom of the second peck will be -6 units along the Z axis from the initial position, which the tool will retract to after the peck is made. Each of the three remaining pecks in the series will be made at a position that is 0.5 units away in a negative direction from the bottom of the previous peck. When the tool retracts after each peck, it will return to the initial position, which in this case, is the Z=0 point.

The drill cycle is ended in line 3, and a rapid move is performed in line 5. The rapid move is executed in absolute motion, as specified by the *Absolute Coordinates* command in line 1.

Illustration

Peck Drilling

first peck increment

dwell is only on last drilling motion

Key

••••••••••••	drill motion at plunge speed
——————————	rapid motion at rapid speed
initial height	height at which rapid motions between drill holes takes place
reference height	height at which drill is moved at rapid speed before drilling takes place
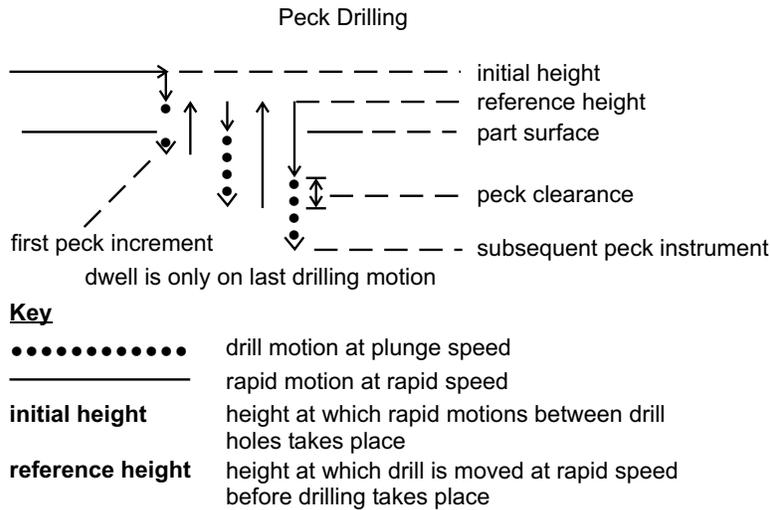
G87   Chip Break Drilling Cycle

Format   **G87XxYyZtZfZs**
                    or
          G87XxYyZtZf

          X,Y,Z = axes on a three axis machine,
          x, y   = distance in incremental units, or absolute coordinates, to which the axes specified by the preceding upper case letter is designated to move
          t = total Z axis depth
          f = first peck increment
          s = subsequent peck increments

Purpose   Provides a series of feed-in, rapid-out motions used for deep hole drilling. This drilling cycle is similar to peck drilling, except that the retract amount is a specified distance. The tool performs a series of plunges, or pecks, and retractions at the same XY position. Chip break drilling focuses on breaking

the surface of the material rather than on withdrawing the tool completely from the piece.

The total depth for the Z axis is specified in the Peck Drill command. The depth of the first peck and of all other (subsequent) pecks are also specified. The total number of pecks is determined by the total Z axis depth and the depths of the peck increments specified in the G-Code command. Assuming that the total Z axis depth is 10, the first peck increment is 4, and subsequent peck increments are 2, there will be a total of 4 pecks. This figure is arrived at by subtracting the depth of the first peck from the total Z axis depth, then dividing the remainder by the value for subsequent peck increments. If the value of subsequent peck increments is omitted from the G-Code command, all peck increments will have the same value as the first peck.

Each peck is done at the plunge speed, and each retraction is done at the rapid speed. The speed at which the Z axis descends into the material will either be the feed rate specified in the .NC file by the *Feed Speed* command or the plunge speed specified in Set-Up. The interface program will only recognize the *Feed Speed* command if **Override Program Speed** is not selected in Set-Up.

In drilling, the command code usually appears on the first line of the cycle only, although there may be several holes drilled in the cycle. Also, the Z axis value is always unsigned (i.e., there is no positive or negative sign in front of it).

Example

G92X0.00Y0.00Z0.00
G90
G87X-3.50Y-6.00Z6.00Z2.00
G80
G0X7.50Y8.00Z2.00

In line 1 of the command set above, the *Absolute Pos*ition command sets the current position as the new software home. Line 2 indicates that subsequent motion commands will be in absolute motion. The line 3 command specifies a chip break drilling cycle to take place at the (-3.5,) position. The Z axis is specified to plunge a total of 6 units deep.

The first peck increment will be made at a depth that is 2 units away from the current Z axis position (0). The tool will then retract .01 inch. From that position, it will plunge to a depth that is 2 units away from the bottom of the first peck. There will be three pecks in all, and after each peck, the tool will retract .01 inch. Since the value of subsequent pecks is not specified, the value of the first peck is used for all pecks.

The drill cycle is ended in line 4, and a rapid move is executed in line 5. The rapid move is performed in absolute motion, as specified by the *Absolute Coordinates* command in line 2.

Chip Breaking Cycle

first peck increment
retract amount
dwell is only on last drilling motion

<u>**Key**</u>

●●●●●●●●●●● drill motion at plunge speed

────────── rapid motion at rapid speed

initial height height at which rapid motions between drill holes takes place

reference height height at which drill is moved at rapid speed before drilling takes place
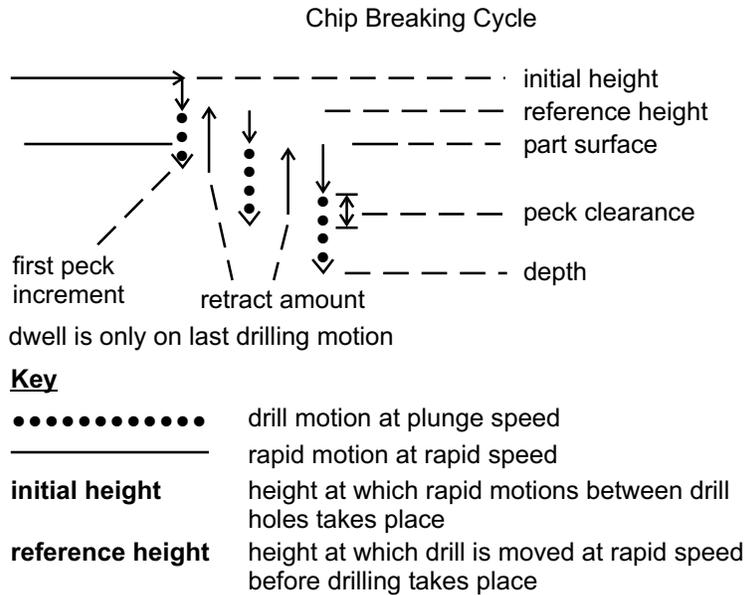
G90 Absolute Coordinates

Format **G90**

Purpose Indicates absolute motion (i.e., movement of the axis to an absolute coordinate) for the motion commands that follow. The values specified for the axes in the motion commands are absolute with respect to the software home position. Absolute motion is the interface program default. All commands are performed in absolute motion unless incremental motion (G91) is specified in the .NC file.

Example **G90**

G1X2.00 Y4.00 Z2.00
G1X-1.00 Y5.00 Z0.00

The first line of this command set specifies that subsequent motion commands will be performed in absolute motion.

The motion command in line two sends the X axis to a point two units away in a positive direction (+2) from the software home, the Y axis to a point four units away in a positive direction (+4) from the software home, and the Z axis to a point two units away in a positive direction (+2) from the software home. The coordinates the axes will move to are (2,4,2).

The motion command in line three sends the X axis to the -1 position in relation to the software home (-3 units away from its previous position), the Y axis to the +5 position (+1 unit away from its previous position), and the Z axis

to the 0 position (the software home location, -2 units away from its previous position). The coordinates the axes will move to are (-1,5,0).

G91   Incremental Coordinates

Format   **G91**

Purpose Indicates incremental motion (i.e., movement of the axis to a position a specified distance away from its previous position) for the motion commands that follow. Once specified, commands continue to be performed in incremental motion until absolute motion is indicated in the .NC file.

Example   **G91**
G1X2.00 Y4.00 Z2.00
G1X-1.00 Y5.00 Z0.00

The first command specifies that subsequent motion commands will be performed in incremental motion.

The motion command in line two sends the X axis to a point two units away in a positive direction (+2) from its current position, the Y axis to a point four units away in a positive direction (+4) from its current position, and the Z axis to a point two units away in a positive direction (+2) from its current position. Assuming the current position of the machine is (1,1,1), the machine will then move to position (3,5,3).

The motion command in line three sends the X axis -1 unit away from its current position of +3, the Y axis +5 units away from its current position of +5, and the Z axis 0 units away from its current position of +3. The machine will move from (3,5,3) to (2,10,3).

G 92   Set Absolute Position

Format   **G92 XxYyZz**

X,Y,Z = axes whose coordinates are being specified
x, y, z = absolute coordinate being assigned to the axis specified by the preceding upper case letter

Purpose   Sets the new origin, or software home, by changing the absolute position of one or more axes. New absolute coordinates are assigned to the axes in their current positions, thereby establishing a new software home. Any absolute coordinate specified in a subsequent command is referenced to the new software home position. The current position may be designated as the new software home by assigning the absolute coordinate of 0 to each axis.

Example 1   **G90**
                       G1X2.00 Y4.00 Z2.00
                       G92X0Y0Z0
                       G1X-1.00 Y5.00 Z0.00

The command on the first line indicates that subsequent motion commands will be performed in absolute motion.

In line two, the X axis will be sent to a point 2 units away in a positive direction from the existing software home, the Y axis will be sent to a point 4 units away in a positive direction from the existing software home, and the Z axis will be sent to a point 2 units away in a positive direction from the existing software home. The coordinates the axes will move to are (2,4,2).

The line three command establishes a new origin, or software home, using Absolute Position. Each axis is assigned a new absolute coordinate - 0 (zero) - for its current position, changing the coordinates of the current position from (2,4,2) to (0,0,0). The current position has been designated as the new origin, or software home. Subsequent motion commands will be referenced to this new software home.

In line four, the axes make a linear cutting move to absolute coordinates. These coordinates are relative to the new software home established in line three. The X axis is specified to move to a position -1 unit away from the new origin (1 unit in a negative direction away from its current position), the Y axis is specified to move to a position +5 units away from the new origin (5 units in a positive direction away from its current position), and the Z axis is specified to move to the new origin (which is its current position, so it does not move). The final coordinates of the axes will be (-1,5,0).

Example 2   **G90**
                       G1X2.00 Y4.00 Z2.00
                       G92X4Y3Z6
                       G1X-1.00 Y5.00 Z0.00

The command on the first line indicates that subsequent motion commands will be performed in absolute motion. In line two, the axes are sent to the same positions designated by the line two command in Example 1. (See Example 1 for a description of axes movement.) The coordinates the axes will move to are 2,4,2.

The line three command establishes a new origin, or software home, using Absolute Position. Each axis is assigned a new absolute coordinate for its current position, changing the coordinates of the current position from (2,4,2) to (4,3,6). The current position is referenced to the new origin, or software home, which is 4 units away in a negative direction along the X axis, 3 units away in a negative direction along the Y axis, and 6 units away in a negative direction on the Z axis. Subsequent motion commands will be referenced to this new origin as well.

In line four, the axes make a linear cutting move to absolute coordinates. These coordinates are relative to the new software home established in line three. The X axis is specified to move to a position -1 unit away from the new origin (5 units away in a negative direction from its current position), the Y axis is specified to move to a position +5 units away from the new origin (2 units away in a positive direction from its current position), and the Z axis is specified to move to the new origin (6 units away in a negative direction from its current position). The final coordinates of the axes will be (-1,5,0).

M0   Program Pause

Format   **M0**

Purpose   Stops the .NC file from running. The file will continue to run from the point where it stopped when you click on the Resume button in the interface..

Example   **G91**
G1X4.00Y-2.00Z1.00
M0
G1X5.00Y0.00Z3.00

The first line of this command set specifies that subsequent motion commands will be performed in incremental motion.

Following the first linear cutting command (line 2), the file is stopped using the Program Stop command. After the Resume button is clicked, the file resumes with another linear cutting move.

M3 or M4   Spindle On

Format   **M3**
or
M4

Purpose   Starts the spindle. After the spindle is turned on, the file is paused for the amount of time specified next to Spindle Delay in Set-Up.

Spindle On commands will only be recognized by the Techno CNC G-Code Interface if Auto is selected in the interface.

Example 1	**M3**
	G90
	G1X2.50 Y4.00 Z-2.00
	G1X-1.00 Y5.50 Z-1.00
	M30

In the command set above, the Spindle On command precedes all motion commands intended to perform cutting (lines 3 and 4), since the spindle has to be turned on in order for cutting to take place. The *End Of Data* command, M30, turns the spindle off as it ends the block of commands.

Example 2	**M4**
	G91
	G1X8.50Y6.00Z-4.50
	G1X12.00Y2.50Z-3.50
	M30

The first command in this set turns the spindle on. Cutting moves are then performed in incremental motion. The *End Of Data* command turns the spindle off at the end of the command set.

M5   Spindle Off

Format   **M5**

Purpose   Turns the spindle off. This command will stop the spindle only if Auto has been selected in the interface.

Example	**M3**
	G90
	G1X2.00 Y4.00 Z-2.00
	M5
	M0
	M3
	G1X-1.00 Y5.00 Z-3.00
	M30

In the command set above, the spindle is designated to be turned on at the beginning, in line 1, and then turned off in line 4, following the linear cutting command. The program is stopped in line 5 to allow for adjustments of the machine, tool, or workpiece before the machine is designated to cut again. After the desired adjustments are made, the file will continue to run from the point where it stopped when the Resume button in the interface program's Run window is clicked. The *End Of Data* command (M30), which marks the end of the file, turns the spindle off.

M6   Tool Change

Format   **M6T#**

T indicates "tool"
\# = tool number

Purpose   Allows the tool to be changed by turning the spindle and coolant off and by stopping the .NC file from running. After the tool change, if the spindle and coolant were on before, they will automatically be turned back on. The tool will be returned to its position prior to the interruption of the .NC file.

After the .NC file stops running as a result of a Tool Change command, the Tool Change window will pop up and show the number of the tool needed for the tool change. The machine can be jogged to a convenient position for changing the tool. The file will continue to run with the new tool in place when the Resume button is clicked.

Example   **M3**
G90
G1X2.00Y4.00Z-2.00
M6T2
G1X-1.00Y5.00Z-3.00
M30

In the command set above, the spindle is turned on in line 1, absolute motion for subsequent motion commands is specified in line 2, and a linear cutting move is indicated in line 3. A change from one tool to another is indicated by the Tool Change command in line 4. After the change to tool #2 is completed, the spindle is automatically turned back on. The tool is returned to position (2,4,-2). After the Resume button in the interface program's Run window is clicked, the file continues to run, and a linear cutting move in absolute motion is performed. Finally, the *End Of Data* command designates the end of this block of commands and turns the spindle off.

M7 or M8   Coolant On

Format   **M7**
or
M8

Purpose   Turns the coolant on. This command will only be recognized by the Techno CNC G-Code Interface if Auto is selected under Coolant in the interface.

Example	**M7**
	G91
	G1X3.00Y5.00Z-3.00
	G1X4.00Y4.00Z-4.00
	M30

In the command set above, the coolant is turned on in line 1, before any cutting is performed. Incremental motion for subsequent motion commands is specified in line 2, and linear cutting moves are indicated in lines 3 and 4. The *End Of Data* command designates the end of this block of commands and turns the coolant off.

M9   Coolant Off

Format   **M9**

Purpose   Turns the coolant off. This command will only be recognized by the Techno CNC G-Code Interface if Auto is selected under Coolant in the interface.

Example   **M7**
G90
G1X2.00Y4.00Z-2.00
G1X-1.00Y5.00Z-3.00
M9
G1X-2.00Y6.00Z-4.00
M7
G1X-3.00Y8.00Z-2.00
M30

The coolant is turned on, using the Coolant On command, at the beginning of this command set. After the first two cutting commands (lines 3 and 4), the coolant is turned off with the Coolant Off command (line 5). It is turned on again in line 7, after the next cutting command. The subsequent cut (line 8) is made with the coolant on. Then, the program is ended with the *End Of Data* command, which also turns the coolant off.

M30   End Of Data

Format   **M30**

Purpose   Designates the end of a block of commands in a file. This command also turns off the spindle and coolant if Auto has been selected in the interface. The end of a G-Code program is normally marked as such using the End Of Data (M30). However, it is not necessary to include this command in an .NC file that will be run in the Techno CNC G-Code Interface. After the interface

program executes the last command in the file, it will stop running the file, and will turn off the spindle and coolant.

Example   **M48**
G90
G1X2.00 Y4.00 Z2.00
G1X-1.00 Y5.00 Z0.00
M30

The block of commands above consists of two linear cutting moves performed in incremental motion. The block is ended by the *End Of Data* command, which also turns off the spindle and coolant.

M90   Output Off

Format   M90 OUT#

Purpose   Turns the designated output off. This command is used in conjunction with the M91 command (see description below), which turns the output on. An output number from 0 to 15 must be specified for this command. Only one output can be designated to be turned off by a single command.

Example   **M91 OUT1**
G90
G1X-5.00 Y2.00
G1X0.00 Y1.00
M90 OUT1

The first command in this example turns on output 1. The command in the second line indicates that the following commands will be performed in absolute motion. A linear cutting move is made: the X axis moves to the -5 position and the Y axis moves to the +2 position. In the next cutting move, the X axis moves to the 0 position and the Y axis moves to the +1 position. Then, output 1 is turned off.

M91   Output On

Format   **M91 OUT#**

Purpose   Turns the designated output on. This command is used in conjunction with the M90 command (see description above), which turns power to the output off. An output number from 0 to 15 must be specified for this command. Only one output can be designated to be turned on by a single command.

Example   **M91 OUT1**
                               G90
                               G1X2.00Y4.00
                               G1X3.00Y3.00
                               M90 OUT1

In this example, output 1 is turned on. Then, absolute motion is specified for the commands that follow. A linear cutting move is made next: the X axis moves to the +2 position and the Y axis moves to the +4 position. In the next cutting move, the X axis moves to the +3 position and the Y axis moves to the +3 position. Then, output 1 is turned off.

V.   Troubleshooting

1. Technical Support

Although most common problems can be solved with this Troubleshooting guide, some specific questions may require help. Please have the following information ready when requesting technical support:

- **controller model**
- **machine model**
- **place of purchase**

E-MAIL support@techno-isel.com: E-mail your questions and background information. Please include your telephone number.

FAX (516) 358-2576: Fax Techno's expert support team detailed background information along with specific questions. Include phone and fax numbers to ensure prompt response.

PHONE (516) 328-3970: Call Techno and specify you need technical assistance. Faxing detailed information before calling is recommended. Please have your information and questions ready.

2. General Problems

This Troubleshooting guide addresses problems that may be encountered while using this software on your machine. It is not intended to answer machining problems that occur independent of this software.

Symptom	Possible Cause	Remedy
No communication between controller and PC	a. COM port set improperly b. Controller connected to non-functional COM port c. Controller power off d. Connecting cable improperly attached e. Emergency stop switch depressed f. Wrong cable	a. Change COM port setting in Interface's Default Configuration. b. See PC manual for functional COM ports and correct jumper settings c. Check controller power switch is On. d. The end of the connecting cable that should be attached to your computer is labeled 'PC'. Make sure this attached to correct COM Port. e. Disengage the emergency stop switch on your machine. f. You must use the cable supplied by Techno. Call if your cable needs replacement.
Motor stalls during travel, loses position, or doesn't return to correct position.	a. Speeds exceeding limit of machine b. Dull cutter c. Machine needs cleaning, lubing, or maintenance d. C-Series motor driver cable needs to be repaired or replaced.	a. Reduce rapid, cutting, and plunge speeds in Setup of file, which is causing problem, and activate the Override programmed speed function. b. Disconnect power and check cutter. If it is worn it must be reconditioned or replaced. c. Disconnect power and check machine parts including cables, cords, cutting tools and accessories to make sure

		they are clean and dry. **All components should be inspected regularly.** d. Contact Techno for support.
No controller power	a. Power switch off (it happens) b. Emergency stop engaged c. Power cord loose or disconnected d. Fuse needs replacement	a. Check controller power switch. b. Disengage emergency stop button. c. Make sure power cord is plugged into a live outlet. d. Check the fuse located directly beneath the power switch of your controller.
Part produced is wrong size	a. Scale factors improperly set. b. GCODE program error c. Incorrect cutter size d. Speeds too high for conditions	a. In Configuration Defaults, check that scale factors are set to match the unit of measure used in your GCODE drawing. Then check that they are correct in the Setup for the particular .ncd file. b. Check your GCODE program for illegal operations. The Interface's Preview function also allows you to see your program executed line-by-line to aid in pinpointing errors. c. Check that cutter size is precisely matched to your specifications. d. Reduce rapid, cutting, and plunge speeds in Setup and activate the Override Programmed Speed feature.
Machine cuts at incorrect depth	a. Z-axis scale factor improperly set b. Loose cutter c. Speeds too high for conditions d. Incorrect programmed depths	a. In Configuration Defaults, check Z-axis scale factor to make sure it is et to match the unit of measure used in your GCODE drawing. b. Disconnect power to

		your machine and tighten cutter. c. Reduce rapid, cutting, and plunge speeds in Setup and activate the Override Programmed Speed feature. d. Check your GCODE program.
Downloaded program doesn't run	.**OUT** file is too large	If possible, reduce the size of the file you are attempting to download. If you cannot reduce the file enough that it can be downloaded, it cannot be run as a standalone program and must be run as a translated program or translated/ executed by activating the trial execute feature.

3. Controller Error Codes

Occasionally, problems encountered with the Interface or your GCODE program will appear as an error code in the status display window of your controller. The diagram below shows the display panel of a DaVinci with each button labeled and briefly explained. Consult your controller manual for more extensive information about the display panel and error codes.

DaVinci Display Panel
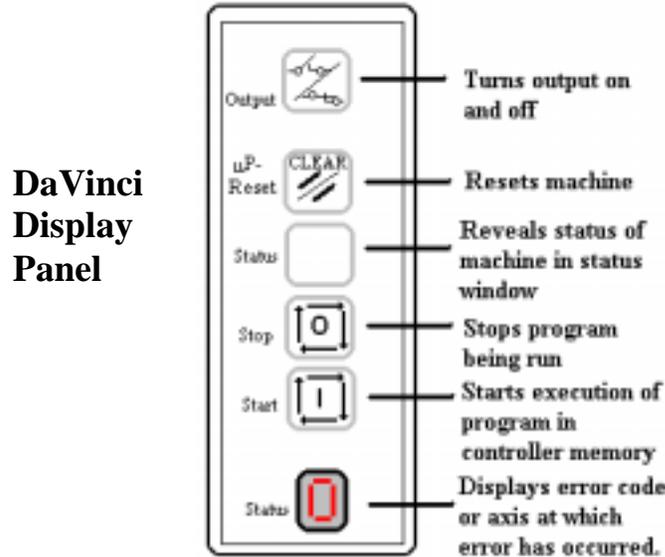
▪ Error Codes

0	No error. Normal display reading
1	Command cannot be interpreted
2	Limit switch encountered
3	Illegal number of axes
4	Axis not defined
5	Syntax error
6	Out of memory
7	Illegal parameters
8	Illegal branch
A	Impulse command parameter must be between 1 – 6.
B	Communication error
C	Carriage return expected
D	Illegal speed specified
E	Loop error; no forward loops are allowed
F	User has pressed stop button
H	Improper data/parameter

For some error codes, pressing the Status button after the error code appears will identify which axis has the error. The following numbers are used to represent the axes:

- 1=X
- 2=Y
- 4=Z
- 7=All

▪ **Press the µP-Reset button to reset the system after any hardware errors are corrected. Operation automatically resumes after the correction of software errors.**

=   Unexpected carriage return received

- Most of these error codes will not occur as a result of anything done in the GCODE Interface. Those that might are mentioned in the table below. If an error code appears which is not included here, consult your controller manual.

Error Code	Possible Cause	Remedy
1	a. GCODE program command cannot be interpreted b. If running a downloaded program, `.OUT` file may be too large	a. Check GCODE program for illegal operation or improper syntax. b. Reduce file size if possible. If file cannot be reduced, it cannot be downloaded to controller's memory and must be run as a translated program.
2	Axis has touched limit switch	Hold Status button until Status Display Window diplays which limit switch has been encountered. Turn knobs on the end of the appropriate motor several turns. Press Reset and Status Display Window should display 0 (no error). If not, repeat. (More than one axis may have touched its limit switch)
5	Syntax error in your GCODE file	Check your GCODE file
6	Controller is out of memory	`.OUT` file is too large and must be reduced or run as a translated program.
D	Illegal speed specified	Reduce speeds set in Configuration Defaults and Setup. Activate Override Programmed Speed function.
H	Improper data/parameter	Check that Configuration Default and Setup settings are within machine's limits. Check GCODE file for syntax errors or illegal commands.

Index