

GCODE 2000

To print using Wordpad use page setup of Left 1.25" Right 1.25" Top 1" Bot 1"

Requires Windows 95/98 2000 or NT.

What Is Gcode, The Product?

Gcode is specifically designed for converting AutoCAD drawings into G-Code programs for use with any 2-D or 3-D machine such as Wire EDMs, Laser Etchers, Flame cutting, Milling , Lathes etc. Also Gcode can be used for creating individual complicated tool paths for 2-D and 3-D machines. Gcode is a low cost opportunity for small independent machine shops to convert their drawings into useful programs for their CNC and DNC equipment without the hassle of figuring out the geometry manually.

Simply draw the outline of the part on a dedicated layer.

Then DXF out the file to a directory .

Gcode can then load the file, process and plot it.

If it is not quite what was expected, simply edit the drawing and repeat the process.

New features:

A "full function editor" is provided up to 1,000,000 line capacity.

Flipping, flopping, rotating and scaling tools are provided.

Conversion from absolute to incremental or incremental to absolute.

Metric to inch and inch to metric conversion.

Support for curves in the Z K and R in all three planes. R goes negative above 180 Deg.

Tool travel distance to help with costing and time study.

Rectangular area used by the part for costing .

Move the largest object to bottom for flame cutting etc allows all interior cutting first.

File transfer both in an out is supported so this system can be hard wired to a machine tool.

The program can be configured to support any axis combination. The characters associated with a command can be changed at will. Even multiple characters can be used
Ie "RO"

Dxf out allows you to take existing Gcode files and read them back into Autocad.

Polylines,LWPolylines and Ellipse can now be read.

AutoCAD:

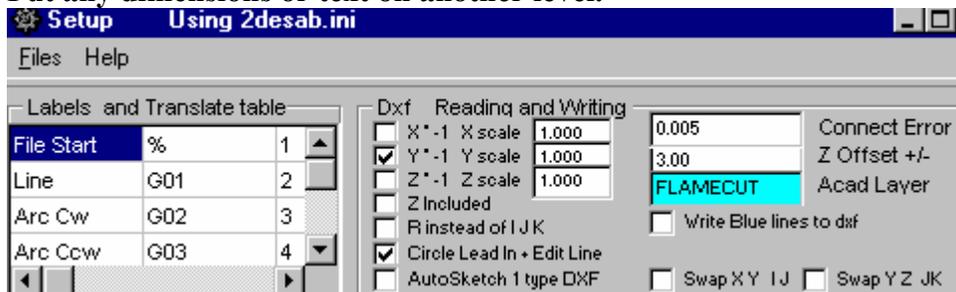
Before using AutoCAD to create the DXF file, check the default measuring system to make sure that angles are measured ccw from 3 o'clock. Also check that moving the cursor up and to the right from world 0,0 increases both X and Y values as positive numbers. These are the default states from a new AutoCAD installation.

Arc, Circle, Line, Polyline, Ellipse, Solid and Trace are the only entities that can be used on the drawing layer.

Draw the object full size on a layer the same name as the one used in the Gcode Setup Window (as shipped this was FLAMECUT see example window). Autocad by default draws on layer 0.

Placing an asterisk * in the Acad Layer box will read in all layers.

Put any dimensions or text on another level.



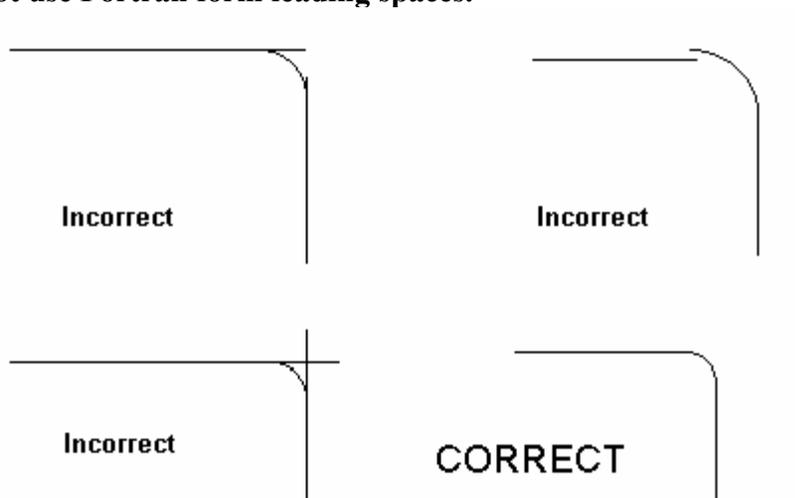
Setting X0 Y0 Z0

Pick a point that will become X0.000 Y0.000 Z0.000 using the commands AutoCAD / Menu / USC

If this is not done then World 0,0,0 will be used. This point will become the machine Zero reference

A good drawing practice is making sure all lines on a path connect. Any space over the specified "Connect Error" will result in Gcode beginning a new path. See examples on the next page.

Autosketch 1 type DXF is checked for Dxf files that only contain X Y data and do not use Fortran form leading spaces.



Autocad Continued:

With 3d drawings always draw the lead in line from 0,0,0 to the beginning of the part first, then continue drawing the tool path. This guarantees that Gcode will detect the correct starting point.

With 3d drawings Use set UCS X Y Z Axis Rotate only in 90 Degree increments extremely unpredictable results can happen if you draw on an inclined plane.

With 3d drawings explore them from different view points before committing them to a file

Drawings may be created in random order.

On any continuous path all lines must connect with no overlap or gap.

Use command "Snap" to prevent gaps and intersect lines.

Once the DXF file has been processed then use cut and paste to change the order of processing.

Use Swap to change the direction of travel.

For instance you might want to cut all the holes in an object before cutting the outside path.

Guaranteed cutting order:

Avoids having to use the Swap command.

Trace over the part shape from another level in the order which you want the part to be processed.

For circles trace from the top center around.

For lines and arcs trace in the order desired.

Do not worry about the fact that AutoCAD draws arcs counter-clockwise.

Polylines are another way to guarantee the cutting order if you create the line in the order that you want.

DXF OUT from Autocad

Select the command DXF OUT and enter the path to the directory desired, and a file name without an extension.

Press Enter to accept the default of 6 Decimal places.

Press ALT TAB or click on the task bar and change to the Gcode Program or start menu and run the Gcode Program.

Then select File Open and load the DXF file.

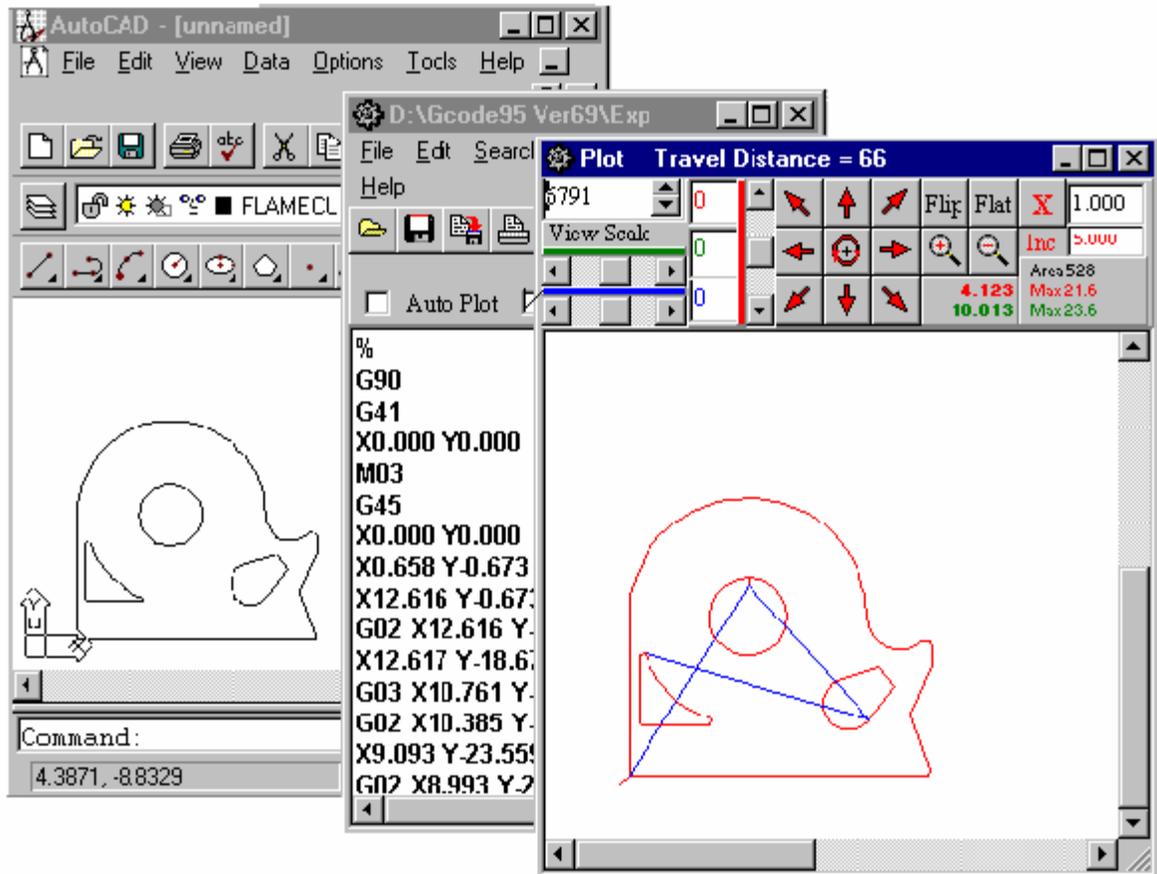
Drag and drop is also supported.

The extension DXF can also be associated with Gcode so that double clicking on any DXF file will open Gcode and process the file.

To associate a file extension with an application use **My computer,View,Options,File Types**.

Concurrent operation of AutoCAD and Gcode

This example shows both AutoCAD and Gcode open at the same time.



This example shows the programs in reduced size windows.

If you wish to work in full screen windows use "Alt Tab" or the Task bar to alternate between the two programs.

Quick Start:

Select: "File" - "Open" Use down arrow keys (highlight) file name: Exp.dxf

Double click or press Enter.

The file will then be loaded and processed. (SEE PICTURE OF WINDOW BELOW)

A small plot will appear in the right window.

The left window contains the Gcode, source file.

To redraw the entire part at a different point in the window:

Moving the mouse to the right window and clicking the left mouse button will re-plot the file at the point where the "mouse arrow" was positioned.

To redraw the part in single steps:

Pick another point in the right window and click the "right mouse button". Each click will plot one line of the "source file" and mark what the right window plotting is using in the source file. Clicking the "right mouse button" repeatedly will draw the part in single steps. Plotting one line at a time. A black cursor in the left window marks the next line to be plotted. One click after the plot is finished and the part will be erased. One more click will restart the drawing process.

Notice:

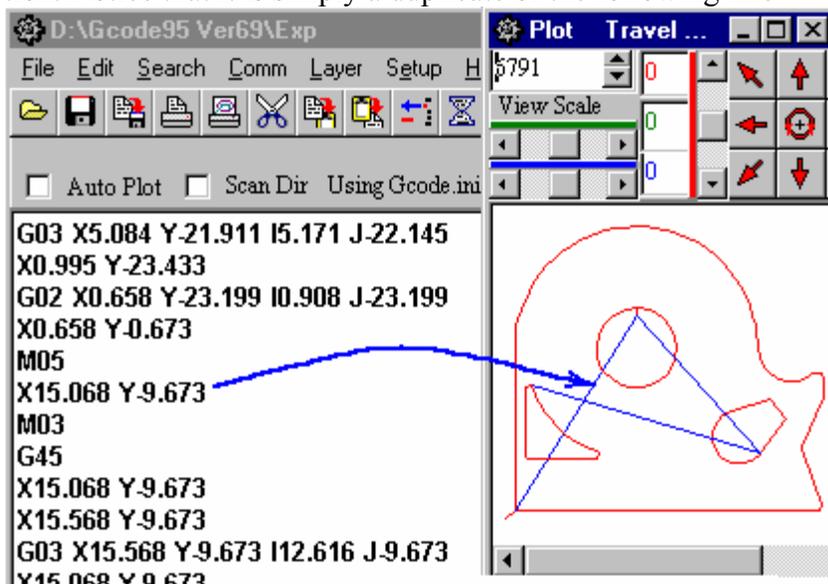
Red Line - is where the cutting will occur.

Blue Line - indicated a move between paths (no cutting will occur)..

This Move is automatically inserted if "Circle Lead in + Edit Line" is checked on the setup screen.

This is the cut off condition for moves between M03 and M05 or those statements defined by you.

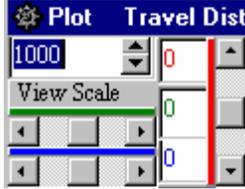
on the setup screen. The commands might be for coolant off coolant on. Although this line is added automatically it may need to be edited manually to place the tool in the desired tion. Notice that it is simply a duplicate of the following line



PLOT:

Scale - Used to control the size of the plot. Either clicking the arrows in the Spin Edit or entering a number changes the scale of the plot. The starting scale is calculated when a file is loaded.

Three Scroll Bars - Two horizontal and one vertical scroll bars rotate the object about X0.00 Y0.00 and Z0.00 by the amount specified in the Setup window "Angle Step". [typically 5 degrees]



First Box: Red Lettering - Display the vertical scroll bar angular position [Normally X axis]

Second Box: Green Lettering - Display the upper horizontal scroll bar angular position. [Normally Y axis]

Third Box: Blue Lettering - Display the lower horizontal scroll bar angular position [Normally Z axis]

These "BUTTONS" are used to actually modify the code.

The amount placed in the "Spin Edit" will modify the code by clicking the buttons to the left.



Warning all of these functions change the source file.

Clicking the Arrow buttons will move the program in the direction of the arrow by the **Inc** edit box amount. The center button will spin the program by the **Inc** edit box number of degrees. May be used for nesting odd shapes into existing material. All these buttons slide the program with respect to zero.

Clicking X Will multiply the program by the amount in the X edit box. As shown this is 1.000 The number may be greater or less than 0.000. This feature can be used to modify a generic part. For instance suppose you have drawn a one inch wrench. This would allow you to scale the wrench up to one and a half inches or down to nine sixteenth inches. Examples: 1.100 would increase the size by 10 % --- 0.900 would decrease the size by 10 %.

Clicking Flip or Flop turns the object over and changes the travel direction May be used for nesting odd shapes into existing material. Warning all of the above functions change the source file.

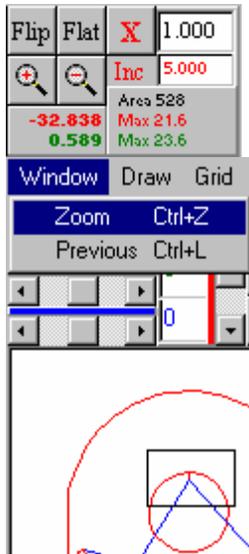
Flat Button

Clicking the **Flat** button returns all three axis to a perpendicular view.

Position below Zoom

-29.598
0.589

Position Displays the X Y Values in the same units as the source file.
Handy for testing if a component can be made from the available material.



Zoom + - buttons double or half the scale.

Zoom Draw a box around a selected part of a plot and re scale.

Previous Returns to the previous view.

Area is the rectangular area used by the shape

Max shows the largest distance traveled in that plane.

Pan

Pressing a ;Shift key and an arrow key causes the plot to Pan 100 pixels in the direction of the arrow. By combining large scale values and **Pan** (Shift and arrow keys) microscopic detail can be seen.

Travel Distance is the distance the tool travels to make the complete part.

Moves in all three axis are combined to make this number.

Often used as an aid to estimate production time or usage charges.

Communications:

By clicking "Comm" the Communications Window will open.

SEND TERMINAL :

The "Send Terminal" is the second window in the Communications Window. Click in the "Send Terminal" window and any data typed will be sent to the remote system when the enter key is pressed.

This is commonly used to initialize the remote system in preparation for receiving a file.

Then select "File" "Send Editor File" to transfer a copy of the main editors contents to the remote system.

If the output to tape punch checkbox has been checked then the right window will show a representation of the tape. (the letters EXP are visible as part of a readable header)

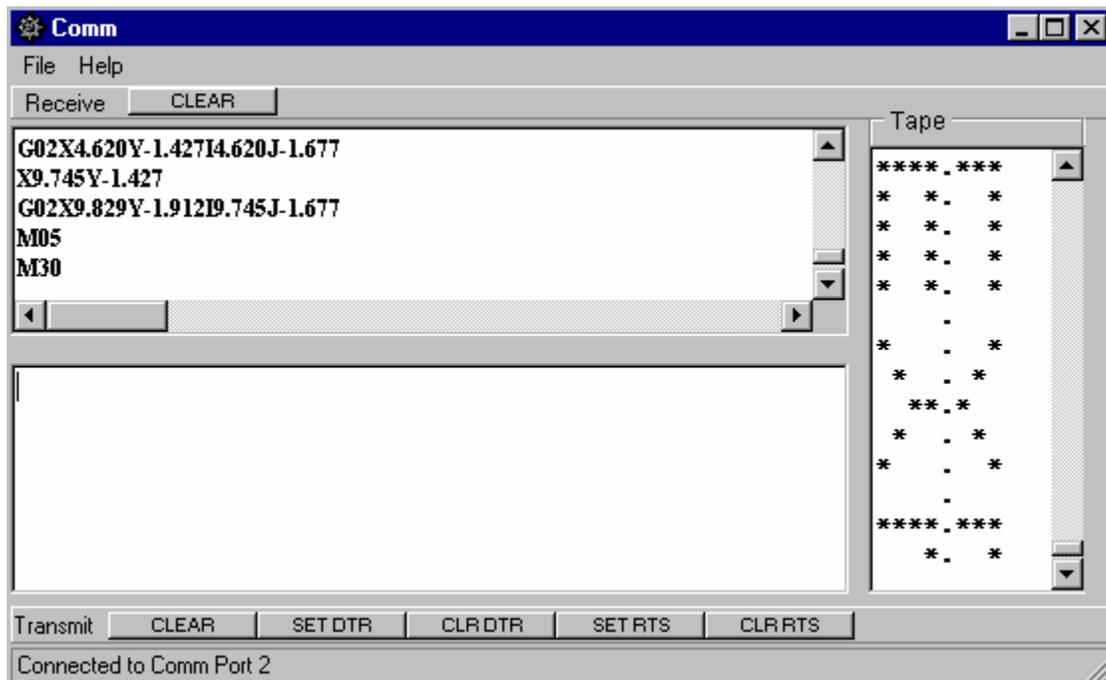
RECEIVE WINDOW:

The "Receive Window" is the upper window in the Communications Window.

Any text sent from a remote system will be displayed here.

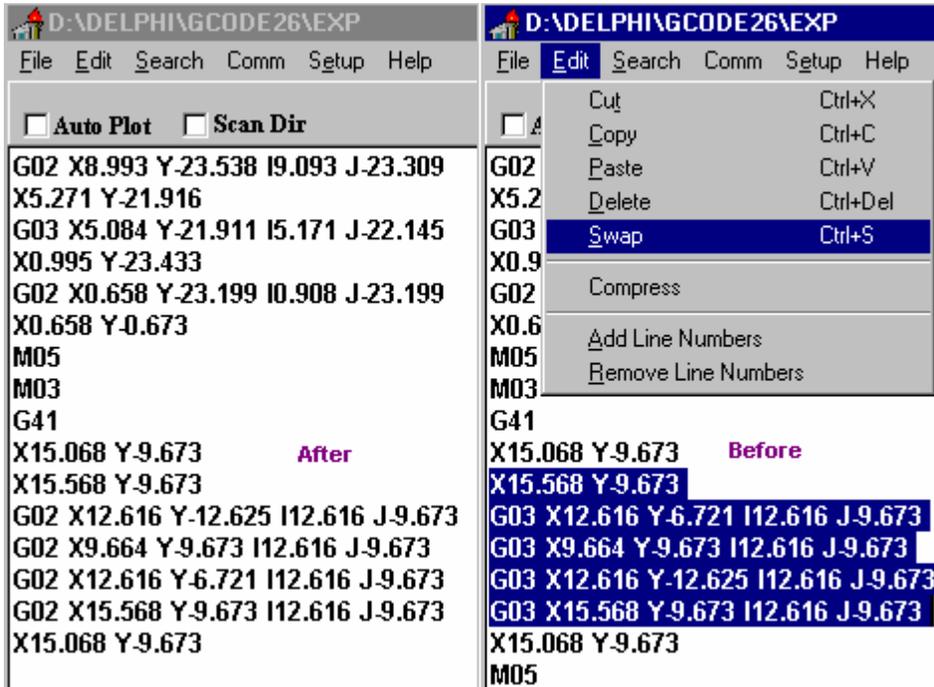
It can be transferred to the main editor by selecting "Copy Received To Editor."

See Ini for further details on communications settings.



Swap [under the Edit command]

Swap is used to change the direction of travel for a block of code. In the example below it is changing the direction of travel for a circle consisting of four quadrants. Note that the moves in and out are not highlighted and consequently will not be modified by swap.



Highlight the code that you want to change the direction of travel.

Select "Edit" "Swap"

The right hand display shows the file immediately before swapping.

The left hand display shows the file immediately after swapping.

A blank line is inserted at the end of a swapped block this helps to identify which area was manipulated. Use the "**Compress**" command to remove the blanks and duplicates.

Tip: Save your work before Swapping blocks as it quite easy to muddle your code.

Ini Files: [Defines machines relationship to Gcode]

INI FILES: [Defines the machines relationship to Gcode]

Setup screen to create an INI file

File Start	%	Line
		1
Line		2
Arc Cw	G02	3
Arc Ccw	G03	4

Dxf Reading and Writing

X*-1 X scale 1.000
 Y*-1 Y scale 1.000
 Z*-1 Z scale 1.000
 Z Included
 R instead of IJK
 Circle Lead In + Edit Line
 AutoSketch 1 Dxf
 DesignCad Dxf

Connect Error

0.005
5.0
FLAMECUT Acad Layer * All Layers
3 Dec Places 3.00 Z Offset +/-
 Write Blue lines to dxf
 Corel /TurboCad Dxf
 Swap X Y IJ
 Swap Y Z JK

Options

Incremental Type IJ
 Disable all Modifiers
 Make IJ Circles 4 Quadrants
 Strip spaces on Transmit
 CR on Transmit
 CR LF on Transmit
 Want CR on Receive
 Strip Blank Lines On Receive
 Enable Translate output file On Transmit
-1 Transmit First Ascii Char if > -1
-1 Transmit Last Ascii Char if > -1
Font Arial 1 Line Num Inc
0 Line Delay Ms Make Plot Match Machine
0 Char Delay Ms Flat X Angle
2000 Scan Time Ms Flat Y Angle
150 Single Step Ms Flat Z Angle

Punched Tape

Output to tape Punch
Head /Tail Len 50
Ascii Char 0
 Readable Header
 Track 8 Parity None
 Track 8 Parity Even
 Track 8 Parity Odd

Circles/Angles

5 Cir Res 1 = Best
5 Inc Angle

Initial Working Directory

Using the "Setup" window create and save your own INI file:

Selecting: "Setup", "Edit or Create INI "File", "Save ini" Enables the machine set up to be saved as a file name of your choice.

Selecting: "Setup", "Use new Machine Tool INI" and picking an INI file - Will retrieve a previously created machine INI files.

Tip: In creating a new INI file leave the setup window open making changes and repeatedly load a DXF file until the desired result is obtained.

Try to use meaningful file names such as BridgeportBtc.ini

Ini Files: continued [Defines machines relationship to Gcode]

Common steps to go through when creating an INI file for a new machine.

Use the commands Setup, Edit Or Create INI, Files Save INI and save the new file as an abbreviation of the machine name.

Type in a short program that you know works maybe a rectangle with one curved end.

Most machines give a few simple examples.

Try to plot it by clicking in the right hand Plot window. Assuming the result is not correct then begin to change INI settings until it is. The two areas to work in are Labels and Plotting.

Suppose the part plots correctly but is rotated 90 degrees then you must interchange the X and Y axis. By Checking the swap XY IJ box.

Four sample INI files Named X+RightY+Down.INI and similar names show the four possible axis direction combinations. One of these will probably form a good basis from which to build on.

One exception is with 3d INI where Z & K are to be used. These files will not work correctly if you interchange the X and Y axis. The problem is because I K arcs are not treated the same as JK arcs and if the axis labels have been changed it is impossible to decide which method to use for drawing the arc. Since the plot can be easily rotated please leave the axis labels alone If the plot has the correct axis orientation but is upside down then use the Plotting X * -1 or Y * -1 to correct this. If arcs are not drawn correctly try interchanging G02 and G03 in the Labels area.

Also check to see if you should have used G2 or G3 instead.

Dxf

Once the small sample file is plotting correctly move on to AutoCAD and draw exactly the same part. DXF the file out and attempt to load it into Gcode Using the commands File, Open

Assuming it is not correct then begin to change settings in the setup DXF area.

Start with the Acad Layer name, this is case sensitive * = all layers.

If the Z axis is to be included check the box Z Included.

Then experiment with X Y And Z being multiplied by -1.

Note: The left hand column of Labels section are only comments, so feel free to add any relevant text to help you in the future. Labels that are of no use to your machine can be replaced with other text. It is generally not a good idea to totally blank a field. Use instead characters that are used to create comments. I.e. (). See page twelve for more on Labels.

DEFINING THE SETUP:

LABLES:

Scroll Box Middle Column - Contains all the characters that will be added to a file as it is converted from DXF to Gcode.

Scroll Box Left Column - Contains comments (Change to suit or to be meaningful).

3DMILL.INI **Often** 2DTORCH.INI

- 1.File.Start.....%.....%.....%....First line of code usually a single character.
- 2.Line.....G01...G1.....Placed in front of a line without curve data if not blank
- 3.Arc.Cw.....G02...G2...G02....Curve Clockwise
- 4.Arc.Ccw.....G03...G3...G03....Curve counterclockwise
- 5.Krf.CmpLeft..G41.....G41....Third line of a program could be anything
- 6.Krf.CmpRight..G42.....G42....Third line if the direction of travel is swapped
- 7.Absolute.....G90.....G90....Second line of code
- 8.Incremental.....G91.....G91...Second line of code if converted to incremental
- 9.Start.Cut.....M08.....M03....Beginning of a series of connected moves.
- 10.Stop.Cut.....G00...G0...M05....End of a series of connected moves
- 11.File.End.....M30.....M30....Last line of code.
- 12.Lead.In.....M07.....G45....Placed on the line after start cut
- 13.N.Letter.....N.....N.....Placed on lines Line numbers
- 14.R.Letter.....R.....R.....Radius Instead of IJ R- xxx > 180 deg
- 15.I.Letter.....I.....I.....Distance to arc center in X axis
- 16.J.Letter.....J.....J.....Distance to arc center in Y axis
- 17.K.Letter.....K.....K.....Distance to arc center in Z axis
- 18.X.letter.....X.....X.....Axis X can be interchanged except 3d
- 19.Y.Letter.....Y.....Y.....Axis Y can be interchanged except 3d
- 20.Z.Letter.....Z.....Z.....Axis Z can be interchanged except 3d
- 21.XY.Plane.....G17.....Inserted in the line before an arc if not blank
- 22.XZ.Plane.....G18.....Inserted in the line before an arc if not blank
- 23.YZ.Plane.....G19.....Inserted in the line before an arc if not blank
- 24.Line.2.....Line two Any text can be used or left blank
- 25.Line.3.....Or type a complete file path to insert a text file
- 26.Line.4.....Ie: A:\HeadingFile.txt
- 27.Line.5.....Ditto
- 28.Rapid.....G00.....Placed on lines before start cut if not blank
- 29.....From.....To.....Headings for translate table
- 30,70.....Any thing in the from column is translated to the to column
- 71,72.....Comment start and finish ususally () if not blank.
- 75,79.....Lines added to the end of the file, anything you want. If not blank
Or a File path
- 90.....Maximum R value for radius if not blank.

For an example of the contents of the To column See 3dbandit.ini

Notice the difference between 3Dmill.ini and 2dTorch.ini. Some lines are blank and others are different commands. The letters for axis may be changed so that X could change to Y **except in**

3D Ini Files

If your machine has Axis A B C D E instead of X Y Z I J, then simply substitute the letters.

Line: G01 could be left blank depending on the machine's need.

Tip: Load and compare the sample INI files provided or using Windows Notepad to print them out for comparison.

DXF:

X*-1 Y*-1 Z*-1 - Boxes are provided for Multiplying X Y Z values by -1 as they are processed from the dxf file. If the box is checked, the values are negated.

X Y Z Scale Multiply incoming values by this. Typically X is multiplied by 2.00 for a Lathe since many lathes program with X as the diameter of the part but you would draw X as the Radius.

R instead of IJK Substitutes the letter R in place of IJK to express a tangential radius. Arcs greater than 180 degrees are expressed as R-xxx.xxx

Z Included Box - a checked box allows the inclusion of the Z axis from Dxf Files.

Circle Lead In + Edit Line - a checked box adds code for a lead in move inside all circles and a line of code ready to edit for the lead in move between unconnected paths. Notice it is simply a duplicate of the following line.

AutoSketch 1 Type DXF For reading in DXF files that only contain XY data and do not have Fortran format leading spaces.

SwapXY IJ Rather than edit the Labels section to change the letter order this does it for you. Used to make the Axis of the machine match your drawing.

Swap YZ JK See above. Typically swapped for a lathe.

Write Blue lines to dxf When a file is saved as type DXF then the lines plotted in blue between cut off and cut on are also included in the file. Note: These lines were typically not present when the file was imported from the original drawing as they are usually lead in/out moves.

Connect Error - This is the allowable space between the end of one line and the beginning of the next. Any lines with a space above this value will be considered to be the beginning of a new path.

Z Offset +/- - A positive or negative value added to all Z Values processed from the dxf file. This number is added to all Z axis values taken from the drawing. For example on a milling machine, it could be used to extend the quill a specific distance.

Acad Layer - The name of the AutoCAD layer to extract the Gcode data. Setting Acad Layer to "0" will use the default layer.

Decimal Places - The number of digits after the decimal point. Caution do not use more than the machine can handle. (Usually 3 or 4) By default up to 6 digits can be used in front of the decimal place.

BAUD DATA PARITY STOP BITS FLOW : Set to match the machine connected to this computer. Your machine manual should give the settings. A common setting is 9600 Baud, 8 Data, No Parity, One Stop Bit, and Flow control None. Set **Comm buffer** size above max file size.

LATHE PLOTTING

X Div 2 I Div 2 If the output is bound for a lathe and DXF Xscale is 2.000 then check these to cause Gcode to plot correctly.

Vectors

5.0 Vector Error Deg Line comp

This feature is used to reduce the size of a file typically that only contains XYZ data such as those produced by scanning If checked then in the above example all vector changes less than 5 degrees will be thrown out. (Not recommended for dxf files containing arcs and circles)

OPTIONS:

Incremental Type I J -

Two methods are used to define the center point of a circle. One is where the move is expressed as an incremental move from the last Absolute position, another is True absolute from 000. Select the appropriate one.

Note : Fanuc is usually Incremental.

Strip spaces Transmit - As the file is transmitted any spaces between characters are removed.

Tip : This can cause problems when code uses spaces specifically i.e. engraving commands.

CR on Transmit - A checked box will send carriage return characters at the end of each line.

CR LF on Transmit - A checked box will send carriage return and line feed characters at the end of each line.

Want CR on Receive - Carriage return characters will be added to received lines.

Tip : If received code is double spaced UN-check this and check Strip Blank Lines on Receive.

Strip Blank Lines on Receive - As the file is copied to the editor the compress routine will be called. Note: This does not do exactly what it says so blank lines may be visible in the receive window, but will be removed as the file is copied to the editor.

Line Delay Ms - A time in 1/1000 seconds to set delay after each *line* is sent. Used to allow slow machines to process the data as it is sent. Usually Zero.

Char Delay Ms - A time in 1/1000 seconds set to delay after each *character* is sent. Used to allow slow machines to process the data as it is sent. Usually Zero.

Single Step Ms - A delay in 1/1000 seconds between 125 and 1000. Used when the right mouse button is held down to single step plot a file. The mouse cursor must be in the right hand plot window for single step to be operational. If more than one line is plotted at a time then increase this value.

Flat XYZ Angle - The graph viewing angle to which the plot will move when the flat button is pressed. This can be used to make the plot orientation match that of the machine. For example if the code is correct and the plot is ok but upside down then this is the tool to fix it.

Font- Uses to select the font for the editor. Color will always default to black.

Line Num Inc - The amount by which line numbers will increment if they are added.

COM PORT: [Either]Com 1 - Select an available one to connect to the machine. Note: The Mouse usually uses Com 1.

Com 2 to 4 - are normally used to connect to the machine.

DATA: [Either]

7 Data - Used when a machine needs a character setting of *seven* data bits.

8 Data - Used when a machine needs a character setting of *eight* data bits

PLOTTING:

X*-1 Y*-1 Z*-1 - Options are provided to negate X, Y, and Z to show the same part in the same orientation as the machine. Example: UN-checking X and Y will invert the part.

Show XYZ - When checked adds a direction indicator in the lower left corner of the plot to show the angler position of each axis. Used to help keep up with your 3d view point.

PUNCHED TAPE:

Output to tape Punch - Check if using the serial port for ASCII punched paper tape output.

Head/Tail - Number of characters punched in the tape beginning and end.

Ascii - ASCII character punched in the Head and Tail of the tape.

Track 8 Party None - If box is checked then no holes are punched in track 8.

Track 8 Party Even - If box is checked then holes are punched in track 8 when the sum of track one through seven is a *even* number.

Track 8 Party Odd - If box is checked then holes are punched in track 8 when the sum of track one through seven is a *odd* number.

Readable Header - If box is checked the name of the file will be punched in the Header of the tape in a dot matrix format. Use with caution it can confuse some tape readers.

Tip: Normally the LF Line feed on transmit is not selected with tape punches.

CIRCLES / ANGLES:

Circle Res - Defines the resolution of circles as viewed on screen. The lower the value the better

the resolution, but more time is taken with higher resolutions, similar in operation to AutoCAD.

This value also effects the accuracy with which DXF files save curved data, 1 is the most accurate but creates large files.

Angle Step - The number effects the degrees a part is rotated each step when rotating the part.

Note the **Flat** button returns to a perpendicular view.

INITIAL WORKING DIRECTORY: Can contain a path to the normal source of files.

If you wish to store files in another directory other than that used by Gcode, specify another path in the "Initial Working Directory". [Example: C:\Lathe]

Tips: **Temporary changes can be made to the Setup window without saving the INI file to observe the effect before committing the settings to a file .Use this method to add a new machine tool with different axis directions to those supplied. [See the files Mill.ini Lathe.ini.] Once developed save the ini file with a unique descriptive name. All ini files should be placed in the same directory as Gcode.**

Use only uppercase characters except in comments. Avoid comments until a file is complete. "G" functions such as G03 is not treated the same as G3 [see Setup Labels] On the setup screen define which of the two to use.[See page 12]

All "G" and "M" functions statements must match exactly and are case sensitive.

The system is based on using absolute G90 programming. All Dxf translations generate G90 code. Limited support for incremental G91 is provided. See Convert to Absolute.

Commands:

File:

- ;New - Clears the editor (left window) for a new file.
- ;Open - Opens a file dialog box to enable loading another file.
- ;Save - Saves the file as it is named.
- ;Save As - Enables file to be saved with the option of using another file name.
- Save As Dxf- Saves in dxf format for reading back into Autocad.
- ;Print Text - Prints contents of editor.
- ;Print Window - Prints the right window and border.
- ;Print Plot - Prints the part at the present scale.
- Print Setup - Used for the selection and setup of printers.

Edit:

- Cut - Removes the highlighted text and places in the clipboard.
- Copy - Copies the highlighted text to the clipboard.
- ; Paste - Places the contents of the clipboard at the point of the cursor.
- ; Delete - Removes the highlighted text.
- ; Swap - Changes the travel direction (cutting path) of the highlighted text.
- ; Compress - Removes duplicate lines and blanks.
- ; Add Line Numbers - Displays the line numbers in the text Nxx.
- ; Remove Line Numbers - Removes the line numbers.
- ; Convert to Incremental G91
- Convert to Absolute G90
- Inch to Metric Multiplies by 25.4
- Metric to Inch Divides by 1/25.4

Search:

- ; Find - Opens a dialog box to define text to be found.
- ; Find Next - Used to find the next occurrence of the text used for the search.
- ; Replace - Opens a dialog box to list search text and give the new text to replace searched text.

Comm: [Communications] Use to send Rs232 serial data to remote systems.

;Use Setup to change Ports Baud rates etc. See your machine manual for cable diagrams.

File:

- ;Send Editor File - Used to transmit a file via a serial port to a remote machine.
- ;Receive From Remote - Used to receive a file from a remote machine.
- ;Clear - Clears the transmit and receive buffers of text.
- ;Copy Received To Editor - Used to copy transferred files to editor.

Setup: ;Opens a dialog box to create an ini files. **Ini** files are used to define the unique way that each machine tool interprets Gcode. Defines the axis relationships to one another. The ASCII characters that are used with the numeric data, and communications settings.

Help: List the help topics as AutoCAD, Cautions, Commands, Ini Files, Overview, Plot, Quick Start, Receive, and Transmit. Any underlined text in Help will list more information.

Window

Zoom - Allows a rectangular zoom box.

Previous - Returns to the previous view, only stores on level.

Draw

Line Arc Circ Start code Stop code.

Grid

1/10 1/100 1/1000 ¼ 1/8 1/16 - Preset scales for drawing.

Pan

Up Down Left Right - Same as shift and arrow keys to pan the view.

Print

Prints the window with border.

External Links.

Drag and drop is supported so that a Gcode text file or DXF file may be dropped onto the Icon on the desktop or open edit window.

The extension DXF can also be associated with Gcode so that double clicking on any DXF file will open Gcode and process the file.

To associate a file extension with an application use My computer, View, Options, File Types.

The program can also be started with a parameter after the program name. IE Gcode2000 Exp.dxf

LATHE

Support for lathes is provided in three areas

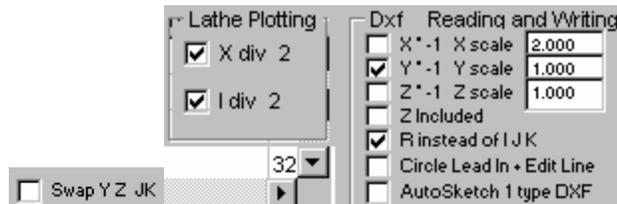
Dxf reading and writing, Lathe plotting and swap YZ

These allow you to draw a part from the centerline.

When it is imported from the dxf file all X values are multiplied by 2.000.

For the benefit of Gcode plot X and I are divided by 2.

The Y and Z axis letters can be swapped since many lathes use X I Z K as axis.



DRAWING

From The (Draw) menu the following items will show.

Line.

Clockwise Arc.

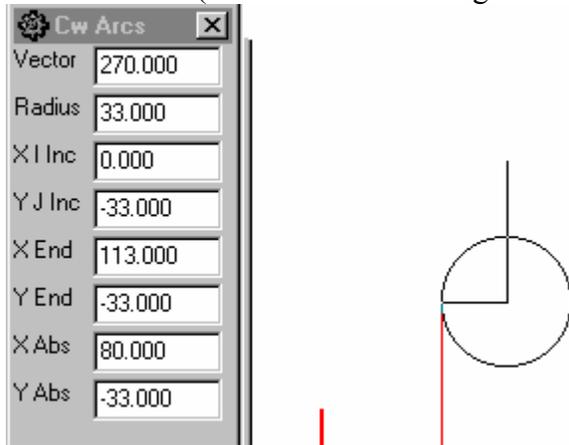
Counterclockwise Arc.

Clockwise Circle.

Counter Clockwise Circle.

Stop Cut M05 (or the command designated in the INI file).

Start Cut M03 (or the command designated in the INI file).



To draw a line.

Click on line then move the mouse pointer into the window and left click at each line end.

Finally Right Click to terminate drawing or press Esc to end.

To draw an Arc.

Click on Arc then move the mouse pointer into the window and position the cursor at the center point of the arc, then left click.

Now move to then ending point of the arc and click.

Note : Arcs and lines always start from the last point drawn.

It is not my intention to try to create a substitute for Autocad. These drawing tools are simple but do provide a means to create a simple part or add a missing hole.

To draw a Circle.

Click on Circle then move the mouse pointer into the window and position the cursor at the center point of the circle then left click.

Now move out to the desired radius of the circle and click.

Note : Circles are a separate entity and can be drawn at any point.

The code generated by drawing will depend on settings in the INI file.

For example if circles have a lead in or if "R" is used instead of "I J".

See (INI) for further details on commands settings.

This tool is provided as a simple way to draw a very elementary shape and is not intended to replace AutoCAD.

Grid

Six predefined grid settings are provided.

1/10 1/100 1/1000

¼ 1/8 1/16

Selecting any one of these will adjust the scale proportionally.

Any desired increment can be achieved with a suitable scale.

Just remember the base scale of

1000 = 1.000 Units per increment of mouse movement.

10000 = 0.100 Units and so on.

Draw

Line

Clockwise Arc

Counterclockwise Arc

Clockwise Circle

Counterclockwise Circle

Stop Cut

Start Cut

All drawing is done with the mouse.

Grid

1/10

1/100

1/1000

¼

1/8

1/16

Grid simply presets the scale.

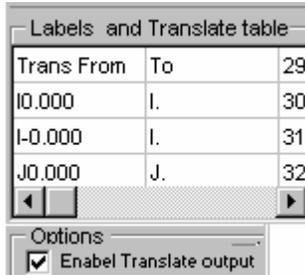
TRANSLATE

If checked **Translate on file load** will process the incoming file through the translate table.

This can be used to change the sign of value **J to J-** and **J-- to J** will change the sign of all J values in the file. Note the warning below about this being a one way trip.

If checked **Enable translate output file on transmit.**

In the Labels and Translate table from line 30 on anything found in the left column will be translated to what is in the right column.



For example :

From To Line

J0.000 J. 30

J-0.000 J. 31

K99 M66 32

H100 33 Blank

Note this is a one way trip so if you read a translated file back it may not.

Plot correctly.

The main purpose of this is to allow support for non standard machines such as Bandit controls.

For an example of its use see the 3dbandit.ini file.

This does not effect the contents of the editor it merely translates the code on the way to the remote machine, or if the Check box **Don't transmit show output in lower window** is checked then the file is just displayed for your review.

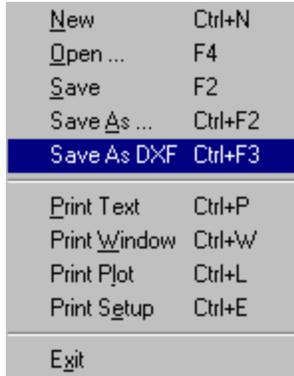
SAVE AS DXF

Data can be saved as a dxf file.

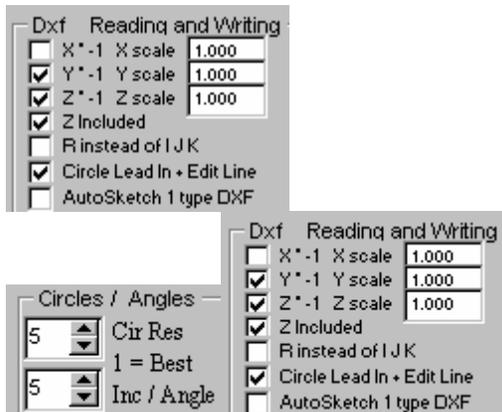
Just make sure the file name extension is DXF

This allows an existing Gcode file to be converted back to an Autocad drawing.

Note : Circles and arcs are drawn as a series of straight lines.



Using Autocad it is a simple matter to convert them back to arcs and circles if required.



The resolution of the circles and arcs is controlled by the Circles/Angles setting on the Setup Screen.

Increments as small as 1/1000 can be achieved with a setting of 1.

But remember the file size of the will be considerably larger.

If Z Included is checked then Z axis data will be read in.

Z Axis data is always written out even if it is zero.

This was intended to provide a simple means of creating a drawing from an existing Gcodefile.

PLOTTING

When the cursor is in the left window a click of the left mouse button will re plot the entire file at the cursor position. Right clicking the mouse will single step through the file.

Highlighted text If text is highlighted in the left window it will appear black in the right window.

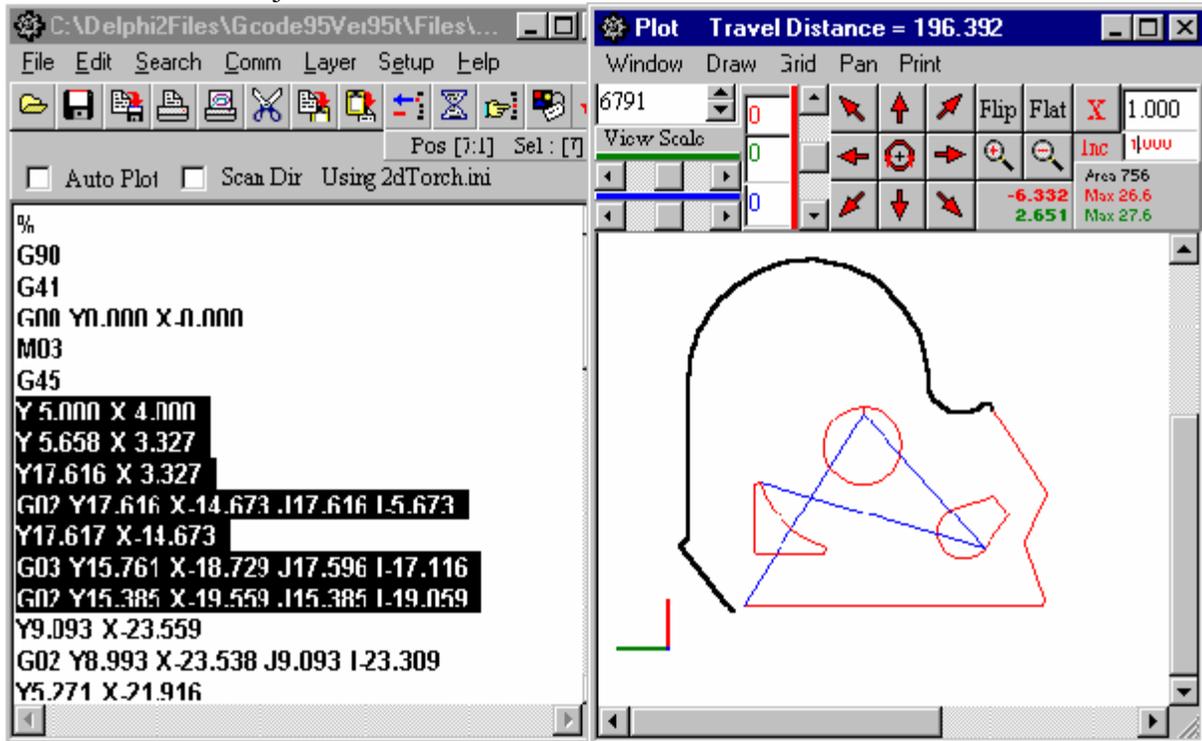
The left window will also show how many lines are selected as Sel[6] .

Any action performed in the right window such as rotation or moving will only be done to the highlighted section of code. If nothing is highlighted the action is done to the entire file.

Using this technique it is possible to relocate a block of code. Note all moves will be the Inc amount which defaults to 5.000 This example shows a highlighted portion of the file Exp. Notice that it has been relocated to the upper left in Increments of 1.000 and the upper left arrow has been pressed.

Sel: [7] indicates that seven lines of code are selected. After performing a move the Black highlight will disappear but the Sel:[7] will remain. To clear a selection simply click in the left window.

If the X 1.000 had been changed to 2.000 and X had been pressed then this block of code would have doubled in size. Many different things can be done with these tools. For example if you need multiple copies of a shape use cut and paste on the text file. Then highlight one object and slide it over the other to the desired position. Flip or Flop will invert the selected object.



Addendum to options on setup screen

Line 13 in INI file added for Rapid travel between cuts G00 can be left blank.

Line numbers increment by a programmable amount See setup screen Options Line Num Inc 1 to 20.

'R' For radius is now supported. In the Setup Screen Dxf area Use R instead of IJK can be selected. Note this also checks make IJ circles four quadrants this allows objects to be rotated.

'R' will not work correctly if you have swapped X to Y to rotate the shape 90' and are performing true 3D contouring with curves in the Z K axis. For this to work correctly you must rotate the drawing 90 degrees before DXFing it out of Autocad.

R when used assumes that it is making a tangential connection to the preceding line. If you have shapes like (**Mickey Mouse Ears, Arcs of more than 180 degrees**) Then R will appear as a negative number. Most machines will interpret this correctly. If not then you must fall back to IJK commands, or use the translate on transmit to change R-values accordingly.

Misc Menu item

Misc	Search	Comm	Setup	Help
Move starting point Forward				
Add Semicolons to line end				
Remove Semicolons from line end				
Sort Objects Ascending				Ctrl+A
Sort Objects Descending				Ctrl+D
Largest Object to bottom Only				Ctrl+B
Line Order N G M X Y Z I J K R				Ctrl+O
Translate Using table from setup window				
Edit Translate table				

Move starting point forward.

Highlite a complete tool path where the beginning meets the end and click this option to move the staring point forward one line. (Used to select a more suitable start)

Add Remove semi colons.

Just as it says. (Some machines require them)

Sort Objects ascending.

The entire file will be sorted to put the smallest complete paths (complete tool path where the beginning meets the end) at the top. Often used in flamecutting to ensure that the perimeter cuts are made last. Also useful when cutting lettering as it is usually important for the middle of letters to be cut first.

Sort Objects descending.

Exact opposite of above.

Largest Object to bottom only.

Like sort ascending except only the longest path is moved.

Line Order N G M X Y Z I J K R

If letters have been swapped this puts line in order.

Translate using table from setup window.

This the second implementation of the translate table.

This method translates the editor file using the table.

One use is to edit a file that was transmitted from the machine tool into Gcode.

The operator might have blocked out some lines using the / character. All of these could be removed at once using the translate label. It is rather like doing up to 30 search and replace commands at once, by placing the slash in the **from** column and a blank space in the **to** column

Edit Translate table.

A shortcut to the table to refine your commands. Starts at line 30 In labels on setup screen.

Command line operation.

Gcode can be operated from the command line.

Gcode2000.exe Filetoload Filetosave inifileouse.ini

Filetosave and inifileouse are optional.

If both Filetoload and Filetosave are present then processing will proceed and gcode will close.

Offset

Used to move the tool path inside or outside the **Highlited code** by the tool radius.

It is best to Highlite a block of code where the start and finish are at the same coordinates.

But as little as a single line can be offset.

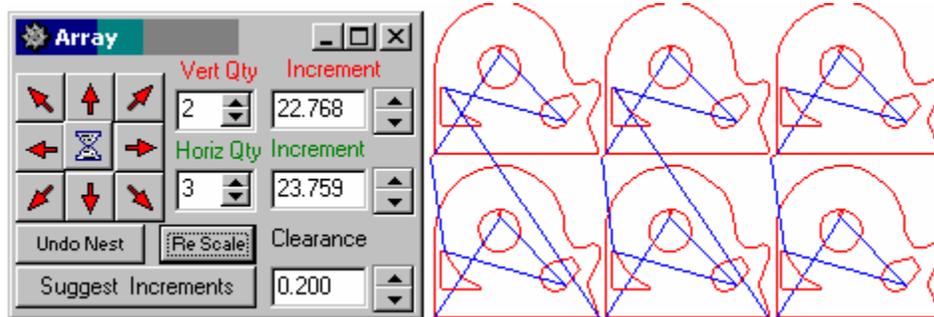
Pocket milling can be done by repeated offsets.

Islands: Offset out from the island, and in from the perimeter until they overlap.

With the cursor in the offset window clicking the right mouse button will singlestep offset file.

Pull down menus all you to place the new code back into the main file.

Note: A curved corner can not be offset to the inside more than its radius.



Array

Used to reproduce multiple parts across a sheet.

Once you have a working Gcode file select Array.

Suggest increments scans the file and attempts to find the size of the part.

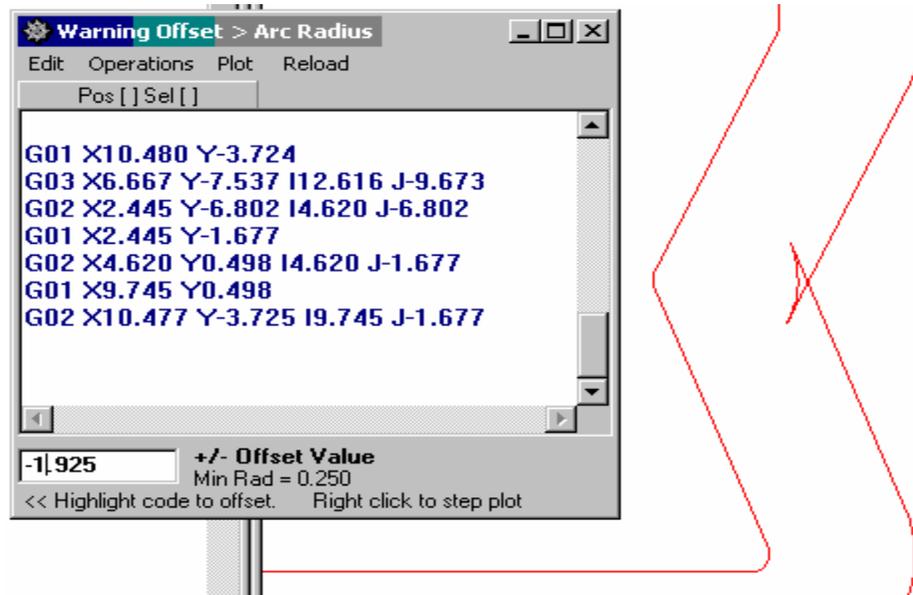
(Arcs projecting beyond the part are not taken into account)

Select the Horizontal and vertical quantity.

Then click on the arrow for the direction of the array.

Note: A curved corner can not be offset to the inside more than its radius.

A butterfly shape will appear if you do this , see below



CNC G-CODES Typical on a Flamecutting machine.

Red Codes are used by this program

CODE FUNCTION

G00 Rapid Traverse

G01 Linear Interpolation (Feed)

G02 Circular Interpolation Clockwise

G03 Circular Interpolation Counterclockwise

G04 Dwell

G40 Kerf Compensation Off

G41 Kerf Compensation Left

G42 Kerf Compensation Right

G45 Lead in to Kerf Compensation part

G70 Input in Inch

G71 Input in Millimeter

G90 Absolute Programming

G91 Incremental Programming

G97 Set Program repeat pointer

G98 Initiate program repeat at G97

M00 Program stop

M02 End of program

M03 Cutting on

M05 Cutting off (Used to swap from blue to red plotting)

M08 Plate marker on

M09 Plate marker off

M30 End of program

Page 27

CNC G-CODES Typical on a Milling machine.
Red Codes are used by this program

CODE FUNCTION

G0 Rapid Traverse
G1 Linear Interpolation (Feed)
G2 Circular Interpolation Clockwise
G3 Circular Interpolation Counterclockwise
G4 Dwell
G8 Modal Deceleration Override Off
G9 Modal Deceleration Override On
G12 Helical Interpolation CW
G13 Helical Interpolation CCW
G17 XY Plane Selection
G18 ZX Plane Selection
G19 YZ Plane Selection
G22 Circular Interpolation, Fillet Input CW
G23 Circular Interpolation, Fillet Input CCW
G30 Mirror Image Off
G31 Mirror Image X On
G32 Mirror Image Y On
G40 Cutter Diameter Offset Off
G41 Cutter Compensation Left
G42 Cutter Compensation Right
G44 Cutter Compensation, Normal Feedrate
G45 Cutter Compensation, Modify Feedrate
G48 Corner Rounding in Cutter Comp Off
G49 Corner Rounding in Cutter Cornp On
G70 Input in Inch
G71 Input in Millimeter
G72 Transformation Off
G73 Transformation Rotation, Scaling
G74 Multi-quadrant Circle Input Off
G75 Multi-quadrant Circle Input On
G77 Zig-Zag Mill Cycle
G78 Pocket Mill Cycle
G79 Bore Mill Cycle
G80 Drill Cycle Off
G81 Z Cycle, Drill (Feed In, Rapid Out)
G82 Z Cycle, Spot Face (Feed In, Rapid Out)
G83 Z Cycle, Deep Hole (Peck, Rapid Out)
G84 Z Cycle, Tap (Feed In, Feed Out)
G85 Z Cycle, Bore (Feed In, Feed Out)

G86 Z Cycle, Bore (Feed In, Stop-Wait, Rapid Out)
G87 Z Cycle, Chip Break (Peck, Rapid Out)
G89 Z Cycle, Bore (Feed In, Dwell, Feed Out)
G90 Absolute Programming
G91 Incremental Programming
G92 Preset Part Programming Zero Point
G94 Feedrate Per Minute Mode
G95 Feed Per Spindle Revolution (pitch) mode.
G96 Restore Base Part Program Coordinate System
G97 Set Work Coordinate System
G99 Deceleration Override
G170 Outside Frame Mill
G171 Inside Frame Mill
G172 Pocket Frame Mill
G173 Outside Face Mill
G174 Inside Face Mill
G175 Outside Circle Mill
G176 Inside Circle Mill
G177 Pocket Circle Mill
G179 Slot Mill
G181-189 Z Cycle (Same as G81-G89) Multi-Hoic
G191-199 Z Cycle (Same as Gg I -Gg9) Frame of floles
M0 PROGRAM STOP (NON MODAL)
M1 OPTIONAL STOP (NON MODAL)
M2 PROGRAM REWIND(NON MODAL)
M3 SPINDLE ON CLOCKWISE **
M4 SPINDLE ON COUNTER-CLOCKWISE **
M5 SPINDLE OFF **
M6 TOOL CHANGE (NON MODAL) **
M7 MIST COOLANT **
M8 FLOOD COOLANT **
M9 COOLANT OFF ** only two of these are used to swap from blue to red plotting.
M19 SPINDLE ORIENT
M20 PROGRAM STOP; GO TO CLEAR POINT (NON MODAL)
M21 OPTIONAL STOP; GO TO CLEAR POINT (NON MODAL)
M22 END OF PROGRAM; GO TO CLEAR POINT (NON MODAL)
M25 QUILL OR HEAD UP (NON MODAL)
M26 TOOL CHANGE; GO TO CLEAR POINT (NON MODAL)
M28 RIGID TAPPING OFF
M29 RIGID TAPPING ON
M30 REWIND TO TOP OF LOCAL PROGRAM (DEFINED BY NUMBER)
M51 AUXILARY FUNCTION TO INDEXER (NON MODAL)
Page 29

CNC WORDS

: DEFINE PROGRAM NUMBER FOR MULTI-PROGRAM STORAGE
/ SLASH - BLOCK DELETE OR VALUE SEPARATION
****N** SEQUENCE NUMBER
****F** FEED RATE
****I,J,K,** ARC CENTER OFFSETS
****X,Y,Z,** AXIS COMMANDS
****S** SPINDLE SPEED
T TOOL SELECT (T1 THRU T99)
****A** ANGLE (IN DECIMAL DEGREES)
****B** INCREMENTAL ANGLE
****E** ANGLE (COLATITUDE); ABSOLUTE AND DECIMAL DEGREES
****R** RADIUS VECTOR DISTANCE *Negative for arcs > 180'*
U,V,W, INCREMENTAL VALUES FOR X,Y,AND Z RESPECTIVELY
****P** PARAMETERS USED IN CANNED CYCLES
****Q** DWELL TIME VALUE; USED WITH G4 AND THE G89 CYCLES
D DIAMETER OF CUTTER

****INPUT VALUES VARY FROM MACHINE TO MACHINE.
CONSULT YOUR MANUAL FOR USAGE**

Samples of the same part using different implementations of Gcode.

The following pages show the same file suitable for a BurneyIII flame cutting machine. The first is in absolute mode G90 The second is Incremental G91. The third absolute with incremental I and J. Notice that the I and J moves are also absolute distances from the program starting position. Burney (Cleveland Machine Controls) is the only machine I know of that uses this method. Most machines in absolute mode still use incremental I and J moves from the last position.

% (works with 2dtorch.ini absolute version)

```
N2 G90
N3 X4.000 Y-6.800
N4 G97
N5 M03
N6 G45 X4.750 Y-6.500
N7 G41
N8 G03 I4.000 J-6.500
N9 X4.000 Y-6.200
N10 M05
N11 X6.000 Y-13.000
N12 M03
N13 G45 X5.500 Y-12.500
N14 X4.000 Y-12.500
N15 G03 X2.000 Y-12.500 I3.000 J-12.500
N16 X0.500 Y-12.500
N17 X0.500 Y-8.000
N18 X2.000 Y-6.500
N19 X0.500 Y-5.000
N20 X0.500 Y-1.500
N21 G02 X5.500 Y-1.500 I3.000 J-4.000
N22 X5.500 Y-13.000
N23 M05
N24 X11.500 Y-6.800
N25 G98
N26 M30
```

%(works with 2dtorch.ini incremental version)

N2 G91

N3 X4.000 Y-6.800

N4 G97

N5 M03

N6 G45 X0.750 Y0.300

N7 G41

N8 G03 I-0.750 J0.000

N9 X-0.750 Y0.300

N10 M05

N11 X2.000 Y-6.800

N12 M03

N13 G45 X-0.500 Y0.500

N14 X-1.500 Y0.000

N15 G03 X-2.000 Y0.000 I-1.000 J0.000

N16 X-1.500 Y0.000

N17 X0.000 Y4.500

N18 X1.500 Y1.500

N19 X-1.500 Y1.500

N20 X0.000 Y3.500

N21 G02 X5.000 Y0.000 I2.500 J-2.500

N22 X0.000 Y-11.500

N23 M05

N24 X6.000 Y6.200

N25 G98

N26 M30

%(works with2d torch.ini Absolute G90 Using incremental IJK checked on setup screen)

N2 G90

N3 X4.000 Y-6.800

N4 G97

N5 M03

N6 G45 X4.750 Y-6.500

N7 G41

N8 G03 I-0.750 J0.000

N9 X4.000 Y-6.200

N10 M05

N11 X6.000 Y-13.000

N12 M03

N13 G45 X5.500 Y-12.500

N14 X4.000 Y-12.500

N15 G03 X2.000 Y-12.500 I-1.000 J0.000

N16 X0.500 Y-12.500

N17 X0.500 Y-8.000

N18 X2.000 Y-6.500

N19 X0.500 Y-5.000

N20 X0.500 Y-1.500

N21 G02 X5.500 Y-1.500 I2.500 J-2.500

N22 X5.500 Y-13.000

N23 M05

N24 X11.500 Y-6.800

N25 G98

N26 M30

%(works with 2dtorch.ini absolute using R instead of I J Only works on newer machines)

N2 G90

N3 X4.000 Y-6.800

N4 G97

N5 M03

N6 G45 X4.750 Y-6.500

N7 G41

N8 G03 I-0.750 J0.000

N9 X4.000 Y-6.200

N10 M05

N11 X6.000 Y-13.000

N12 M03

N13 G45 X5.500 Y-12.500

N14 X4.000 Y-12.500

N15 G03 X2.000 Y-12.500 R1.000

N16 X0.500 Y-12.500

N17 X0.500 Y-8.000

N18 X2.000 Y-6.500

N19 X0.500 Y-5.000

N20 X0.500 Y-1.500

N21 G02 X5.500 Y-1.500 R2.500

N22 X5.500 Y-13.000

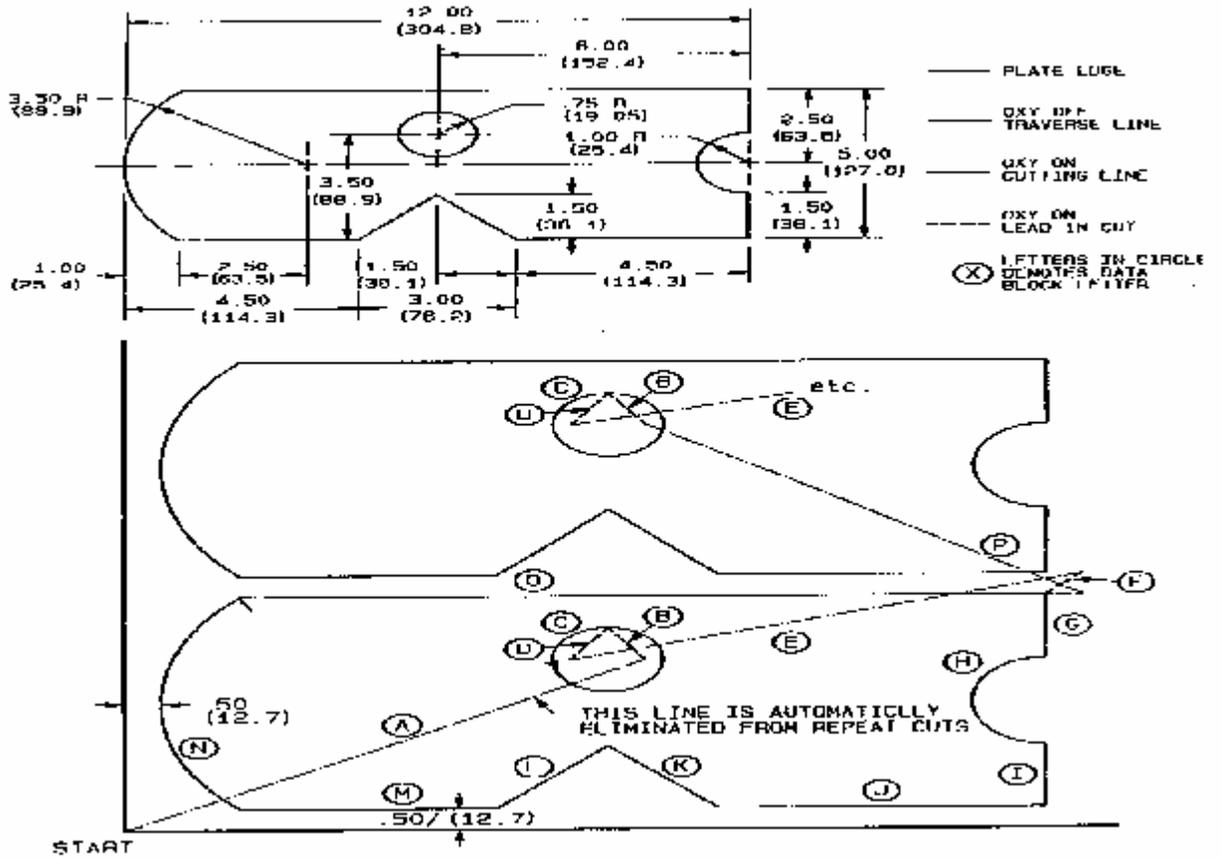
N23 M05

N24 X11.500 Y-6.800

N25 G98

N26 M30

Part Drawings - Sample Program 1
 ENGLISH / METRIC DIMENSIONS



INDEX FOR GCODE

3-D.....	3,11,12,15
7.Data.....	14
8.Data.....	13,14
Addendum.....	24
Angle.Step.....	6,15
ASCII.....	15,16
AutoCAD.....	1-17
Array.....	26
Baud.....	13,16
Blue.lettering.....	6
Blue.line.....	5
Buttons.....	6,7
Char.Delay.Ms.....	14
Circle.Res.....	15
Circles/Angles.....	15
Com.1.2.3.4.....	14
Com.Port.....	14
Comm.....	8,16
Command.Line.Operation..	26
Compress.....	9,14,16
Communications.....	8,16
Connect.Error.....	2,13
CR.LF.on.Transmit.....	14
CR.on.Transmit.....	14
Data.....	8,12,13,14,16
Default.states.....	2
Directory.....	1,3,14,15,17
Drag.and.Drop.....	17
Drawing.(in.Gcode).....	19,20
DXF.file.....	2-17
DXF.Out.....	1-3
Edit.....	1-17
File.....	1-17
Flat.button.....	7
Flip.Flop.....	1,6
Flipping.....	1
Flow.....	13
G91.programming.....	12,15,16
Green.lettering.....	6
Head/Tail.....	15
Help.....	1,9,11,15,16
Highlighted.text.....	23
Incremental.Type.I.J.....	14
Initial.Working.Directory...	15
Labels.....	11,15
Lathe.....	13,15,17
Left.mouse.button.....	5.
Line.Delay.Ms.....	14
Line.Order.....	25
Load.ini.....	12
Offset.....	27

Output.to.tape.Punch.....	8,15
Pan.....	7
Parity.....	13
Plotting.....	5,11,14,23
Plotting.one.line.at.a.time.	5
Punched.tape.....	15
Readable.Header.....	15
Receive.....	8,14,16
Red.lettering.....	6
Red.line.....	5
Right.mouse.button.....	5,14
Rotate.....	6,11,15
Save.as.dxf.....	16,22
Save.ini.....	10,11
Scale.....	6,7,16
Scan.Dir.....	3,4,17
Scan.Time.Ms.....	14,17
Scroll.Bars.....	6
Search.....	16
Send.....	8,14,16
Send.Terminal.....	8
Setup.....	2-16
Show.XYZ.....	15
Single.Step.....	5,14
Snap.....	3
Sort.....	25
Source.file.....	5,6,7
Stop.Bits.....	13
Strip.spaces.Transmit.....	14
Swap.....	1,3,9,12,16
Tip.....	9,10,12,14,15
Track.8.Party.....	15
Translate.....	21
Transmit.....	14,15
USC.....	2
Vector.Reduction.....	13
Want.Cr.on.Receive.....	14
Warning.....	6
Windows.....	1,4
World.0,0,0.....	2

Ver Gcode2000v26

Thank you for using Gcode2000

A final note the **Company** name from the **About** window is now shown at the top of graphic printouts

Web Page:

<http://members.aol.com/andrewc119/index.htm>

Fax 704 630 9544

Email: andrewc119@aol.com