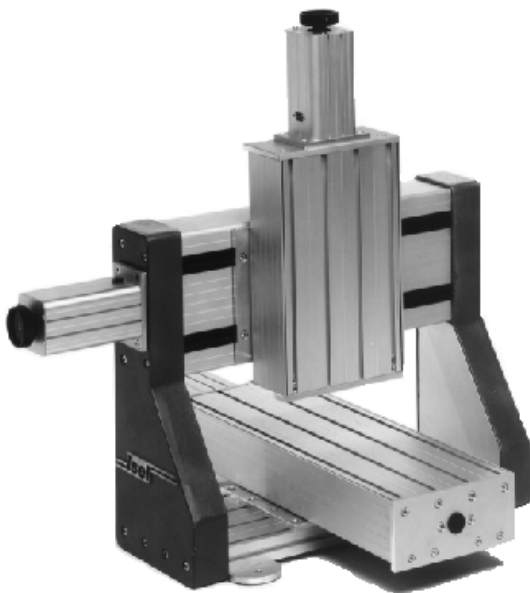


Techno*isel*

GCODE INTERFACE

FOR DAVINCI & C-SERIES CONTROLLERS
WINDOWS VERSION



HTM00490103

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



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

WARNING! IMPROPER OR UNSAFE OPERATION OF THE MACHINE MAY RESULT IN PERSONAL INJURY AND/OR DAMAGE TO THE EQUIPMENT.

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.

DO NOT OPERATE MACHINE IF YOU ARE UNFAMILIAR WITH THESE SAFE OPERATING INSTRUCTIONS. DO NOT OPERATE MACHINE WITHOUT KNOWING WHERE THE EMERGENCY STOP SWITCH IS LOCATED.


Table of Contents

	<u>Page</u>
I. The Techno GCODE Interface	4
System Requirements	5
Installing the Software	5
GCODE Interface Menu Map	6
II. Quick-Start Tutorial	7
III. Explanation of Functions and Terms	14
1. Configure Defaults	15
Explanation of Configuration Parameters	16
Practice Lesson: Trial Executing a File	20
Practice Lesson: Running a File with Trial Execute Off	21
Practice Lesson: Using Offsets	23
Practice Lesson: Saving Jogged Position as Offsets	24
2. Jog	25
3. Translate/Execute	29
Translating/Executing a GCODE File	30
4. Run Translated Program	35
Running a Translated Program	35
5. Download	37
Downloading and Running a Standalone Program	37
6. Preview	39
Previewing Your GCODE Toolpath	39
IV. GCODE Command Summary	41
1. Set-up Commands	41
2. Routing Commands	42
V. Troubleshooting	43
1. Technical Support	43
2. General Problems	44
3. Controller Error Codes	47
Index	49

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.



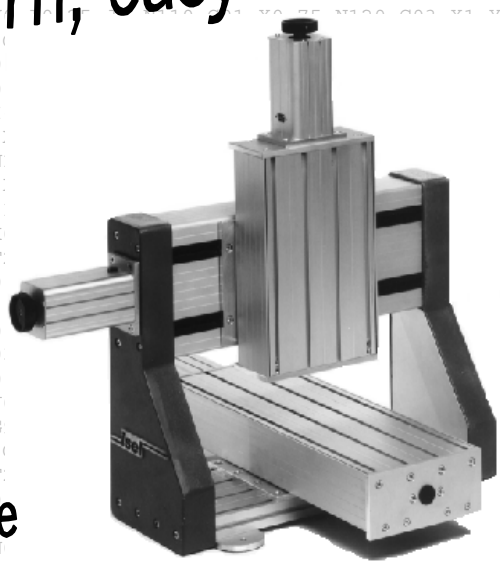
Techno*isel*
GCODE Interface for
DaVinci and C-Series
Windows Version 1.18
(c) 1998 Techno, Inc.
H22T59-GCODE-C

Quick to learn, easy to use!

Download and run files from your controller

Preview toolpaths

Create custom configurations for each file



N030 G90 N040 M03 N050 G00 X0.75 Y1.0 Z0.2 N060 G01 Z-0.1 F5.0 N070 X0.25 F20.0 N080 G03 X0 Y0.75 I0.25
I0.25 N060 G01 Z-0.1 F5.0 N070 X0.25 F20.0 N080 G03 X0 Y0.75 I0.25 J-.125 N
01 Z-0.1 F5.0 N070 X0.25 F20.0 N080 G03 X0 Y0.75 I0.25 J-.125 N090 G01 Y
F20.0 N080 G03 X0 Y0.75 I0.25 J-.125 N090 G01 Y0.75 I0.25 N100 G01 X0
.75 I0.25 J-.125 N100 G01 X0.75 I0.25 J-.125 N110 G01 X0
N090 G01 X0.75 I0.25 J-.125 N120 G01 X0.75 I0.25 J-.125 N130 G01 X1 Y0.25 I0
G01 X0.75 I0.25 J-.125 N140 G01 X1 Y0.25 I0.25 J-.125 N150 G01 X1 Y0.25 I0.25
5 I0.25 J-.125 N130 G01 Y0.75 N140
0 G01 Y0.75 N140 G03 X0.75 Y1 I
1 I-0.25 J0 N150 G00 Z0.2 N160 :
0 G00 Z0.2 N160 X0.677 Y0.823 N
3 N170 G01 Z-0.1 F5.0 N180 G03 :
.1 F5.0 N180 G03 X0.323 Y0.823 N
.323 Y0.823 I0 J0.25 N190 G03 X
N190 G03 X0.5 Y0.896 I-0.177 F
0.0 N200 G01 X0.177 Y0.677 N210
X0.104 I0.177 J-0.177 N210 G03
N210 G03 X0.104 I0.177 J-0.177 N220
N220 G03 X0.177 Y0.323 I0 J0.25 N
N230 G03 X0.177 Y0.323 I0 J0.25 N
N250 G03 X0.677 Y0.177 I0 J0.525 I
N260 N170 G01 Z-0.1 F5.0 N180 G03
N170 G01 Z-0.1 F5.0 N180 G03
N200 G01 X0.177 Y0.677 N210 G03 X
G01 X0.823 Y0.323 N030 G00 X0.75
Y0.75 N140 G03 X0.75 Y1

I-0.25 J0 N150 G00 Z0.2 N160 X0.677 Y0.823
0 X0.677 Y0.823 N170 G01 Z-0.1 F5.0 N180 G0

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

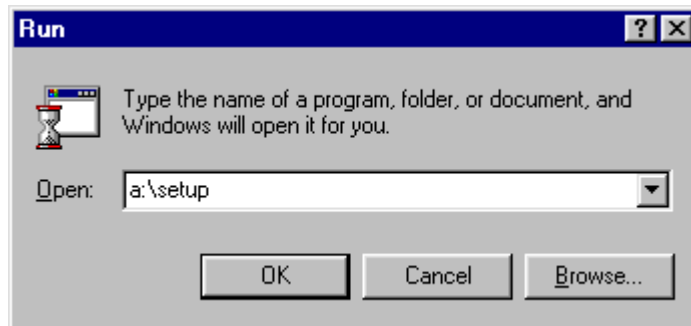
System Requirements


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

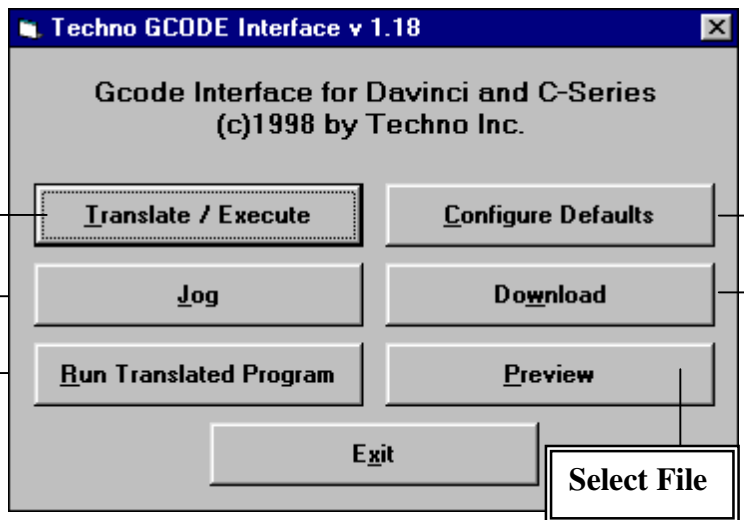
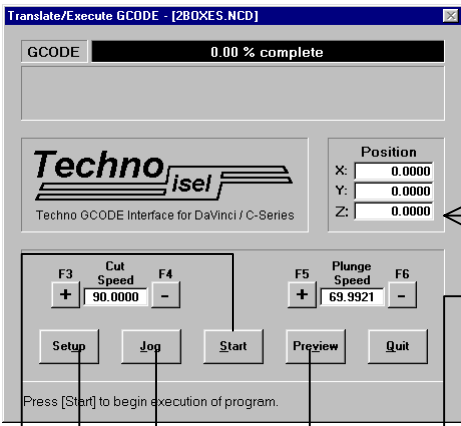
1. Turn on your computer and wait for Windows (95 or 98) to boot up.
2. Insert the GCODE Interface disk.
3. Click 
4. Select and click 
5. Type **a:\setup** in the Run Dialog Box.



6. Click 
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.

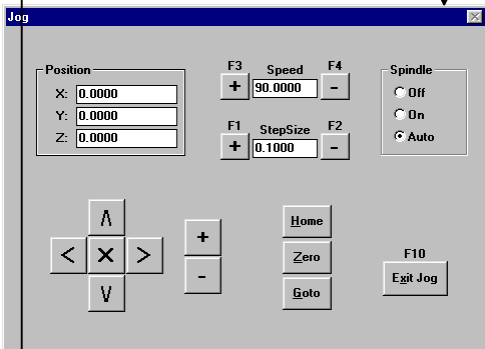
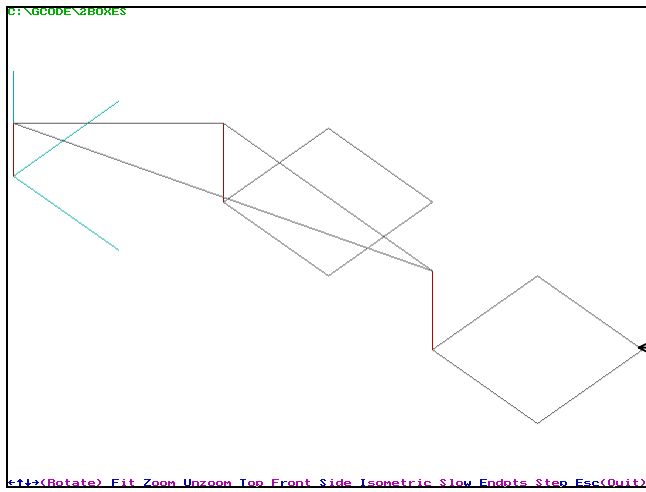
GCODE INTERFACE MENU MAP



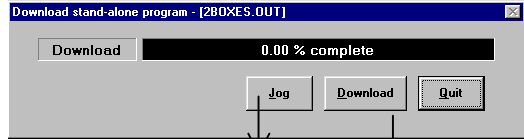
Preview your toolpath

Jog to your toolpath origin.

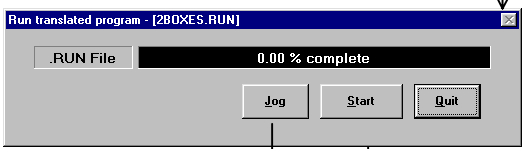
Setup system for your file



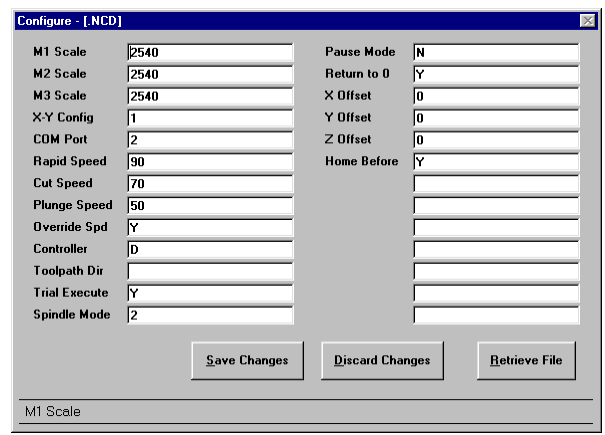
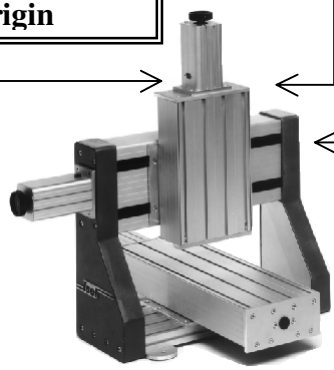
Select File



Jog to your toolpath origin




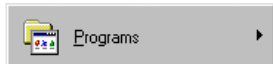

Jog to your toolpath origin



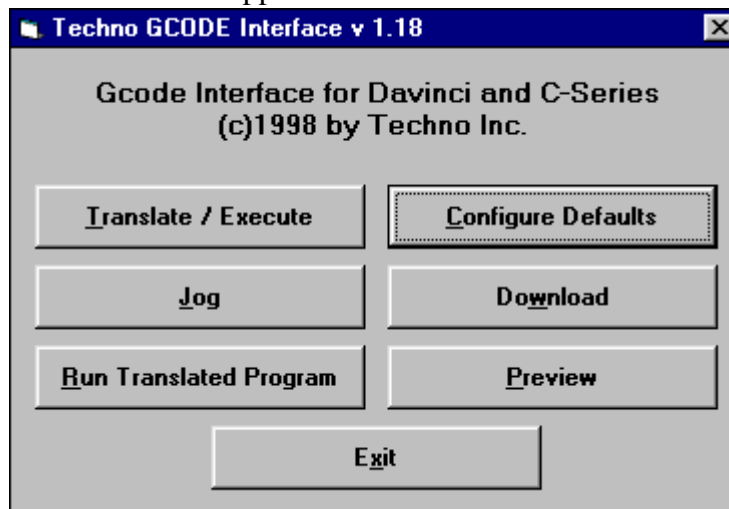
II. Quick-Start Tutorial

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  menu, select  and then 

The Interface main menu appears:



Click 

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.

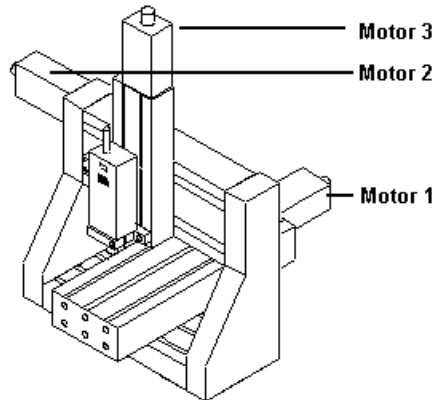
Configure - [.NCD]

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 **DaVinci**
W for **Wizard**
4 for **C-Series 4.0** (e.g. C10)
5 for **C-Series 5.0** (e.g. C142)



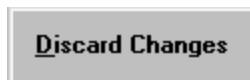
- k) **TOOLPATH DIRECTORY** shows the directory where toolpath files are located.
- l) **TRIAL EXECUTE** lets you translate files with or without executing them.
- m) **SPINDLE MODE** turns the spindle on and off, or sets it to automatic mode.
- n) **PAUSE MODE** lets you override toolchange and other 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



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

Or, click



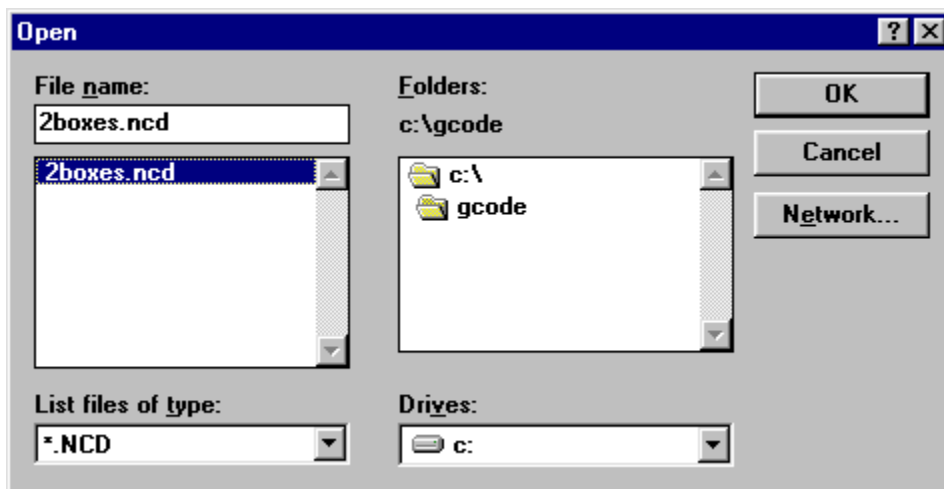
to simply return to the main menu.

4. Click

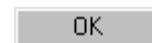


from the main menu.

An Open File dialog box appears.



5. Select **2boxes.ncd** by double clicking it, or highlighting and clicking




The Translate/Execute window opens.



6. Click  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.

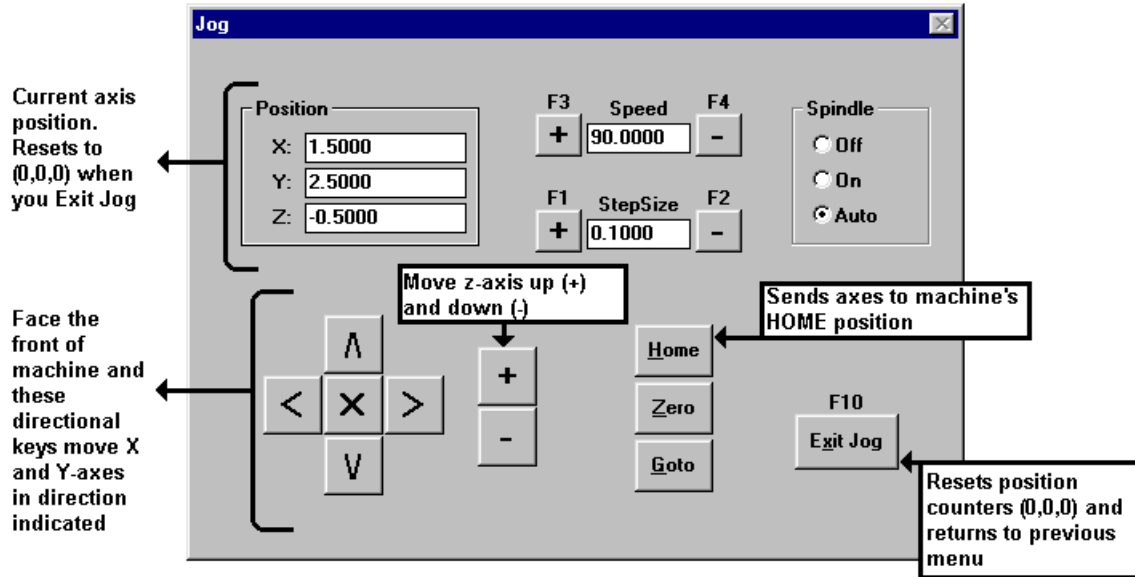
7. Click  to return to Translate/Execute.

8. Click 

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

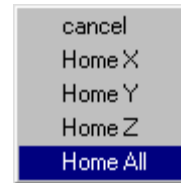
The Jog window is shown below with brief explanations of the functions used in the tutorial. All Jog functions are discussed fully in the *Explanation of Functions and Terms*. Again, if this terminology is new to you, we recommend reading the entire manual before proceeding.

The Jog window opens.



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).

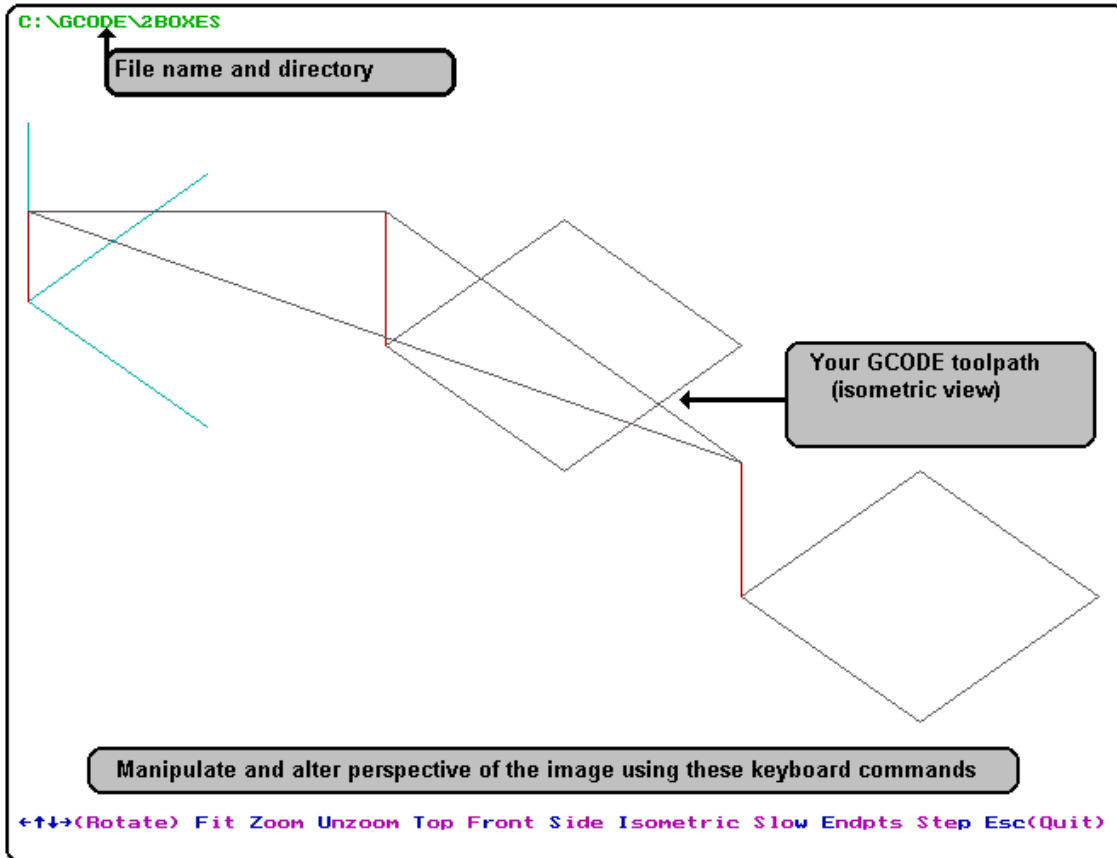
10. Click \wedge \vee \gt \lt $+$ $-$ icons to jog machine to the position displayed at upper left in the window above. (This position is an approximation. Do not worry if you are not at this precise position.) This will be the toolpath origin.

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.



13. Press the **Esc** key to Quit Preview and return to Translate / Execute. You are now ready to run **2boxes.ncd**.

14. Click 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:



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



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).

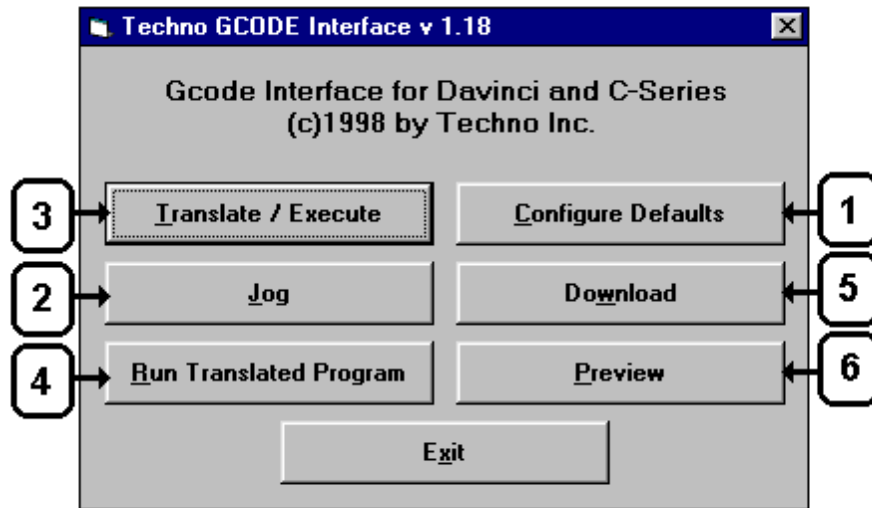
- **The interface automatically adds appropriate file extensions to your program.**

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:



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.



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

Parameter	Value	Parameter	Value
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		

The parameter being set and the options available are briefly described here.

Save Changes: Saves new parameters in .cfg file and returns to the main menu

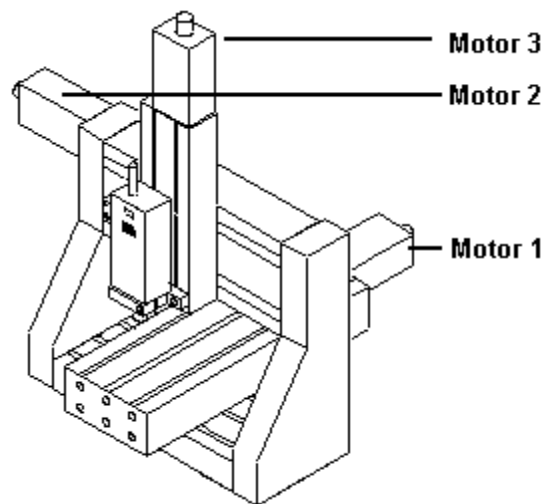
Discard Changes: Ignores changes and returns to main menu

Retrieve File: Open and modify saved configuration files.

M1 Scale

- 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.



Scale factor can be determined using the formula:

$$\frac{\text{Your unit of measure expressed in mm}}{\text{Screw pitch in mm}} \times \left(\frac{400}{\text{motor resolution in steps/rev.}} \right) = \text{Scale Factor}$$

NOTE: 1 inch=25.4 millimeters

Example: Your GCODE drawing is in inches, and the screw pitch on your machine is 10.

$$\frac{25.4}{10} \times 400 = 1016.$$

Your scale factor will be 1016.

- DaVinci units are factory-equipped with 4mm screws.

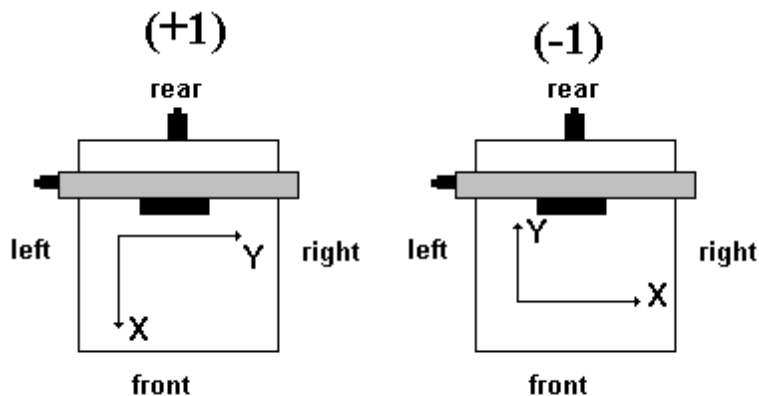
Common Scale Factors

Scale Factor (steps/unit)	Screw Pitch (mm)	Measurement Unit of Drawing
5080	2	Inch
2540	4	Inch
2032	5	Inch
2000	2	Centimeter
1000	4	Centimeter
800	5	Centimeter
200	2	Millimeter
100	4	Millimeter
80	5	Millimeter

DaVinci scale factors are in boldface.

- 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.



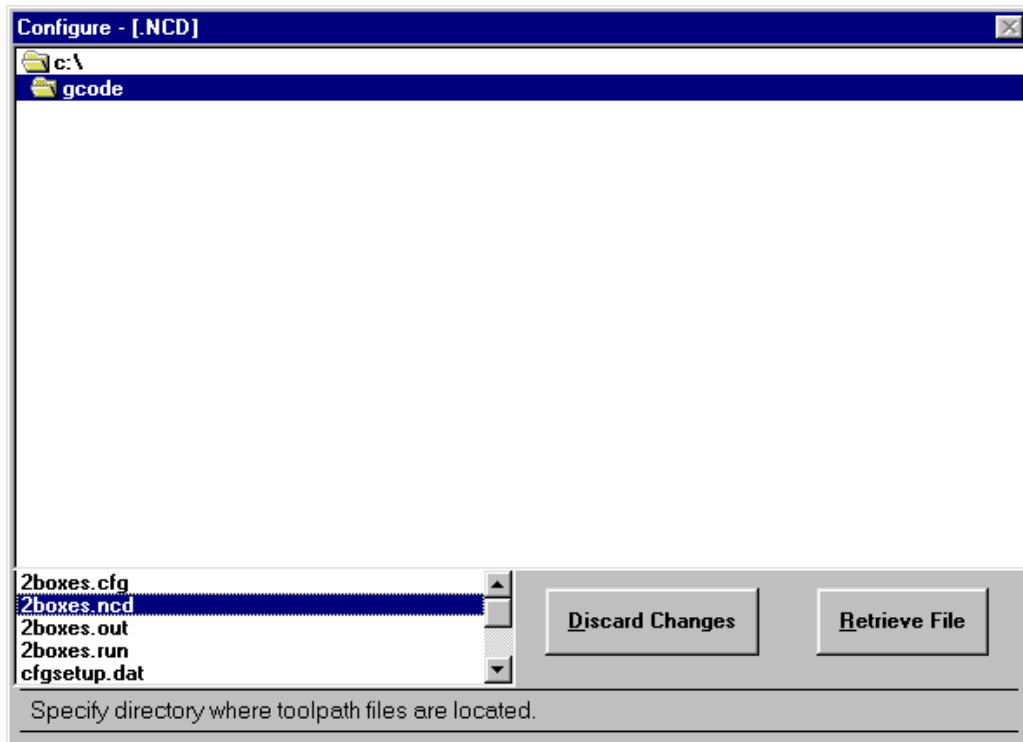
- X-Y Configuration also effects jogging. Although directional keys will move the machine in the same fashion, axes are assigned differently for each configuration.
- e) **COM PORT** should be set to **1** or **2**, depending on which communications port on the back of your PC the controller is plugged 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 **DaVinci**
W for **Wizard**
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 and

Click to return to the previous window.

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

1) **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

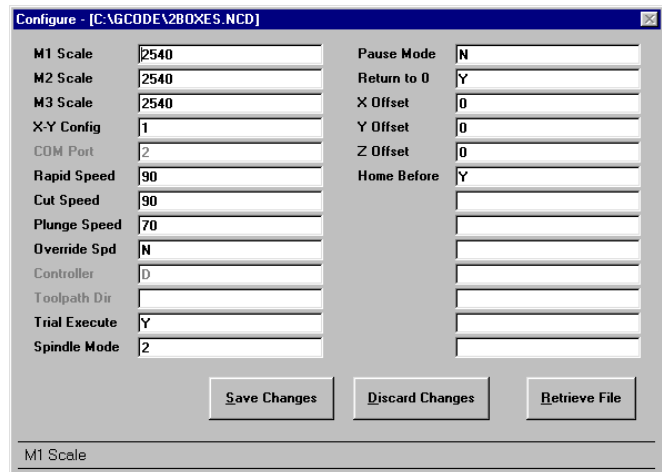
machine to the toolpath origin.

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.



PRACTICE LESSON: RUNNING A FILE WITH TRIAL EXECUTE OFF

1. Click  from the main menu.

2. Select and open `2boxes .ncd`

3. Click 

- 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 

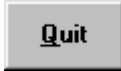
5. Click  from Translate/Execute.

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  when prompted.

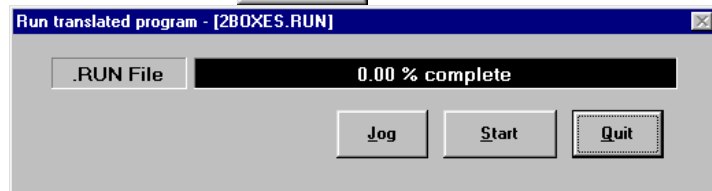
7. Click  to return to the main menu

8. Click 

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

10. Click  to position axes and then  to return to the Run

Translated Program window




11. Click  and your machine runs the file using the new settings.

12. Click  and  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  and open 2boxes .ncd.

2. Click 

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

X Offset	<input type="text" value="1.5"/>
Y Offset	<input type="text" value="2.5"/>
Z Offset	<input type="text" value="-5"/>

3. Click  to return to Translate/Execute.


4. Click  from the  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  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  and open `2boxes.ncd`.

2. Click  from the Translate/Execute window.

3. Click  and select  from the pull-down menu.

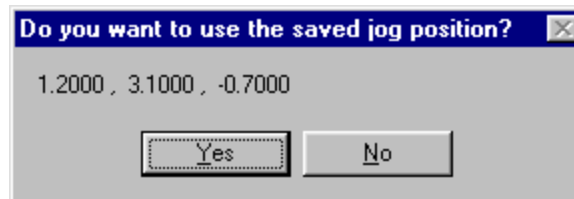
4. Use directional buttons to jog the machine to (1.2, 3.1, -0.7).

5. Click  to return to Translate/Execute.

6. Click 

▪ 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.



8. Click 

▪ The jogged position now appears as the XYZ Offsets.

9. Click  to add these new values to the Setup of the file.

10. Click  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  and  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.

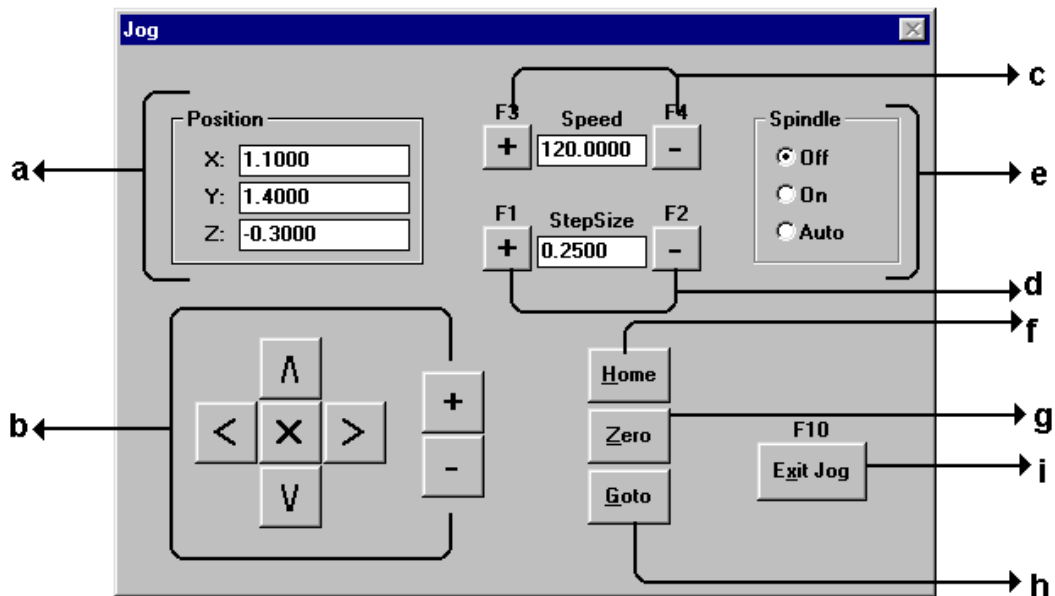
▪ Home Before Run, in conjunction with Offsets and Return to 0, is useful in establishing a common toolpath origin. See the lesson above.

2. Jog

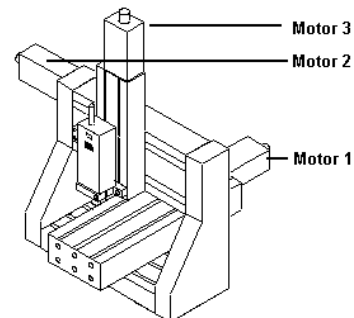
JOGGING manually positions the axes. The axes can be moved to any point you wish to consider your toolpath origin.

- 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



- 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.
- 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 ($\uparrow, \downarrow, \rightarrow, \leftarrow, +, -$) 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.

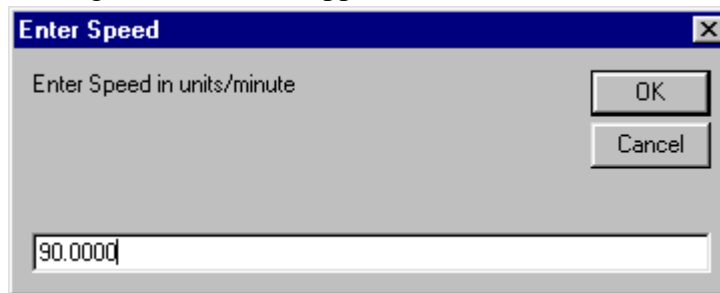



- Directional keys jog the machine as follows (face front of machine):

Using Jog Directional Keys

Icon	Motor # moved	Direction	+1 Config. Axis	-1 Config. Axis
^	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	—	—

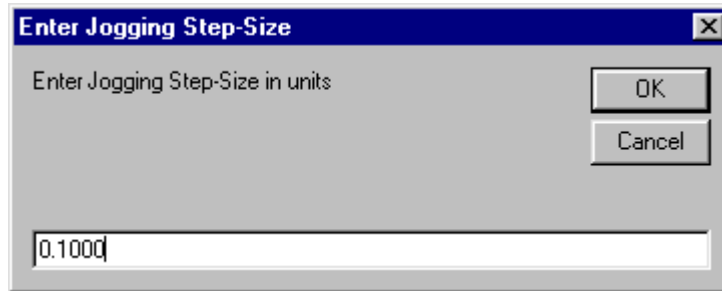
- c. **SPEED** is the rate (units/minute) axes move during jogging. Increase speed by clicking the (+) icon to the left of the speed display or pressing the **F3** key. Decrease speed by clicking the (-) icon to the right of the display or pressing **F4**.
- You can also enter specific speeds by double clicking in the speed display bar. The dialog box below will appear:




Enter the desired speed in units/minute and click 

- 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.

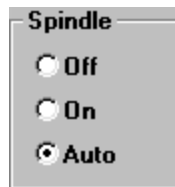
- Step-size can be set to specific values in the same manner as Speed. Double click on the display:



Enter your value in units and click 

- 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.



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.


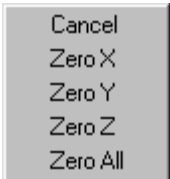
f. **HOME** sends specified axis to the machine's home position.

When you click  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.

When you click  a pull-down menu 

asks which axis you want to zero. You may choose any or all.

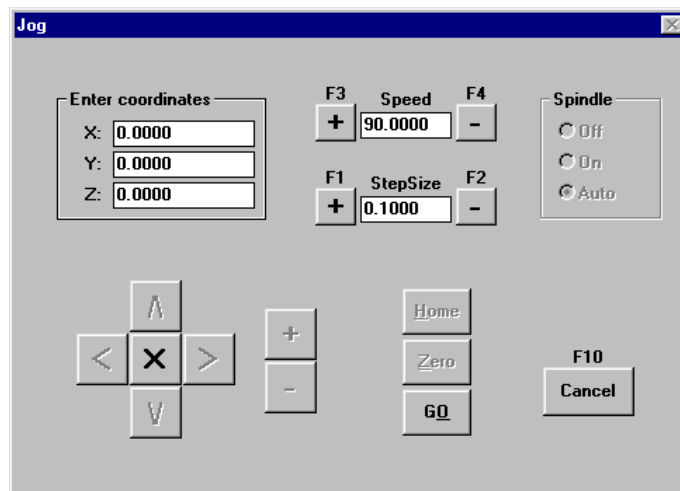
- Zero All is automatically performed every time you exit Jog.

- 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.

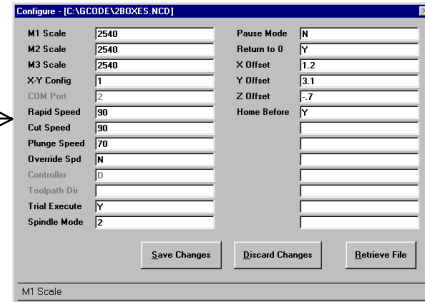
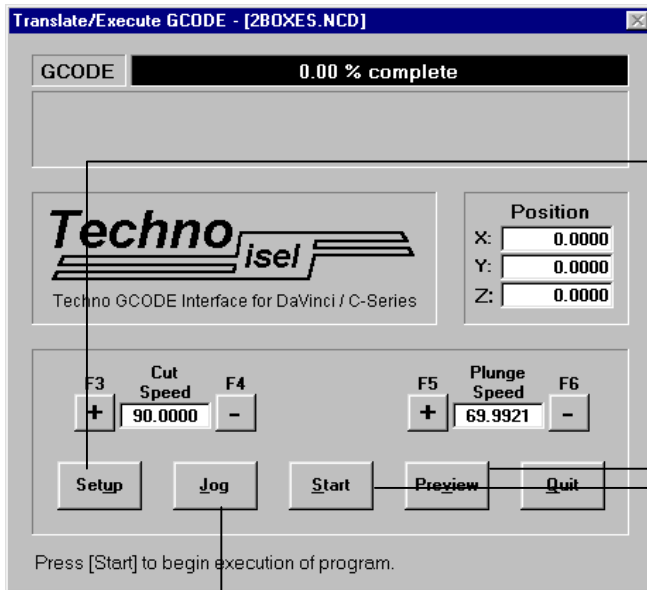
Click **Cancel** to return to the Jog window.

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

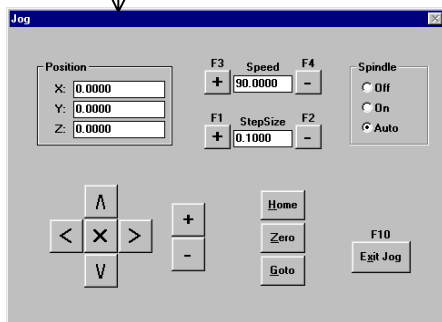
3. Translate / Execute

Translate / Execute

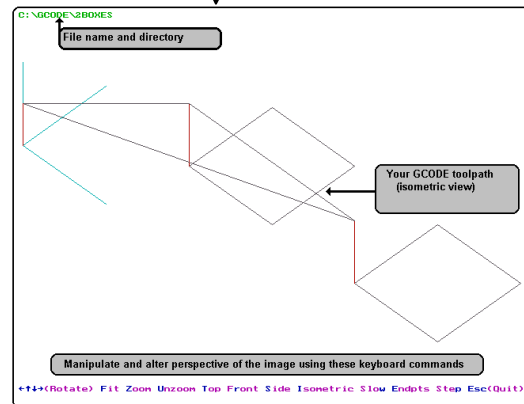
converts GCODE programs into files that can be understood by Techno DaVinci and C-Series controllers, enabling your machine to move along the translated toolpath and produce a part. When you Translate/Execute a file, the Interface writes **.RUN** files that can be run later and **.OUT** files that can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



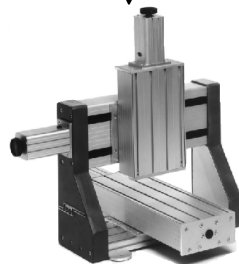
Set system parameters for your **.ncd** file.



Jog the machine to your toolpath origin.




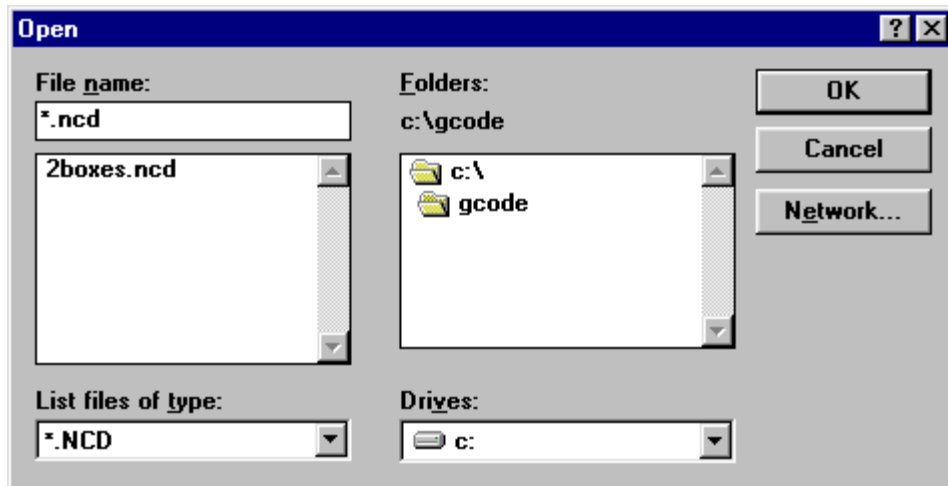
Preview the toolpath before running a program.









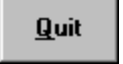
Translate and/or execute your program.

Translating / Executing a GCODE file:

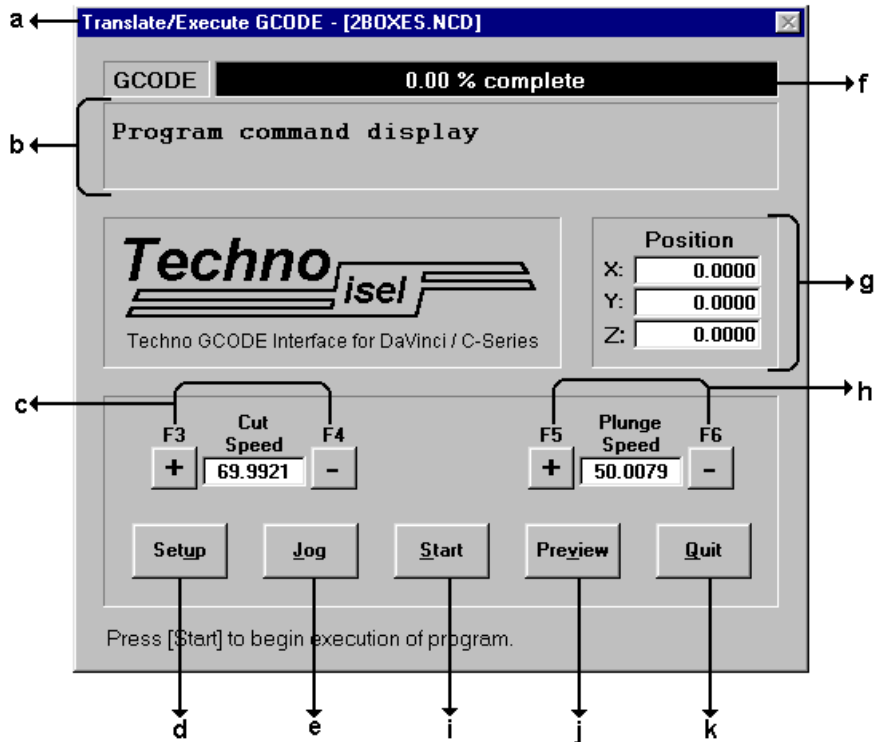
1. Click 
2. An Open File dialog box appears. The toolpath directory shown will be the one specified in Configure Defaults.



Browse folders by double clicking on them.

3. When you've located your file, select it and click 
4. Click  and then  if you've adjusted any parameters.
5. Click  to position the machine manually if necessary.
6. Click  from Translate/Execute.
7. When complete, click  then 

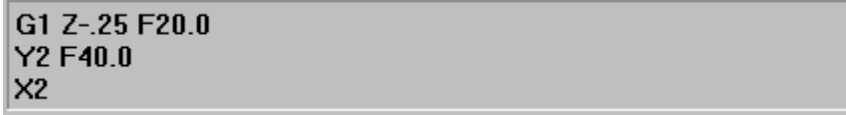
Translate/Execute



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

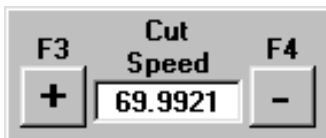


- b. **PROGRAM COMMAND DISPLAY** displays GCODE commands currently executing.



- 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.

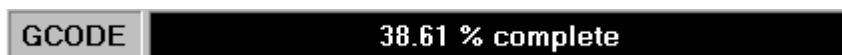
- d. **SETUP** opens the configuration file of your program. Setup functions are the same as Configure Defaults (section 1 of this chapter) **except** they are applied only to the program you are running. From Setup, you can adjust any parameter except Controller and COM Port, which must be changed from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

Clicking  in  will add new values to the **.CFG** file you are running.

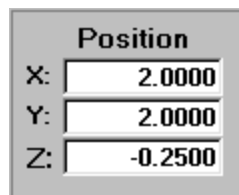
- You may also retrieve saved configuration files in Setup. Remember though that any changes made in Setup will be to the file you are translating and/or executing.
- When you are running a file for the first time, or if a configuration (**.CFG**) file does not exist, the Interface reminds you to check the Setup and that it will create a **.CFG** file for your program using system defaults until changes are made.



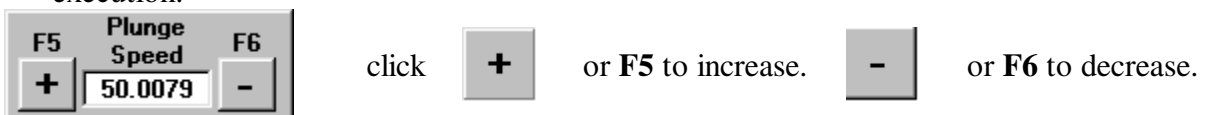
- e. **JOG** accesses the Jog window described in the previous section of this chapter. All keys, icons, and functions are identical.
- Remember that speeds set in Jog only apply while jogging.
- f. The **TASK COMPLETION DISPLAY BAR** displays the percentage of your program completed.



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



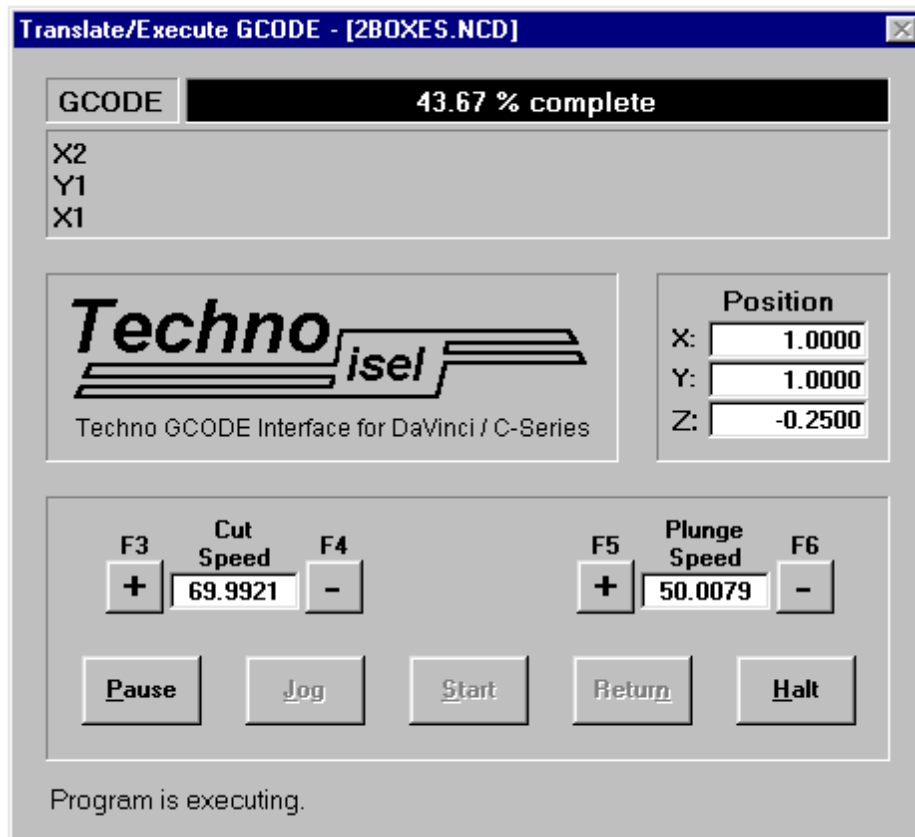
- h. **PLUNGE SPEED**, like cutting speed, can be adjusted manually during translation/execution.



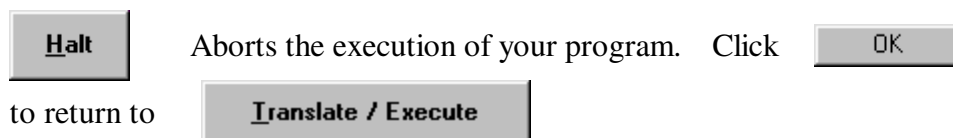
- Altering plunge speed while your program is running may effect production of your part. This may be helpful, however, in determining ideal plunge speed.






i. **START** tells the Interface to begin translating and/or executing your program.

- If Trial Execute is disabled (N) in **Setup** your machine will not execute the File. The Interface will write a file with a **.RUN** extension that can be run later from the **Run 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 **Download** (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:



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



	freezes the program and offers the following options:
	Executes the next step of your program and returns to Paused status.
	Continues execution of your program.
	Returns the machine to the toolpath origin.
	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.

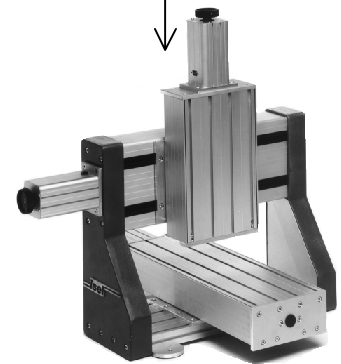
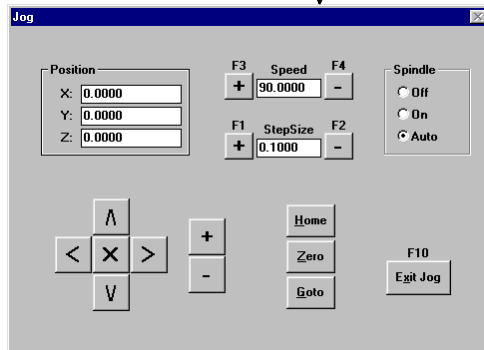
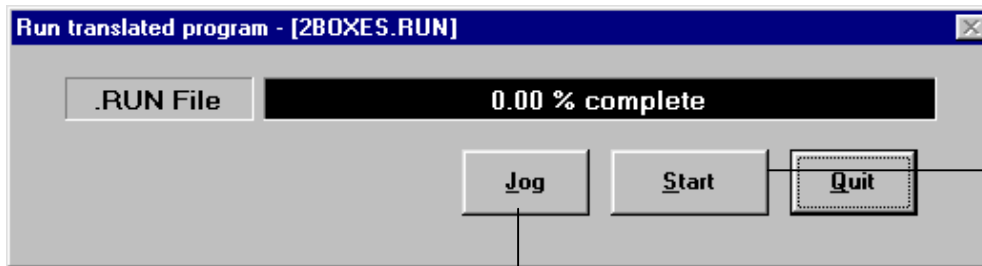
4. Run Translated Program

Run Translated Program

executes previously translated GCODE files.

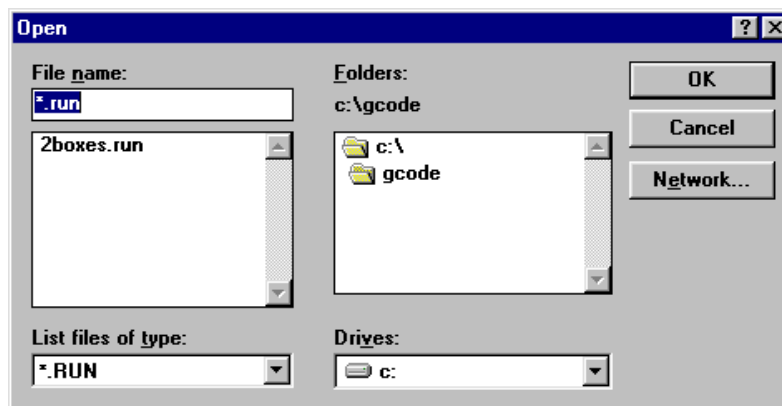
These files have already been written by the Interface and have a **.RUN** extension.

- Setup changes cannot be made from the Run Translated Program window.



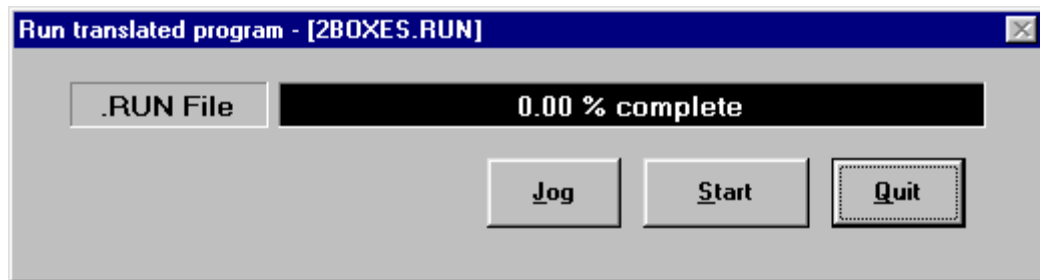
Running a Translated Program:

- Click **Run Translated Program** from the main menu.
- An Open File dialog box shows your toolpath directory and all **.RUN** files.



- 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.

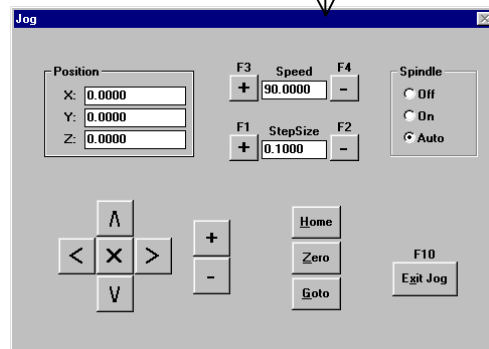
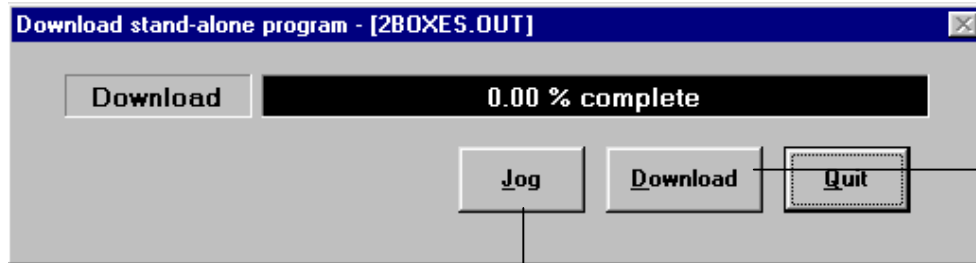


5. Click **Jog** if necessary to manually position axes at your toolpath origin.
6. Click **Exit Jog** to return to Run Translated Program.
7. Click **Start** to begin execution.
8. Click **Quit** when your program has completed.
- The **Pause** and **Halt** options available in Translate/Execute are also available in Run Translated Program.
 - Using Offsets rather than Jogging is helpful in Run Translated Program since Setup cannot be accessed.

5. Download




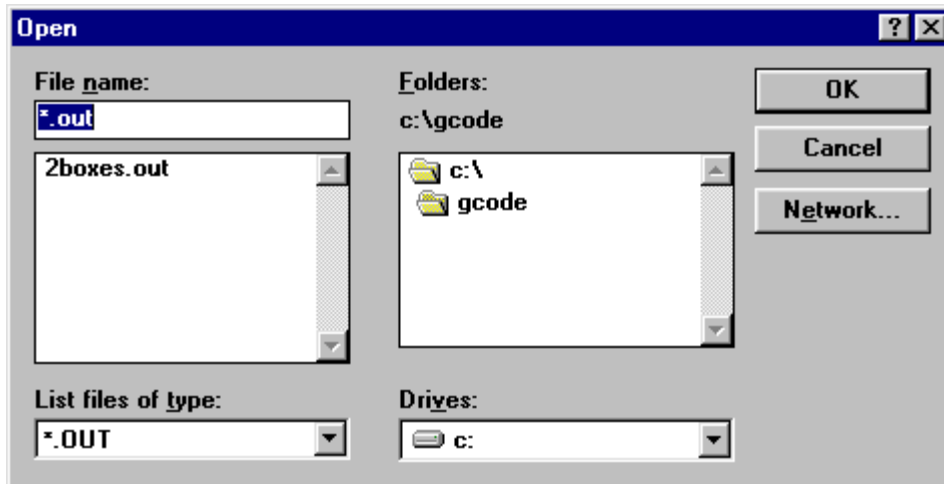
sends standalone programs written by the Interface (files with **.OUT** extensions) to your controller's memory. Once the file is downloaded, it can be run directly from your controller by simply pressing Start on the machine's display panel.

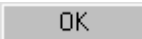


- Your controller can only run the most recently downloaded program. Once a new program has been downloaded, or if the controller's power has been turned off, any previously downloaded program is erased and must be downloaded again. This process takes only a few seconds.
- Standalone files are created for every program translated.

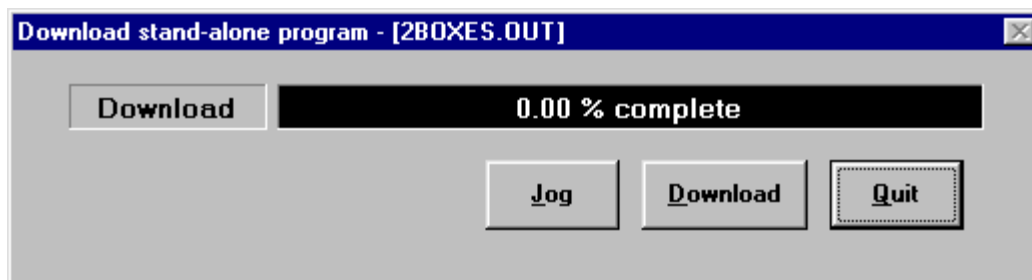
Downloading and Running a Standalone Program


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 

4. The  window opens:




5. Click  if necessary to manually position to your toolpath origin.

6. Click  to return to Download.

7. Click  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  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

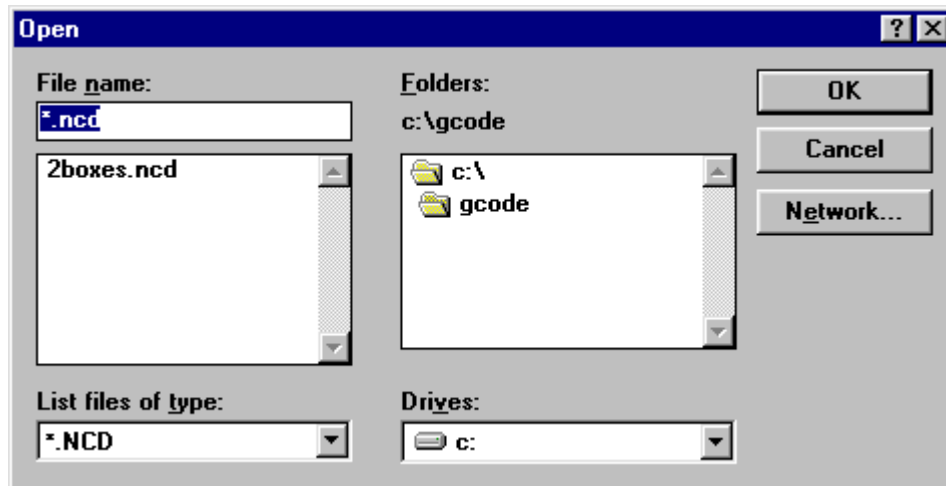
Preview

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.

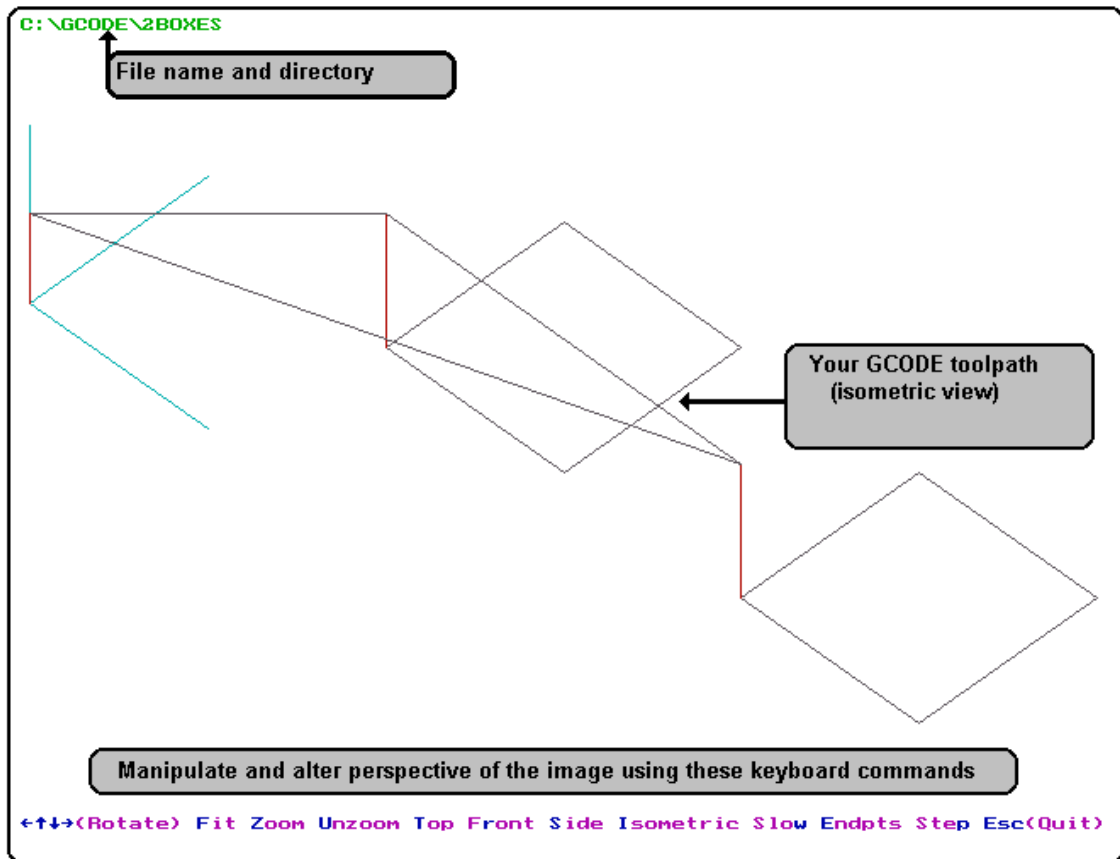
Previewing Your GCODE Toolpath:

- Click **Preview** from the main menu.
- An Open File dialog box appears.



- Select the file you'd like to Preview and click **OK**.
- The Preview of your file appears. Manipulate the image and view it from different perspectives using keyboard commands.
- 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
$\leftarrow\uparrow\rightarrow$(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

The following is a summary of GCODE commands recognized by the Techno GCODE Interface for DaVinci and C-Series Controllers. This summary is intended only as an abbreviated reference of basic GCODE commands and not as a guide for writing GCODE programs, for which we recommend reading a GCODE language manual.

1. Setup Commands

COMMAND	FORMAT	EXPLANATION
Absolute Mode	G90	Motion commands following this will all be relative to the origin.
Incremental Mode	G91	Motion commands following this will be relative to previous position of axes.
Absolute Zero Point	G92 X0Y0Z0	Sets an origin for all following absolute motion mode commands.
Dwell	G04/t	Specifies and initiates controller delay. 't' is variable time in seconds.
Set Feed Speed	F#	Indicates feed rate (speed) for all axes. # is rate in user specified units/second.
Turn Output On	M91n	Turns specified output on. n is the predefined symbol for the specified output.
Turn Output Off	M90n	Turns output off. n is specified output.

2. Routing Commands

COMMAND	FORMAT	EXPLANATION
Linear Motion	G1 X#Y#Z# F#	Moves axis in straight line at feed rate the number of units (#) specified away from previous axis position in G91 mode or from absolute position specified in G90 mode. F# is optional feed rate command. # is feed rate in units/min.
Rapid Move	G0 X#Y#Z# S#	Moves axes # of units specified at non-cutting (rapid) speed away from previous axis position in G91 mode or from absolute position specified in G90 mode. S# is optional rapid speed command. # is rate in units/min.
Clockwise Circular Motion	G2 X#Y#I#J# or G2 X#Z#I#K# or G2 Y#Z#J#K#	Draws an arc clockwise in the plane specified from current axis position to the end point (first set of coordinates) around a center point (second coordinates). This endpoint is relative to the start point in G91 and to absolute position specified in G90 mode. Center point is always relative to the start point.
Counterclockwise Circular Motion	G3 X#Y#I#J# or G3 X#Z#I#K# or G3 Y#Z#J#K#	Draws an arc counterclockwise in the plane specified from current axis position to the end point (first coordinates) around a center point (second coordinates). This endpoint is relative to the start point in G91 and to absolute position specified in G90 mode. The center point is always relative to the start point.

V. Troubleshooting

The Techno GCODE Interface includes a README file containing updated information about the software not included in this manual. This file may be able to answer some of your questions not answered by the Troubleshooting Guide.

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.

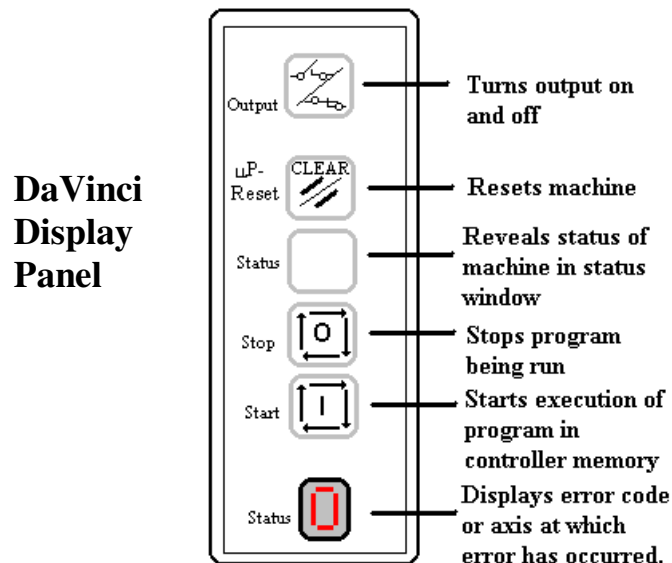
Symptom	Possible Cause	Remedy
No communication between controller and PC	<ul style="list-style-type: none"> a. COM port set improperly b. Controller connected to non-functional COM port c. Controller power off d. Connecting cable improperly attached e. Emergency stop switch depressed f. Wrong cable 	<ul style="list-style-type: none"> 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.	<ul style="list-style-type: none"> 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. 	<ul style="list-style-type: none"> 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

		<p>check machine parts including cables, cords, cutting tools and accessories to make sure they are clean and dry. All components should be inspected regularly.</p> <p>d. Contact Techno for support.</p>
No controller power	<p>a. Power switch off (it happens) b. Emergency stop engaged c. Power cord loose or disconnected d. Fuse needs replacement</p>	<p>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.</p>
Part produced is wrong size	<p>a. Scale factors improperly set. b. GCODE program error c. Incorrect cutter size d. Speeds too high for conditions</p>	<p>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.</p>

<p>Machine cuts at incorrect depth</p>	<ul style="list-style-type: none"> a. Z-axis scale factor improperly set b. Loose cutter c. Speeds too high for conditions d. Incorrect programmed depths 	<ul style="list-style-type: none"> a. In Configuration Defaults, check Z-axis scale factor to make sure it is set 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.
<p>Downloaded program doesn't run</p>	<p>.OUT file is too large</p>	<p>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.</p>

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.



■ 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
- = Unexpected carriage return received

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

- 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 displays 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

- 1 Step, 34
- 2boxes.ncd, 7-13
- Automatic spindle mode, 22
- . CFG files, 14
- Com port setting, 7,8,10,18
- Configure defaults, 15-24
 - difference from Setup, 15
 - explanation of parameters 16-24
- Controller setting, 7,8,10,19
- Controller error codes, 47
- Cutting speed, 8,18
 - during translation/execution, 31
- DaVinci display panel, 47
- Default configuration file, 15-16,32
- Default settings, 8
- Discard changes, 9,16
- Download, 37-38
- Endpoints of commands, 40
- Explanation of functions, 14-40
- File extensions, 14
- Files
 - downloading to controller, 37-38
 - extensions, 14
 - previewing toolpath, 11-12,39-40
 - running translated files, 21,35-36
 - running with offsets, 23
 - setting parameters, 7,10,32
 - translating/executing, 7-13,30
 - trial execute off, 21,33
 - trial executing, 20,33
- Fit, 40
- Front view, 40
- GCODE commands, 41-42
- GCODE Interface
 - installing, 5
 - menu map, 6
 - system requirements, 5
- GoTo position, 28
- Halting translation/execution, 13,33
 - from Run Translated Program, 36
- Home All, 11,27
- Home Before Run, 9,24,25
- Home position, 7
- Homing axes, 11,27
- Isometric view, 40
- Jog, 10,11,24,25-28
 - entering speeds, 26
 - from Download, 37
 - from Run Translated Program, 35
 - from Translate/Execute, 32
 - GoTo position, 28
 - homing axes 11,27
 - positioning axes, 11,25
 - saving positions as offsets, 24
 - step-size, 26-27
 - using directional keys, 11,26
 - zeroing counters, 27,28
- Main menu, 7,14-15
- Motor numbers, 8,16,25
- Motor resolution, 16-17
- . NCD files, 14
- Offsets, 9,22-24
 - running a file with, 22-24
- . OUT files, 14,21,29,33,37
- Override speeds, 8,19
- Pause mode, 9,22
- Pause options, 33
- Pausing the Interface, 13,33
 - from Run Translated Program, 36
- Plunge speed, 8,18
 - during translation/execution, 32
- Position display
 - Translate/Execute, 13,32
 - Jog, 11,25
- Practice lessons
 - jogged positions as offsets, 24
 - trial execute off, 21
 - trial executing a file, 20
 - using offsets, 23
- Quit, 34
- Preview, 39-40
 - from translate/execute, 34
 - previewing toolpaths, 11-12,39-40
 - using keyboard commands, 40
- Program command display, 13,31
- Rapid speed, 8,18

- maximum, 18
- Readme file, 43
- Resume, 34
- Retrieve file, 16,32
- Return, 34
- Return to 0, 9,22
- Rotate, 40
- . RUN files, 14,21,29,33,37
- Run Translated Program, 21,33,35-36
- Sample program, 7-13
- Save changes, 9,16
- Scale Factor, 8,16
 - formula for determining, 17
 - common values, 17
- Screw pitch, 16,17,18
- Setup, 10,20,21,23,24,32
 - and Configure defaults, 15,32
- Side view, 40
- Slow image movement, 40
- Spindle mode, 9,22
 - in Jog, 27
- Standalone programs, 14,21,29,33,37
- Start command, 12,33
- Step by step preview, 40
- Step-size, 26
 - entering specific values, 27
- Task completion display, 13,32
- Technical support, 43
- Title bars, 15,31
- Toolpath directory, 9,10,19
 - browsing, 19
- Toolpath origin, 10,20
- Top view, 40
- Translate/Execute, 10,13,29-35
 - explanation of features, 13,31-35
- Trial execute, 9,20,21,33
- Troubleshooting, 43-48
- Tutorial, 7-13
- Unzoom, 40
- X-Y configuration, 8,17
 - and jogging, 25
- Zero, 27,28
- Zoom, 40