GCODE 95

Requires Windows 95 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 that Gcode is watching.
Gcode will capture 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 can now be read in.

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, 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).

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

**Put any dimensions or text on another level.**
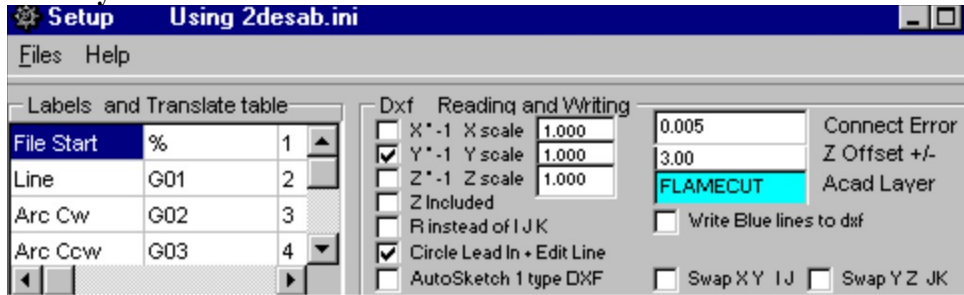
Setting X0 Y0 Z0

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.

*Page 2*_____
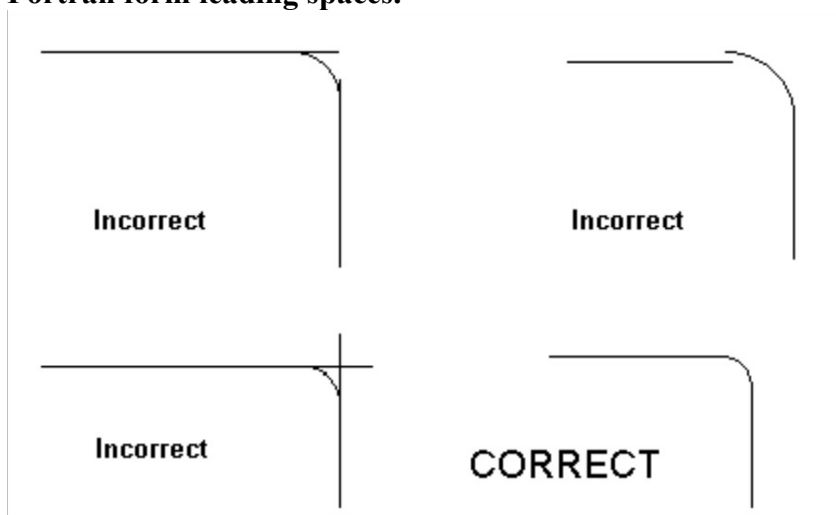
Autocad Continued*:*

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 <u>Use set UCS X Y Z Axis Rotate only in 90 Degree increments</u> 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.
If Gcode had been left running with Scan Dir. checked then the file should already have been loaded and processed.
Provided AutoCAD sent the DXF file to same directory that Gcode was watching.
Otherwise 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.
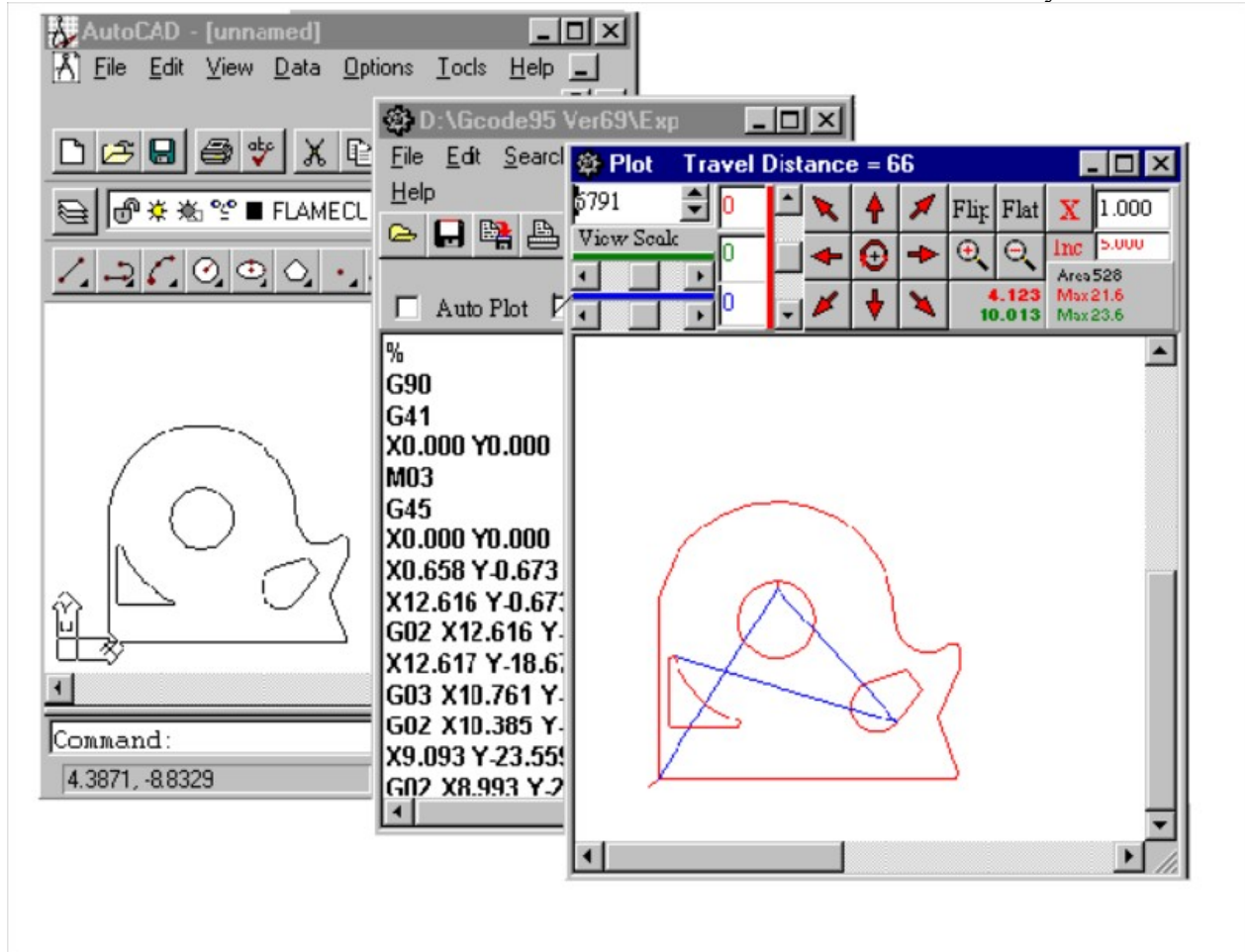To associate a file extension with an application use **My computer,View,Options,File Types**.

Page 3_____

Concurrent operation of AutoCAD and Gcode

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

Notice Scan Dir. is checked in Gcode and that is looking in C:\GCODE95\
If the dxf file from AutoCAD is sent to C:\GCODE95\ then it will be automatically loaded.

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 position. Notice that it is simply a duplicate of the following line
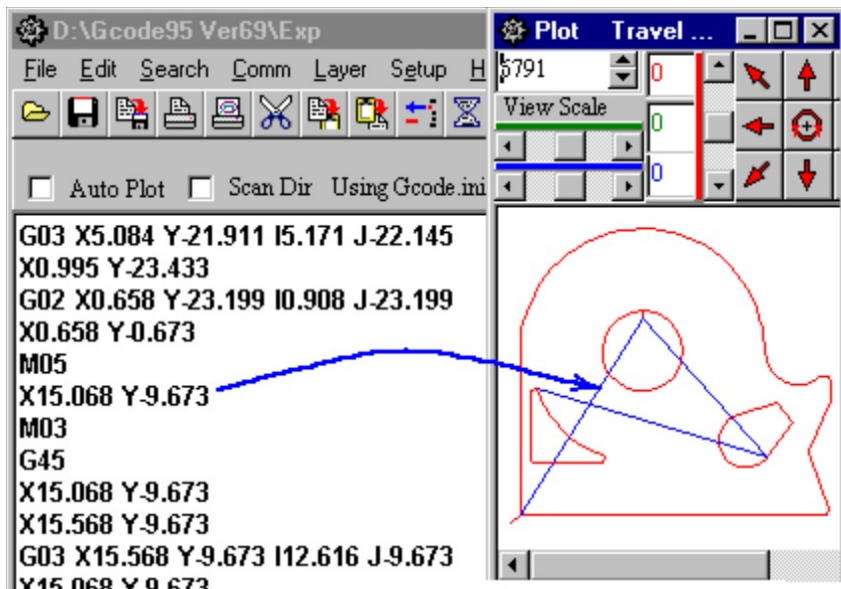

*Page 5_*

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]
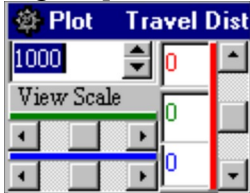
First Box: Red Lettering - Display the vertical scroll bar angular position
[Normally X axis]
Second Box: Green Lettering - Display the upper horizontal scroll bar angular position.
[Normally Y axis]
Third Box: Blue Lettering - Display the lower horizontal scroll bar angular position
[Normally Z axis]

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.
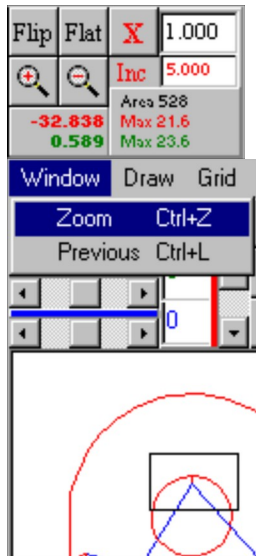Warning all of the above functions change the source file.

*Page 6*_____

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

Position **below Zoom**

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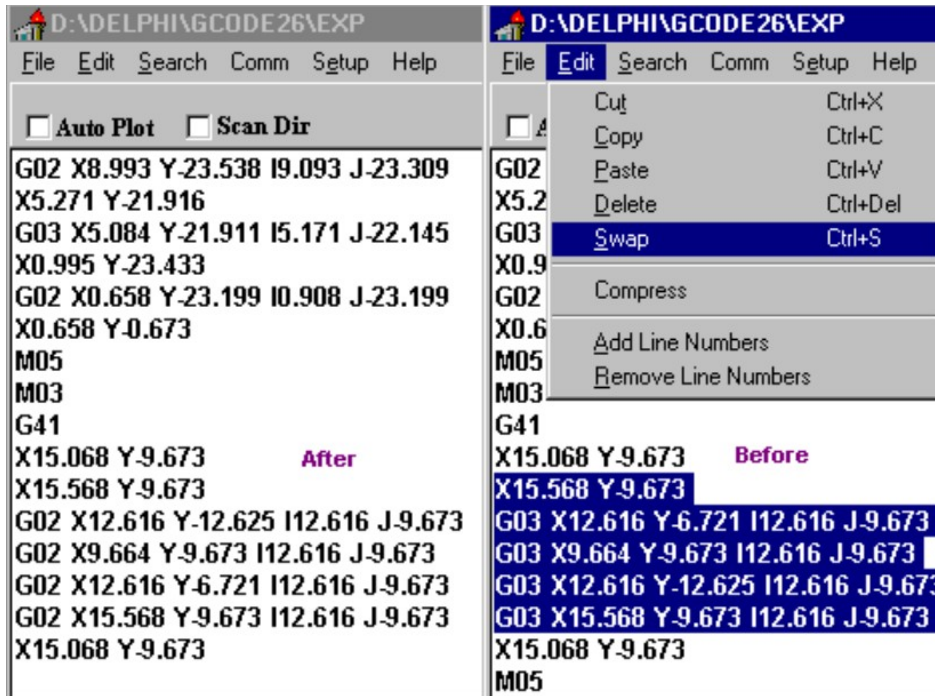Often used as an aid to estimate production time or usage charges.

*Page 7*_____

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.

```
D:\DELPHI\GCODE26\EXP                          D:\DELPHI\GCODE26\EXP
File  Edit  Search  Comm  Setup  Help          File  Edit  Search  Comm  Setup  Help

  Auto Plot    Scan Dir                          A        Cut              Ctrl+X
                                                           Copy             Ctrl+C
G02 X8.993 Y-23.538 I9.093 J-23.309     G02      Paste            Ctrl+V
X5.271 Y-21.916                         X5.2     Delete           Ctrl+Del
G03 X5.084 Y-21.911 I5.171 J-22.145     G03      Swap             Ctrl+S
X0.995 Y-23.433                         X0.9
G02 X0.658 Y-23.199 I0.908 J-23.199     G02      Compress
X0.658 Y-0.673                          X0.6
M05                                     M05      Add Line Numbers
M03                                     M03      Remove Line Numbers
G41                                     G41
X15.068 Y-9.673       After             X15.068 Y-9.673      Before
X15.568 Y-9.673                         X15.568 Y-9.673
G02 X12.616 Y-12.625 I12.616 J-9.673    G03 X12.616 Y-6.721 I12.616 J-9.673
G02 X9.664 Y-9.673 I12.616 J-9.673      G03 X9.664 Y-9.673 I12.616 J-9.673
G02 X12.616 Y-6.721 I12.616 J-9.673     G03 X12.616 Y-12.625 I12.616 J-9.673
G02 X15.568 Y-9.673 I12.616 J-9.673     G03 X15.568 Y-9.673 I12.616 J-9.673
X15.068 Y-9.673                         X15.068 Y-9.673
                                        M05
```

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.

Page 9

Ini Files: [Defines machines relationship to Gcode]
INI FILES: [Defines the machines relationship to Gcode**]**

Setup screen to create an INI file
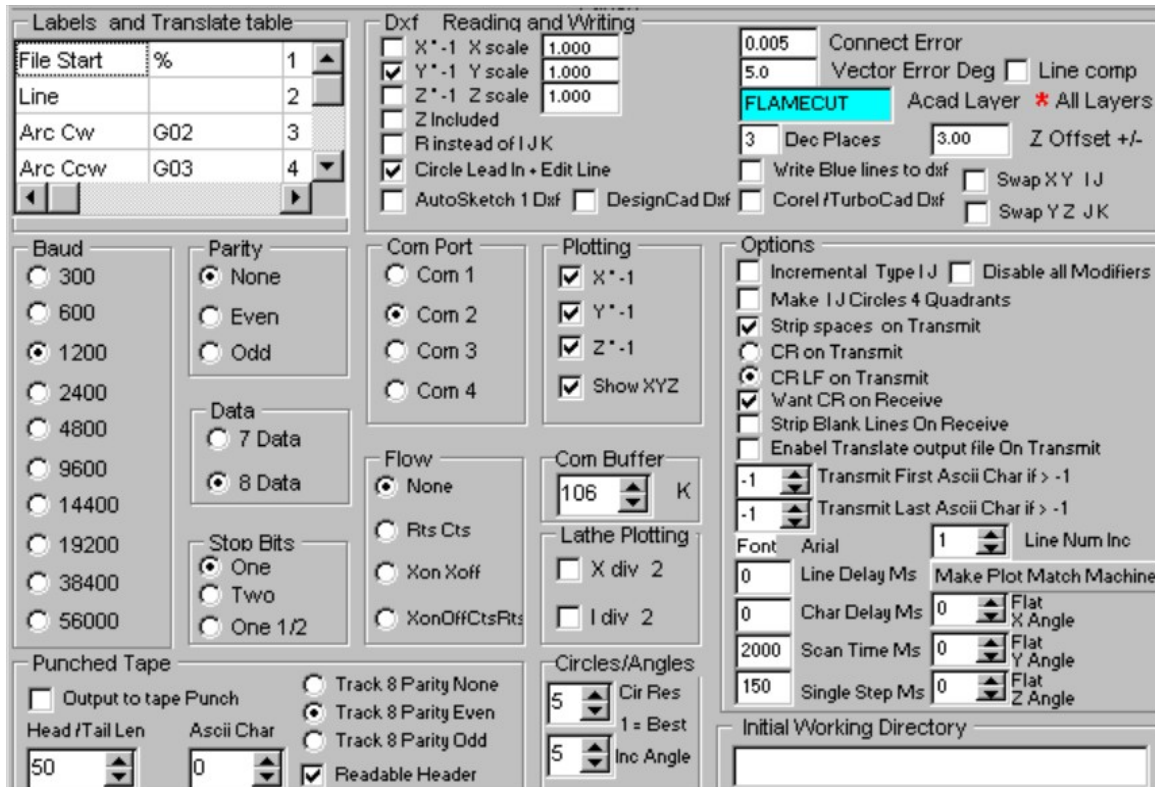
Using the "Setup" window create and save your own INI file:
Selecting: "Setup", "Edit or Create INI "File", "Save ini" Enables the machine set up to be saved as a file name of your choice.
Selecting: "Setup", "Use new Machine Tool INI" and picking an INI file - Will retrieve a previously created machine 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

Page 10_____

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 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.**

 Page 11_____

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 <u>arc if not blank</u>
22.XZ.Plane…...G18...................Inserted in the line before an <u>arc if not blank</u>
23.YZ.Plane…...G19...................Inserted in the line before an <u>arc if not blank</u>
24.Line.2.................................…........Line two Any text can be used or left blank
25.Line.3.............................…….....Ditto
26.Line.4.............................…….....Ditto
27.Line.5.............................………Ditto
28.Rapid...…......G00...................Placed on lines before start cut <u>if not blank</u>
29…………...From.....................To.......Headings for translate table
30...................................Any thing in the from column is translated to
For an exaple 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:
<u>X*-1</u> <u>Y*-1</u> <u>Z*-1</u> - 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.
<u>X Y Z Scale</u> 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.
<u>R instead of IJK</u> Substitutes the letter R in place of IJK to express a tangential radius. Arcs
greater than 180 degrees are expressed as R-xxx.xxx
<u>Z Included Box</u> - a checked box allows the inclusion of the Z axis from Dxf Files.
<u>Circle Lead In + Edit Line</u> - a checked box adds code for a lead in move inside all circles

and a line of code <u>ready to edit</u> for the lead in move between unconnected paths. Notice it is simply a duplicate of the following line.

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

<u>SwapXY IJ</u> 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.

<u>Swap YZ JK</u> See above. Typically swapped for a lathe.

<u>Write Blue lines to dxf</u> 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.

<u>Connect Error</u> - 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.

<u>Z Offset +/-</u> - 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.

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

<u>Decimal Places</u> - 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.

<u>BAUD</u> <u>DATA</u> <u>PARITY</u> <u>STOP BITS</u> <u>FLOW</u> : 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.

<u>LATHE</u> 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 recomended for dxf files containing arcs and circles)

Page13_____

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.

<u>Strip spaces Transmit</u> - 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.

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

<u>CR LF on Transmit</u> - 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.
Scan Time Ms - A delay in 1/1000 seconds between searching the current directory for a new file.
Too small a value can cause a slower computer system to hang if the directory has a lot of files.
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.

Page14_____

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.

Page15_____

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.

Page16 _____

Auto Plot: When checked any edit change is immediately plotted.

Scan Dir: When checked causes the program to constantly look for a new file to arrive in the current directory. When a new file is found, it is loaded, plotted, and flashes the path being scanned for the new file. This is how AutoCAD can output a DXF file and immediately have it processed and displayed. See Scan Time Ms. Checking **Scan Dir** will automate the loading of the latest file to appear in the directory being scanned. This is not a recommended tool on slow machines as it places quite a demand on the system.

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 Gcode95 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.
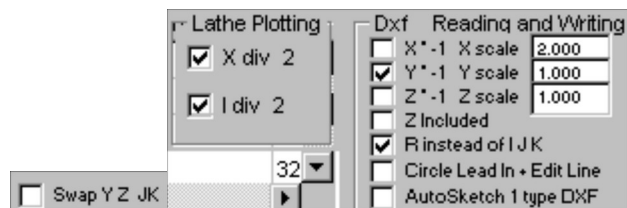
DRAWING

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).
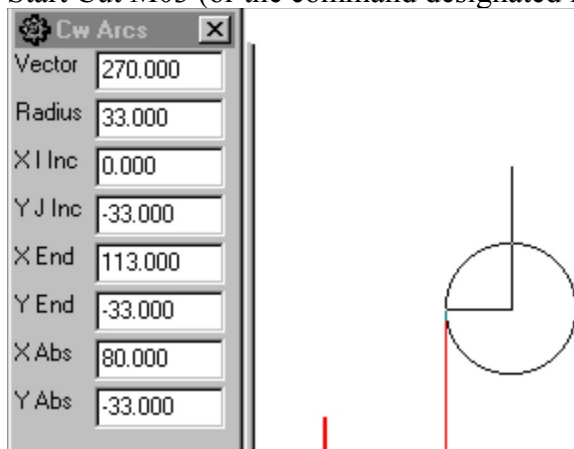
To draw a line.

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 bieng 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.

Labels and Translate table		
Trans From	To	29
I0.000	I.	30
I-0.000	I.	31
J0.000	J.	32

Options
☑ Enabel Translate output

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.

New	Ctrl+N
Open ...	F4
Save	F2
Save As ...	Ctrl+F2
Save As DXF	**Ctrl+F3**
Print Text	Ctrl+P
Print Window	Ctrl+W
Print Plot	Ctrl+L
Print Setup	Ctrl+E
Exit	
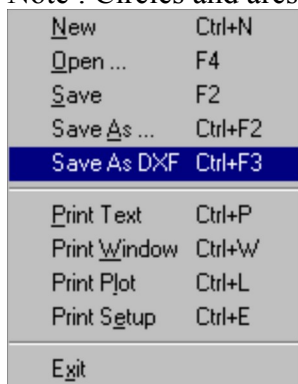
Using Autocad it is a simple matter to convert them back to arcs and circles if required.

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 I J K
- ☑ Circle Lead In + Edit Line
- ☐ AutoSketch 1 type DXF
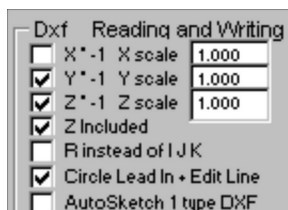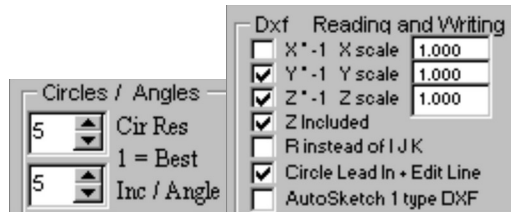
The resolution of the circles and arcs is controlled by the Circles/Angles setting on the
Setup Screen.
Increments as small as 1/1000 can be achieved with a setting of 1.
But remember the file size of the will be considerably larger.
If Z Included is checked then Z axis data will be read in.
Z Axis data 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.

Page 22 _____

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.
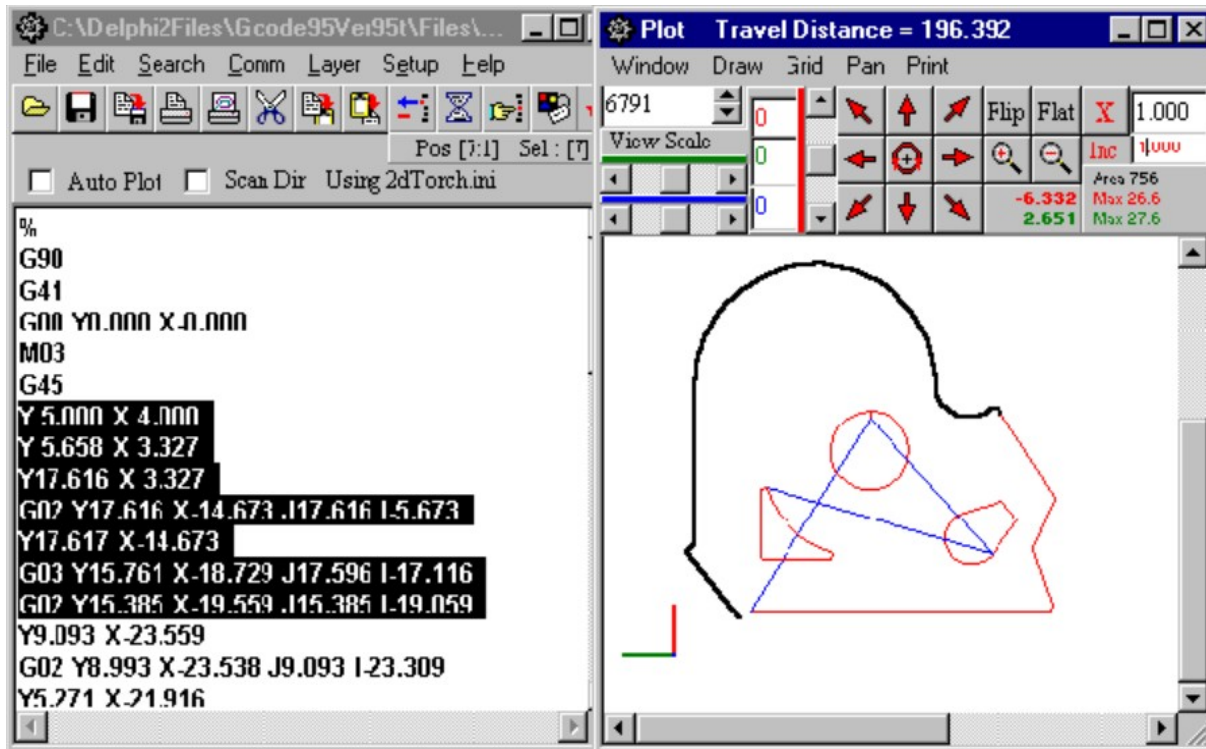
Addendum to options on setup screen

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

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 entitre 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 assending 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.

Page 25_____

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.
Gcode95.exe Filetoload Filetosave inifiletouse.ini
Filetosave and inifiletouse are optional.
If both Filetoload and Filetosave are present then processing will proceed and gcode will close.
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

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 28⎯⎯⎯⎯⎯⎯⎯⎯⎯⎯⎯⎯⎯⎯⎯⎯⎯⎯⎯⎯

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 (Tl 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 adsolute 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
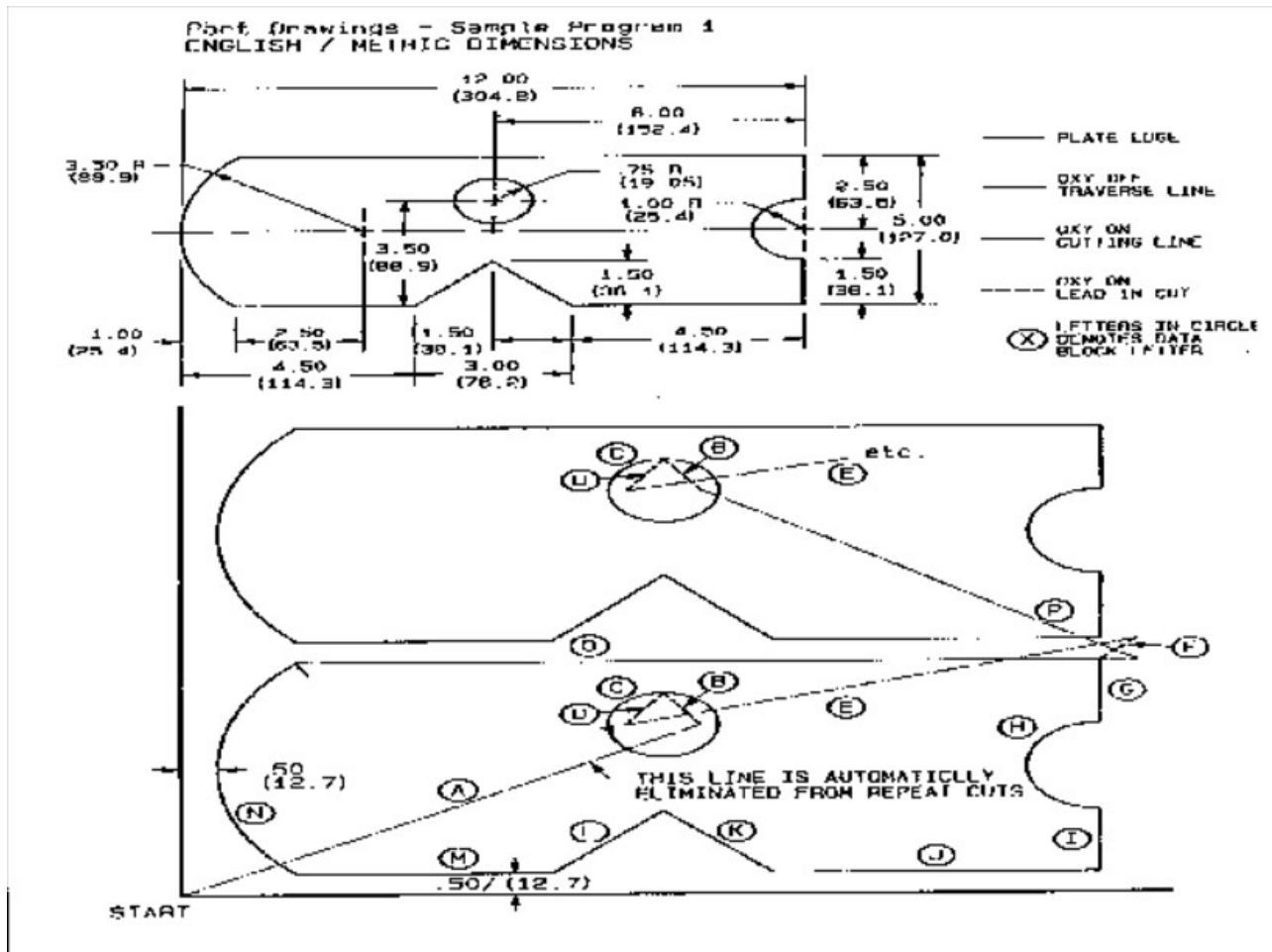N19 X0.500 Y-5.000
N20 X0.500 Y-1.500
N21 G02 X5.500 Y-1.500 I3.000 J-4.000
N22 X5.500 Y-13.000
N23 M05
N24 X11.500 Y-6.800
N25 G98
N26 M30

%(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

VerGcode95v16