DeskCNC

Setup & Operating Manual

for 3 axis mill

Draft version - March 2004

DeskCNC Manual

Contents   Section abbreviation

DeskCNC Manual

Introduction

This manual is an attempt to give enough details to allow the setting up of a 3 axis milling machine driven by stepper motors, using deskcnc, and its controller board. It does not specifically refer to any specific milling machines or drive systems.

This manual is based on the original produced by Peter Dunster. It is available in PDF format, and it is hoped that it will be kept up to date with recent DeskCNC software releases. It may be freely copied and distributed in its complete form for non-commercial purposes, i.e. You should not have to pay for it. Please refer any suggestions/corrections to cnc@yertiz.com.

There are many stepper motor driver boards/components available from many suppliers. This manual refers to the Carken 4axis board, but virtually any driver that uses step and direction signals will do. It is essential, however, that you use the DeskCNC controller board, since it converts the serial output from the PC into the requisite step/direction data streams.

Page numbering System, etc..

DeskCNC is software under development. This means that functions will be added and other improvements will be made as time goes on. When software changes occur, then it is hoped to be able to update this manual at the same time, so a conventional sequential page numbering system would rapidly become out of step as pages were inserted or removed.

At the bottom of each page, there is therefore a section abbreviation, a date, and a section page number. The contents page refers to each section giving its section abbreviation. A change to an existing section or the insertion of a new section will mean that only a few pages need be printed off, not the whole manual.

One of the appendices has a summary of the software changes. If you refer to this you may be able to determine if you need to update your copy of this manual.

Information has been obtained by Ray West from sources believed to be reliable. However, because of the possibility of human or mechanical errors by our sources or ourselves or others, Ray West does not guarantee the accuracy or completeness of any information and is not responsible for any errors or omissions or the results obtained from the use of such information.

DeskCNC Software

DeskCNC is a Windows based CAM/CNC solution for creating toolpaths, copying existing models, and running CNC machines. DeskCNC can create toolpaths from DXF, STL, Gerber, Excellon, and Image file formats as well as Digitize existing part geometry (requires optional Probe or Laser Scanner). DeskCNC can smoothly run any CNC machine from within Windows. It includes a full G-code interpreter/editor and runs in any version of Windows 95/98/Me/Nt/2000/XP.

The CNC Teknix DeskCNC Controller handles all linear and circular interpolation and times the output pulses with a resolution of 800 nanoseconds directly from Windows. It accepts a 24 bit (+- 8 million steps) absolute position and determines the ramp up, slew, and ramp down velocities based on a linear acceleration ramp profile. Continuous Motion Contouring is available for smooth cutting. It outputs standard +5v logic high and 0v logic low step/direction/output signals. The controller can be commanded to turn on four auxiliary relays (not included) if desired. It accepts Home switches, a limit switch, a Digitizing Probe, and an Emergency Stop switch. It is capable of providing a step pulse rate between 40 - 40,000 Steps per Second.

DeskCNC Requirements

• Windows 95/98/Me/Nt/2000/XP operating system.
• Recommended: 32 MB and OpenGL compatible Windows Pentium computer.

The Three Modes of Operation

DeskCNC integrates CAM (the creation of toolpaths), Verification (the displaying and editing of toolpaths), and CNC Machine Control (the actual running of the CNC machine) into one software package.

Drawings or artwork in many different formats can be imported into DeskCNC through the Files Menu. Once imported and displayed on screen they can be used as the basis for various toolpathing functions to create the desired model, which can then be saved as a G code program for use on your CNC machine. DeskCNC uses a variety of Postprocessors to create these files, which are customizable to suit the end users requirements. The file can then be used in the Control Mode to run the machine to create the model. Before this mode can be used however the user will need to set the machine parameters into DeskCNC.

Setting up DeskCNC

This section will now take you through the steps necessary to set up your DeskCNC software so you can start using your machine.

Assuming you now have your controller connected to your computer (see appendix 3) you have to configure your software to control your machine. The settings will change from machine to machine but the procedure is the same in all cases.

To start the software configuration you will need to go to the Menu bar at the top of the screen and click on Setup/Machine Setup. The Machine Configuration window will open, being similar to the window shown below. The values may well be different for you in this and any other windows shown in this documentation.

Select the Machine's Name to edit an existing Machine or enter a new unique name to create a new Machine Setup. If the Machine Setup was previously created, the Machine's parameters will be filled in. Select an existing Machine before making any changes to the Machine's parameters. These setup options are used for the DeskCNC Controller and are not required when no Controller is connected.

In this window there are 5 Tabs, these are:

• *Machine Velocities Tab*

• *Axes Setup Tab*

• *DeskCNC Setup Tab*

• *Tool Sensor/Probe*

• *Machine*
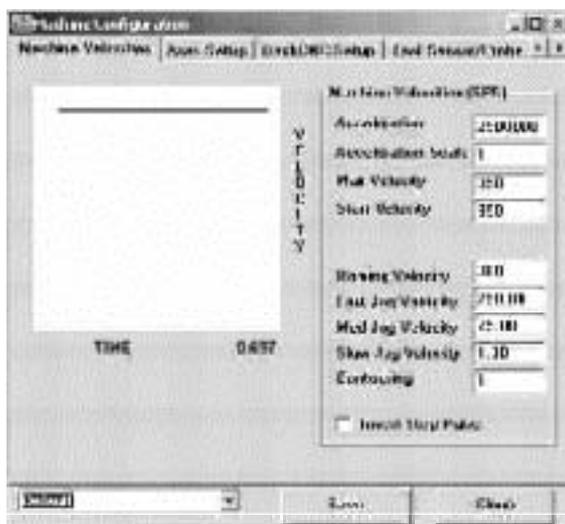
Tool Sensor/Probe

This is discussed in Appendix 1, in the mean time set to disable.

Machine

Select the right-hand tab 'Machine' (This will be visible when you click on the right-hand arrow at the top of the window), and the following window will appear
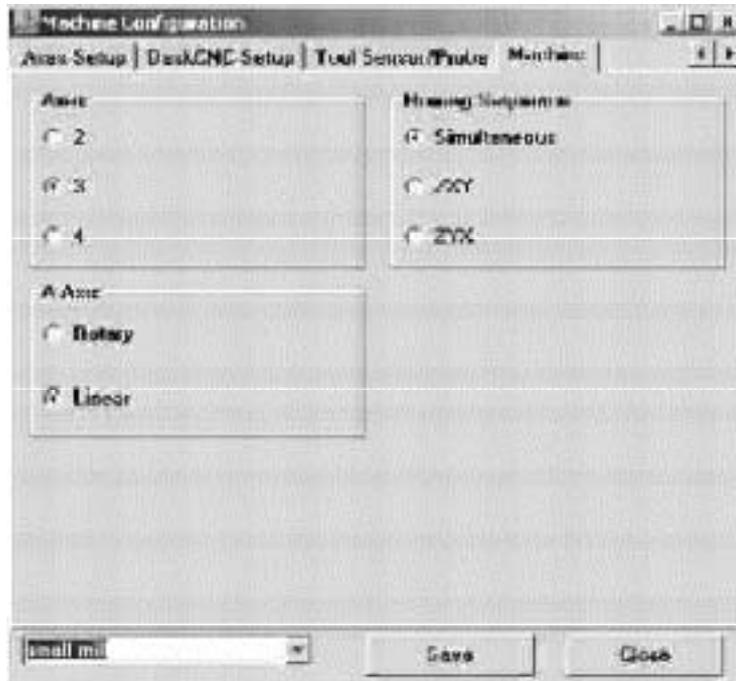
Enter the number of axes for your machine, the machine name (if required) and the preferred order for the homing sequence. At the moment, the rotary 4th (A) axis is not available.

Machine Configuration

This page is used to define your machine workspace, the screen display, the connection to the controller and your base unit, metric or imperial.

Machine Size Set the Machines useable table size. The displayed Grid will reflect this size. It is used for display purposes only and does not represent a soft stop.

Grid Size Set the size of the Display Grid.

Resolution Set the Resolution of the OpenGL rendering. This value is in Pixels and determines the graininess of the rendering. Smaller values have a better display at the expense of drawing speed. Adjust for a comfortable rendering display for your computers capabilities.

Zero Plane Set the Z Height of the OpenGL rendered images. This value denotes where 'cutting' begins. If you are using different CAM software and creating 'positive' toolpaths (A 'cut' above Z = 0), then set this value to the Z value where cutting begins.

Jog Distances Set the values that will be available in the Jog Box when jogging the Machine. To add a distance 1.25 for example, enter 1.25 and hit the Add button. To Delete a value, highlight it and click the Delete button.

Speed Key Macro's Set the values that will be available in the MDI Box when pressing the Function Keys. To add the G-Code G01 X1.2 Y3.4 for example, enter 'G01 X1.2 Y3.4' and hit the Add button. To Delete a value, highlight it and click the Delete button. These G-Code commands are then entered automatically (in the MDI Box) when a function key (F1 etc.) is pressed.

Com Port Enter the Port that is connected to the DeskCNC Controller.

Decimal Separator Enter the separator used in your Country to separate numbers from fractions. In the US use a decimal point (1234.567). In some European Countries, use the comma (1234,567).

Highlight G-Code Highlights each line of G-Code as it is being interpreted. Due to the look ahead features of DeskCNC, this G-Code line will almost always be well ahead of the actual line of G-code being processed by the controller. Highlighting G-Code requires a bit of computing power and is recommended only for strong computer systems.

Units Select the default start up units for your Machine. This can be overridden by using the G20, G21 functions.

Next you need to define the machine, its axes setup and resolution, this can be done under the **Axes Setup** tab.

Axes Setup
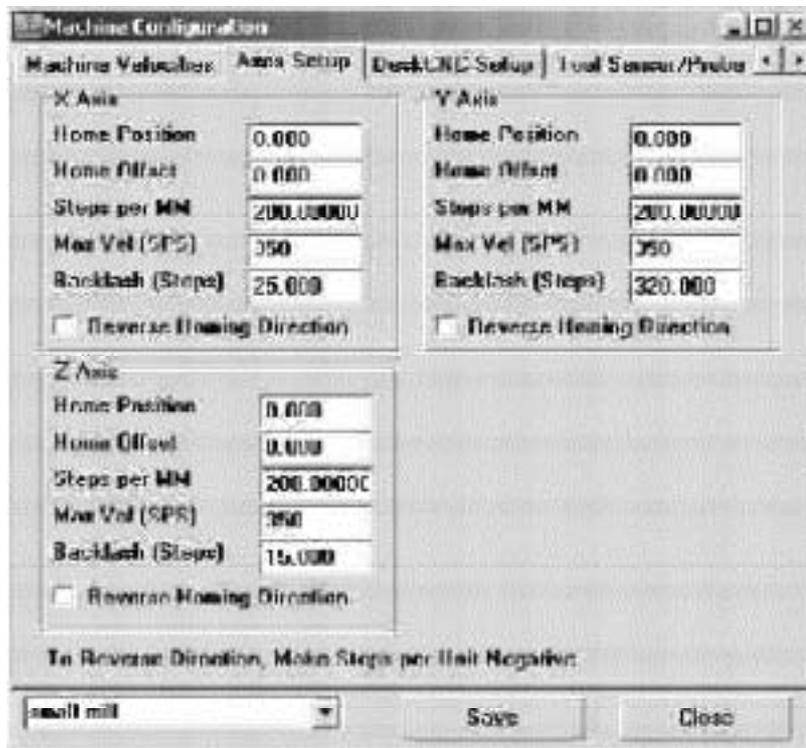
Home Position If your machine is fitted with home switches you can set their Position (in Absolute user units). After a Homing operation, the coordinates of the Axes are set to this position.

Home Offset Set the Home Offset Position (in Absolute user units) while performing a Homing Operation. After a Homing operation, the Machine will move to this position. By setting the Home Position to a negative value, the origin of the Machine can be set a short distance away from the switch. For example: An X Home Position of -0.25" and a Home Offset of 0.0 will set the Machines X Axis Zero Position 0.25" away from the X Axis Home Switch.

Steps per MM Set the number of steps (or encoder pulses if servomotors) to move 1 mm for each axis. For Example: 8000 Steps per inch of movement = 8000 / 25.4 = 314.960662992 Steps per MM. If you are not sure of this resolution try setting it to an approximate value and move the axis a fixed distance according to the display. Measure the actual distance moved and calculate the resolution by [approx. Resolution * (distance moved/measured distance)] . This value will always be required to be entered in millimetres (metric), even if the rest of the machine settings are set to inches (imperial).

 1 inch is exactly 25.4 mm.

Max Velocity (SPS) Set the Maximum Velocity of each axis in Steps per Second, this value is used for moves with the G0 command. If you are unsure of this value try setting it to an arbitrary value and execute a G0 move, if it stalls then the value is too high, reduce it and try again. Be sure to set the acceleration first.

Backlash Enter the amount of Backlash for each axis. Backlash Compensation in software can never approach the quality of a tight machine. It is recommended to install anti backlash screws in the Machine rather than rely on Backlash Compensation in Software. See appendix 5 for more details on this setting

Reverse Homing Direction Check this box if the Machine moves in the wrong direction to seek an axis' Home Switch.

Machine Velocities
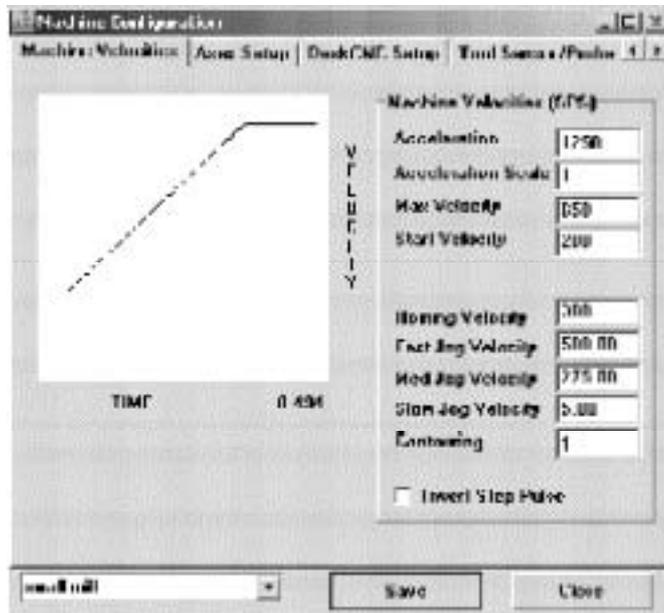
Setting up your Machine Velocities DeskCNC utilizes a linear acceleration profile throughout its 40 - 40,000 SPS range. To help set the velocity and acceleration parameters, a graph of the acceleration profile is made available.

Parameters for viewing the acceleration profile

All velocity values for this window are in units of steps per second.

• **Start Velocity** - Enter the instantaneous start velocity. This is the velocity the motors can instantly start at without the need for acceleration.

• **Max SPS** - Enter the Max Steps per Second of the Machines Motors.

• **Acceleration** - Keep increasing the Acceleration value until the acceleration graph goes from the bottom left to the top right corner of the graph. The profile should end with a plateau (a small section that is flat). It is OK to have high acceleration values that increase the plateau's length but you should never use a low acceleration value that produces a spike at the end of the graph. At the bottom right of the graph will be the time in seconds to accelerate to full velocity as specified by the Max SPS. Note that at high Max SPS values, the acceleration value will be very large. This is normal. The acceleration will later be scaled to slow it down if necessary. See Acceleration Scale below.

• **Acceleration Scale** - This is a scale factor that will reduce the acceleration most notably in high SPS ranges. Entering an Acceleration Scale greater than 1 will decrease the acceleration. The Time value in the bottom right corner of the graph will increase as the Accel Scale is increased. This is the time the controller will spend ramping up or down from the Instant Start Velocity to the Max SPS.

Homing Velocity - set value in steps per second

Fast, Medium, and Slow Jog Velocities - Set the velocities for Continuous Jogging.

Contouring - Turns Continuous Motion Contouring (CMC) On/Off and sets the threshold. When Contouring = 0, CMC is off and all moves will decelerate to the Machine's Start Velocity. When Contouring = 1, CMC is set to Automatic. In Automatic CMC, each move will decelerate only the amount necessary to make a smooth transition into the next move. If Contouring is set to a value greater than 1, CMC is turned on with a threshold equal to this value. In this mode, if any axis has a change in Steps per Second that is greater than this value it will decelerate to the Start Velocity. If all axis' have a change in velocity that is less than this value, then no deceleration takes place.

Invert Step Pulse - If you are running Firmware Version 1.34 or later, the step pulse can be changed from a High to Low pulse to a Low to High pulse.

Tool Library Setup

The tool library holds the definitions of the tools and is used to quickly enter tooling information when creating toolpaths. It is also used (required) by the rendering engine to display the finished part as a 3D solid using OpenGL technology .

Number The Number entry is the tool number called by the Txx code.

Length The Length parameter is the Tool Length Offset.

Diameter The Diameter is the Tool Diameter.

Type The Tool Style (End, Ball, or Vee) is used to properly render a 3D Solid of the finished part.

Ang/Rad For Vee cutters, enter the included Vee angle as the 'Ang/Rad' parameter. All other cutters do not require an 'Ang/Rad' parameter.

Comment Enter the name of the tool along with any additional info to help you choose the proper cutter.

Add Tool Click this button and fill in the tools parameters. For tool Type, use End, Ball, or Vee only. These must be spelled exactly.

Delete Tool Highlight a tool and click the Delete button to remove a tool from the Tool Library.

Post Processor Setup

DeskCNC utilizes a programmable Post Processor that can output many machine languages. Many of these can be read back in by the editor for viewing of the toolpaths. The post 'DeskCNC' is a standard G-Code Post Processor that the program DeskCNC can read and interpret.

Name - The name for this Post Processor. It should be a unique easily identifiable name.

File Extension - The file extension for the saved files.

Comment Characters - DeskCNC inserts comments at the beginning of the file to denote time of creation, tool diameter and number, and the name of the Post Processor. Enter the character that begins a comment for this machine.

Tokens - Commands are broken down into 'Tokens'. A Token consists of an Address Label and a Value. The Token 'X2.5' begins with the Address of 'X' (in this case representing the X axis) and its value of 2.5. Tokens can begin with a space to make the output file more readable. All Tokens must be created before they can be used. To create a Token, select the Token you wish to create from the Tokens group and a variable. Available Tokens are: Xt for the X Position, Yt for the Y Position, Zt for the Z Position, It for the Arc Center X Position, Jt for the Arc Center Y Position, Rt for the Radius, Nt for the Line Number, Ft for the Feed rate, St for the Spindle Speed, Tt for Tool Change, Ct for tool clearance, Misct for any miscellaneous commands.

Available variables are: None, Sx for the Start Point X of a tool path, Sy for the Start Point Y of a tool path, Sz for the Start Point Z of a tool path, Ex for the End Point X of a tool path, Ey for the End Point Y of a tool path, Ez for the End Point Z of a tool path, Sa for the Start Angle of an Arc, Ea for the End Angle of an Arc, Rad for the Radius of an Arc, Cx for the Center X of an Arc, Cy for the Center Y of an Arc, Ix for the Incremental Center X of an Arc, Jx for the Incremental Center Y of an Arc, Cv for the Clearance Value, Nv for the Line Number Value, St for the Spindle Speed, Tn for the Tool Number. Enter the number of decimal places for this Token and whether it needs a '+' to denote positive numbers and press the Create button. An example of the newly created Token will be displayed at the bottom.

Movement Script This is the script that defines how a command line will be formatted in the output file. It consists of adding a Command along with the previously defined Tokens. The typical G-Code line N100 G01 X2.5 Y3.6 Z 4.7 consists of a 'Line Number Token', Command 'G01', 'X Position Token', 'Y Position Token', and 'Z Position Token'. Movement Scripts can be either a single command line or a block of continuous commands as set by the 'Single Move' or 'Continuous' selection. A Modal Token will only be written to the output file if it is different from its previous value. Movement Script Example: Select 'Linear'. Select 'Single Move'. Enter 'G01' in the Command box. Select the 'Nt' Token and press the Add button. Press the Add Command button Select the 'Xt' Token and press the Add button. Select the 'Yt' Token and press the Add button. Select the 'Zt' Token and press the Add button. Select 'Ft' and press the Add Modal button. As each button is pressed, an example of the

formatted line is displayed at the bottom. The feed rate Token (Ft) is displayed in brackets to show that it will only be written if it differs from a previous value.

Header / Footer Here you can add any miscellaneous start up or end commands that will be written to the output file. you can also add your own comments.

Save Saves the Post Processor definition to a file so it can be used when saving toolpaths.

Delete Deletes the Post Processor selected under 'Name'. You will be prompted for confirm.

Edit Loads a previously created Post Processor for editing.

DeskCNC Options

Here you can change the Options and Preferences in DeskCNC.

Tolerances Tab

Min Arc Length Enter the minimum length of an Arc segment. Arcs less than this length will be converted to line segments. The length of an Arc should be less than the length of a few 'steps' as defined by your CNC machine.

Min Arc Radius Enter the minimum radius of an Arc segment. Arcs less with a radius less than this value will be converted to line segments.

Tangential Deviation Enter the minimum deviance that adjoining linear segments can have. Linear segments that are less than this value are joined into one linear segment. This value can reduce the size of the Machine Code file.

Editor Tab

Color Enter the color of the Command, Address, or Comment that is displayed in the Edit Box when Show Syntax is enabled. Show Syntax allows easier reading of the Machine Code file by 'Highlighting' key words based on the currently loaded Post Processor.

Style Enter the style of the Command, Address, or Comment that is displayed in the Edit Box when Show Syntax is enabled. Show Syntax allows easier reading of the Machine Code file by 'Highlighting' key words based on the currently loaded Post Processor.

Misc Tab

Default Direction of Closed Regions The orientation (Clockwise or counterclockwise) that all closed regions will be set to when a DXF File is loaded. Tool paths will be calculated from this direction. If you change this setting, you should reload your DXF file for the changes to take place.

Mark Direction of Regions Set to Mark First Entity Only if you want just the first entity in a region to have an arrow displayed at its midpoint. Set to Mark All Entities if you want all the entities to have an arrow displayed at its midpoint. Less screen clutter is present under 'First Entity Only'. Only available if Show Directions is checked in the View menu.

When Machining Tops of Islands at Multiple Depths Check this box if you want each island to have its contour traced when machining its top. By default, the tops of islands are machined with the current tools offset equal to half its diameter. By selecting this option, the entire top of each island will be machined.

Tip If your islands are always separated by any other region by at least 1/2 the current tools diameter, then you can safely trace the contour of each island. If an island is

nearer to a boundary than 1/2 its diameter, then tracing the islands contour will gouge the boundary where it is less than 1/2 the diameter apart.

Tip With Mark First Entity Only, you can easily determine the first segment of a region or tool path.

Show Boundaries when Redrawing Tool paths If View Tool paths is selected in the View menu, you have the choice of viewing just the tool path segments or the tool path segments and the selected regions (If the selection hasn't changed then the selected regions represent the original boundaries and islands and contours).

Pocket Milling Tool Overlap When in the Pocket Window, the X-Y Step Size will automatically be calculated as this percent of the tool diameter. It can always be overridden.

Post Processor Select the default Post Processor that is loaded when DeskCNC is started. Machine code files will then be save according the definitions associated with this Post Processor.

Text Tab

Font Select a default True Type Font. All text references in the DXF file will be converted to vectors while loading.

Precision Each curve in a True Type Font glyph will be converted to a series of line segments. The number of line segments per curve is set in the 'Precision' parameter.

STL Tab

STL Depth The depth at which to set the top of the STL geometry on load. All geometry will be translated in the Z direction so that the Maximum Z value of the geometry is located at this depth. The Blank's Z2 value will be set to this depth.

X and Y Step as percentage of cutter diameter The X and Y step size will be automatically set to a percentage of the cutter diameter when entering the cutter diameter in the machining parameters.

Digitize Tab

Tip Diameter
If you are using a digitizing probe (5 way switch) you can enter the diameter of the tip of the probe. DeskCNC will use this as an offset to calculate the actual surface of the digitized object.

CAM MODE

Once the setup has been completed and saved you can now start importing your files and creating toolpaths. To import a CAD file click on FILE menu tab and select the format to suit the filetype you are importing.

Working with DXF files

Once imported, your DXF file should be seen in the centre of the edit screen. Unlike STL file formats, DeskCNC cannot automatically generate toolpaths around the object. Before any toolpaths can be created selected regions of the drawing and the machining operation need to be selected. The selection menu can be accessed by pressing the right mouse button whilst the mouse is in the edit window. The selections are:

Select All This command will automatically select all islands within all boundaries, all boundaries within all islands (Nested boundaries and nested islands), and all open regions. Use this command when you want to generate toolpaths for the entire DXF File or the majority of it. **To Cancel select operation** Press Esc.

Deselect All This command will Deselect everything. All boundaries, islands, and selected contours will revert back to unselected regions. **Tip** Use the Information command to see which islands belong to which boundaries. It is easier to select more regions than desired and then Deselect what you don't want than to select what you want one at a time.

Select All Open This command allows you to select only open regions. Open regions can be processed by the Contour function only.

Deselect All Open This command will deselect all selected contours.

Select All Closed This command will select only those regions that are closed. It will automatically determine islands within boundaries and boundaries within islands (Nested boundaries and islands). It is most useful when utilizing the Pocket command.

To Cancel select operation Press Esc.

Deselect All Closed This command will deselect all selected closed boundaries and islands.

Select Window This command will select all boundaries, islands, contours, nested boundaries, and nested islands within the selection box. To use this command, hold the left mouse button down and drag a box around the area you are interested in.

Deselect Window This command will deselect everything within the deselection box. To use this command, hold the left mouse button down and drag a box around the area you want to deselect.

Select Region Use this command by clicking the left mouse button on the outermost closed region that you are interested in. It becomes a boundary. All islands to this boundary, contours, nested boundaries, and nested islands within this boundary will automatically be detected and selected.

Deselect Region Use this command by clicking the left mouse button on the outermost selected closed region that you are interested in. All islands to this boundary, contours, nested boundaries, and nested islands will be deselected. **Tip** If your geometry has only one boundary (NOT counting nested boundaries) then this command will be a little faster than the Select All command with the same results.

Select Single This command will select a single region. Click the left mouse button on the region you want to select. DeskCNC will determine whether this region is a contour, boundary, or an island to an existing boundary.

Deselect Single This command will deselect a single boundary, island, or contour. Click the left mouse button on the region you want to deselect. **Tip** Select an outer boundary first, then select its islands when using this function.

Select Empty Circles This command will select all circles that have no other regions inside of them. It is used mainly for the Drill function to select the drill points.

Deselect Empty Circles This command will Deselect all circles that have no other regions inside of them.

Declare Boundary This command will select a closed region and force it to become a boundary. Click the left mouse button on the region you want to select as a boundary. You may never need this command. Use this command if one of DeskCNC's automatic selection tools fails to select a region as a boundary and you know it should be!

Declare Island Use this command to force a region to become an island. To use this command you must first declare the boundary that this region will be an island to. Left click on the boundary (It does not have to actually be selected). Now left click on the region you want to force to become an island to this boundary. You may never use this command. Use it if one of DeskCNC's automatic selection tools fail. Tip Use the Information command afterwards to verify that the island is actually an island to the boundary you declared.
For more information on working with DXF files see the tutorial in the appendix at the back of this manual.

Editing DXF Files

Delete Selected Regions This command is used to clean up a DXF File that contains a large number of Regions that you are not interested in. You may never need to use this command. It is available because it may make the geometry more readable and slightly speed up certain Pocket or Contour tool path calculations. This command permanently alters DeskCNC geometry. It can not be undone. The only way to undo this command is to reload the DXF File. **Example** If you want to Pocket a small portion of a file that contains numerous open Regions. It may speed up the tool path calculations slightly and make the View easier to see if you Delete the Open Regions.

Reverse Direction of Selected Regions When a DXF File is loaded, open and closed Regions are automatically created and their orientation set to either clockwise or counter clockwise (Depending on what option is set under Options). If needed, use this command to reverse a regions direction. All closed regions should be orientated in the same direction in order for the Tool path commands to work properly. This can be verified by the Show Directions command.

Change First Entity This command changes the first entity in closed Region. It can be useful to set the starting point of a tool path loop. Use this command by Selecting Change First Entity from the Edit Menu. Then use the left mouse button to click on the entity you wish to be the starting segment of a closed Region. It will be marked with an arrow at its midpoint. To Cancel operation Press Esc.

Set Machining Order With this function the order the Regions are machined can be controlled. DeskCNC automatically determines an efficient order of machining when the DXF file is loaded. The machining order can be determined from the use of the Show Information function. When Set Machining Order is invoked, a Machining Order window pops up. Enter the first region that you want to change and press OK. Simply click on each Region in the order you want them machined. Example - If the first four Regions are in the correct machining order then enter 5 and press OK. You then proceed to select the next (in this case fifth) Region to be machined. The machining order of Islands and nested Boundaries are automatically calculated and can not be changed.

Explode selected This function will reduce a region to its individual entities. It will create a separate region for each entity which can then be joined together to form a new region. It is useful when entities intersect or coincide. To use this function, first select the regions you want to explode, and then press Explode Selected.

Join selected This function will join together open regions and form new regions. To use this function, select all entities you want joined into new regions. They do not have to be selected in any order, nor do they all have to be connected to each other. DeskCNC will join the ones that are connected and form a new region. It will not force an entity to be joined with another if they are not connected, therefore multiple regions may be created. If a new region is closed, it is automatically identified as closed. **Tip** If

your DXF geometry has intersections, you may need the explode/join functions to create a region as it was intended to be.

Selecting Multiple Depths Any Region can be set at any depth before executing any of the Tool path functions. It is recommended that Show Information is active to display the current depth of each region. By selecting 'Change Depths' from the Edit menu, a small window is displayed. Islands are automatically updated to reflect the new depths. If an Island's depth becomes deeper that its parent Boundary then it becomes a nested Boundary and changes color to reflect this. therefore, it is recommended that the depths of Boundaries are changed before the depths of Islands. All depths need not be set under this command. Selected Boundaries that have a depth of Zero will take the depth set in one of the tool path functions. To set a new depth, enter a new depth and select which Regions to apply it to. The new depth can be applied to all of the Selected Boundaries, all of the Selected Islands, or to an individual Region. Pressing the Apply button sets the depth(s) and updates the display to reflect the changes.

Action of the Multiple Depth function during pocketing While machining islands at different depths, DeskCNC will first remove material around an island until it reaches a depth greater than the depth of the island. DeskCNC will then remove material from the top of the island according to that islands depth.

Selecting Wall Profiles When this command is invoked, a DXF file named Profiles.dxf will be loaded. Depths should be set prior to adding wall profiles. It is recommended that Show Info is active when using this function. By default, all regions have no (0) profile which is the same as a straight vertical wall. To select a new profile, enter the region number in the region entry box. Select a new profile in the profile entry box and enter its Delta value. The profile will be displayed in the profile viewing box. Press Apply to attach this profile and Delta value to this Region.

Delta This is the offset of the profile at the bottom of the Region. By changing the offset at the bottom of the profile, a single profile can represent an infinite number of profiles. Therefore, a single straight line can represent a tapered wall at any angle and a quarter circle can represent any quarter elliptical profile. By default, the delta value for each region is equal to its height. At any time, entering a - (negative value) will cause the default delta value to be set. Entering a value greater than the default will increase the bottom offset of the profile.

Creating New Profiles In your CAD program, load the file profiles.dxf. Find an empty area and draw a new profile. Keep the aspect ratio consistent with what you intend to use this profile for. In other words, draw the profile exactly as you would want to use it. The size of the profile is unimportant. The profile will be scaled to the depth of the region it is attached to. Save the new file as Profiles.dxf.

Common Tooling Parameters

These definitions and guidelines are common to all tool path functions. For help on an individual tool path function see the help for Pocket, Contour, Drill, or Route.

Tool Number and Name Select a tool from the drop down tool selection box or enter a Tool number directly. This number will be reflected in the Posted Machine Code. Enter a name in the Tool Name box. This name will be commented into the Posted Machine Code file when Tool paths are saved. When selecting a tool from the tool library, tool parameters are automatically entered.

Rapid height The height at which rapid traverses occur . Enter a value the amount above the work piece (Z=0) to allow clearance for the tool when it is traveling to its next position. This value is always positive.

Spindle RPM The speed (In RPM) to set the spindle to. Use a value that reflects your cutters diameter and the material you are machining.

Plunge rate The rate to drop the tool or drill.

Coolant Select either On or Off.

Configuration Select a user defined configuration from the entry box and press Load. All saved parameters will be loaded. To save a new configuration, enter all tooling parameters first. Then enter a unique identifying name in the entry box and press save. If the configuration already exists, you will be asked to overwrite it. To delete a configuration, select it from the entry box and press delete.

Tool Number and Name Select a tool from the drop down tool selection box or enter a Tool number directly. This number will be reflected in the Posted Machine Code. Enter a name in the Tool Name box. This name will be commented into the Posted Machine Code file when Tool paths are saved. When selecting a tool from the tool library, tool parameters are automatically entered.

Rapid height The height at which rapid traverses occur . Enter a value the amount above the work piece (Z=0) to allow clearance for the tool when it is traveling to its next position. This value is always positive.

Spindle RPM The speed (In RPM) to set the spindle to. Use a value that reflects your cutters diameter and the material you are machining

Plunge rate The rate to drop the tool or drill.

Coolant Select either On or Off.

Configuration Select a user defined configuration from the entry box and press Load. All saved parameters will be loaded. To save a new configuration, enter all tooling

parameters first. Then enter a unique identifying name in the entry box and press save. If the configuration already exists, you will be asked to overwrite it. To delete a configuration, select it from the entry box and press delete.

Entry and Exits These settings are available for the contour function and for the finishing pass of the pocket function. By default, entry/exits will turn away from the contour. The side of which the entry/exit is located can be reversed by entering a negative value for the length or radius. Entering a negative value for a tangent entry/exit has no effect because a tangent lead has no side.

Arc Adds a quarter circle of the specified radius tangent to the beginning or end of a contour. Enter the radius.

Perpendicular Adds a perpendicular lead of the specified length to the beginning or end of a contour. Enter the radius.

Tangent Adds a tangent lead of the specified length to the beginning or end of a contour. Enter the length.

Creating Tool paths

Use these commands to create a Machine Code File from the DXF File geometry.

The Pocketing Function Definitions and Guidelines X-Y Step Size The amount to offset the next loop in the spiral pocket. It is automatically calculated if the diameter of the tool is known and the Tool Overlap in the options is >0. X-Y Step Size = Tool Overlap x Diameter. Enter a new value to override.

Pocket Depth The finished depth of the pocket. Enter a positive value. If depths were added using the Change Depths function, then only Boundaries with a depth of Zero will be set to this depth. All others keep their current depth.

Z Step Size The depth to cut each roughing pass. If the depth of a Boundary is greater than this value, multiple passes will be created.

Stock to leave The amount of material left over for the finishing pass.

Feedrate The machining speed to perform the pocketing operation.

Use Finishing Pass check box When checked the finishing parameters will be enabled. A finishing pass consists of removing material left by the roughing pass's). The Stock To Leave field in the roughing pass section does not have to be filled in. You should always use a finishing pass when your geometry has islands as the roughing pass will not completely trace the entire outline of the islands.

Finish Island / Boundaries / Both Islands only will only finish the islands during pocketing. Islands and boundaries will finish both islands and boundaries. Both + Bottom will also finish the bottoms and tops of islands and boundaries.

The Contouring Function

Contouring Depth The finished depth of the contour. Enter a positive value. If Depths were added using the Add Depths function then enter a value of zero here.

Z Step Size The depth to cut each pass. Entering a value less than the depth of a region will create multiple passes in Z Step increments until the final depth is achieved.

Feedrate The rate to perform the contouring operation.

Cutter Compensation (None) Setting cutter compensation to off will produce tool path segments that trace the selected regions exactly.

Cutter Compensation (Left, Right) Setting cutter compensation to Left will produce tool path segments that are offset by the tool diameter on the left side of Regions. Left or Right are calculated from the direction of the start of the first segment to the end of the first segment. DeskCNC will offset the toolpaths by the diameter of the tool on this side of the contour.

Cutter compensation (Cutout) Setting cutter compensation to cutout will offset all islands by the tool diameter on the inside first and then offset all boundaries on the outside. This will effectively cutout a selected region. It has no effect on open contours.

Working with STL Files

These tool path functions are used to create toolpaths from 3D STL files. For Definitions and Guidelines common to all Tollbooth functions go to Common tooling parameters.

Blank Size The Blank is the dimensions of the solid material from which the part is to be machined from. By default, it is set to the extents of the STL geometry. Decreasing the default Blank size will machine only a portion of the STL geometry. X1, Y1, and Z1 represent the bottom left corner while X2, Y2, and Z2 represent the top right corner of the Blank.

Male/Female Selecting Male will cut around the loaded geometry to the extents of the Blank size. Selecting Female will only cut the geometry and is equivalent to 'inverted' male machining.

Direction Tool paths can be created along the X or Y axis and follow a Zig Zag pattern. For best surface finish, the Finish pass should be opposite the Roughing pass.

Tool type Three different tool types can selected for machining - End, Ball, or Vee shaped tools. When a Vee tool is specified, the included angle is required. An angle of 180 represents an End mill.

X, Y, and Z Step The X Step size is the distance along the X axis for each segment of the tool path. The Y Step size is the distance along the Y axis for each segment of the tool path. The X and Y step sizes are automatically entered for a given tool diameter as specified under Options- X and Y Step size as a percentage of cutter diameter. The Z Step size is the depth for each pass.

Working with Gerber and Excellon Files

DeskCNC features trace isolation and hole drilling for creating prototype PCB boards from Gerber/Excellon files. It imports Extended Gerber (2.3 Leading Zero format) and Excellon (Trailing Zero Suppression) file formats. The trace isolation geometry can be saved as a DXF file format for processing by other CAD applications. In addition to the trace isolation capabilities, DeskCNC can create CNC g-code from Excellon standard drilling format files. The routing tool can pre-spot the through holes to improve the accuracy of the drilled holes, and to reduce the chances of breaking a drill due to wandering tip.

After the routing and drilling of the PCB, it can be cut out from the copper clad PCB material.

Importing Gerber and Excellon PCB Files DeskCNC is designed to import Gerber Photoplot files that follow the Extended Gerber file format. This is all that is necessary for Gerber II files. In addition, DeskCNC can load the Aperture files from Tango and Protel (PCB CAD Software) photo plots separately. When importing PCB files from Tango or Protel, always load the Apertures first before importing the Gerber Photoplot.

When importing Excellon Drill files, the Drill Definitions may or may not be included in the Excellon Drill file. In such cases, the Drill Definitions can be imported separately after the Excellon Drill file has been imported.

Creating Tool paths- After selecting Tool paths - Route, a dialog Window will be displayed.

Information
The number of selected traces, selected holes, and drill sizes are displayed. If a drill definition has not been successfully loaded, the number of missing drill definitions are also displayed.

Operation When a Gerber file is successfully loaded, the Route Operation will be available. Likewise, on successful loading of an Excellon file, the Drill operation will be enabled. Both Routing and Drilling can be created by selecting the Route & Drill operation.

Routing

Spot Drill Depth Spot drilling of the through holes with the routing tool can be accomplished by selecting this option. When enabled, enter the depth for the spot drilling, being careful not to go too deep with a large Vee cutter thereby increasing the intended size of the through hole.

Depth The depth for the Routing operation. Enter a positive value.

Feedrate The rate to perform the routing operation.

Tool Number and Name Select a tool from the drop down tool selection box or enter a Tool number directly. This number will be reflected in the Posted Machine Code. Enter a name in the Tool Name box. This name will be commented into the Posted Machine Code file when Tool paths are saved.

Rapid height The height at which rapid traverses occur . Enter a value the amount above the work piece (Z=0) to allow clearance for the tool when it is traveling to its next position. This value is always positive.

Spindle RPM The speed (In RPM) to set the spindle to. Use a value that reflects your cutters diameter and the material you are machining.

Plungerate The rate to drop the tool or drill.

Configuration Select a user defined configuration from the entry box and press Load. All saved parameters will be loaded. To save a new configuration, enter all tooling parameters first. Then enter a unique identifying name in the entry box and press save. If the configuration already exists, you will be asked to overwrite it. To delete a configuration, select it from the entry box and press delete.

Drilling

Clearance height The height at which rapid traverses occur . Enter a value the amount above the work piece (Z=0) to allow clearance for the tool when it is traveling to its next position. This value is always positive.

Depth The depth for the Drilling operation. Enter a positive value.

Feedrate The rate to perform the drilling operation.

Board Cut Out Margin The distance between the extents of the traces/pads and the outline of the PC Board.

Depth The depth for the Cut Out operation. Enter a positive value.

Plungerate The rate to drop the routing tool to initially pierce the PC Board material.

Feedrate The rate to perform the drilling operation.

Tool Number Select a tool from the drop down tool selection box or enter a Tool number directly. This number will be reflected in the Posted Machine Code.

Creating the toolpaths and Routing a PC Board Click the Create G-Code button to create the toolpaths. After the toolpaths have been successfully created, select the menu item Tool paths - Run Machine. You will be prompted to save the toolpaths. The toolpaths will then be automatically loaded and the **mode** switched to Machine to run your CNC Machine.

Working with Image Files

DeskCNC features a computer image to Tollbooth converter that is used to route, carve, or mill images using a CNC machine. Images are converted to Gray automatically on load for processing. DeskCNC includes a set popular filters for basic and advanced image manipulation. DeskCNC supports a variety of the most popular image formats including TIF, GIF, JPEG, PCX, BMP, RLE, ICO, CUR, PNG, EMF, WMF, TARGA, PPM, AVI. Portions of Images can be processed separately. It includes advanced raster to vector conversion for use in different engraving styles such as Topography.

Modifying the Image

Features

Invert Inverts all colors of the Image creating a Negative of the Image. Useful for switching between a Raised Carving or a Sunken Carving.

Resample/Resize Resizes the Image. Selecting 'Aspect Ratio' keeps the Width/Height ratio constant. Various resizing filters are available to improve image quality when increasing the image size. **Tip -** Increase the Image Size (In Pixels) to increase the XY Step Resolution (decrease the XY Stepovers) when Carving.

Crop Rectangle, Polygon, Ellipse, or 'Magic Wand' Selection tools are available for selecting areas of the image. Click and Drag the Left mouse button to define the Rectangle or Ellipse. Click a color of interest to invoke the Magic Wand. The Magic Wand will follow the outline of the selected color. Click the Left mouse button to define the edges of the polygon selection tool. Double click the Left mouse button to close the polygon. Once a selection has been made, the cursor changes to up/down arrows to move the selection window. When you are satisfied with the selected area, click on the Crop button to crop the Image to the selected area.

Modify Image Brings up a modify Image Dialog window. Most basic Image manipulation functions are available such as Brightness, Contrast, 3x3 Convolute Filters etc. The Color Depth can also be reduced to any number of colors from 2 to 255. Click OK to apply the changes to the main Image. Click the Preview button to apply the changes to the thumbnail image in the modify Image Dialog window. Depressing the Lock button locks the Preview and any changes made to the preview image will be displayed in real time.

Revert to Original Image Reloads the current image. Useful if too many image modifications have corrupted the Image beyond repair.

Carving Images

The following is available by selecting the Carve Tab.

Size X Enter the width of the finished carving. If the "Keep aspect ratio" box is checked, then the height will be entered automatically.

Size Y Enter the height of the finished carving. If the "Keep aspect ratio" box is checked, then the width will be entered automatically.

Step X or Step Y Enter the step over along the X or Y axis. If the "Keep aspect ratio" box is checked, then the other Step value will be entered automatically and is equal to the Step value entered. The Step can not be less than one pixel in length which is the default (the value will turn red to indicate that you have entered a value that is not supported). Increasing the Step values will reduce the file size and time required to route the image but will give a product with lower detail. To increase the resolution of the Carving, Resample/Resize the image using any of the available resample filters. Better results are obtained from the more advanced filters when increasing the image size. **Depth** Enter the maximum depth that you want to route. This depth is assigned to pure Black. If you image does not contain pure Black, then the finished Carving will have a maximum depth that is less than the entered depth.

Depth / Pass Enter the maximum depth the machine should cut in a single pass. Each pass will be optimized and unnecessary rapid moves will be automatically filtered out.

Feedrate Enter the feed rate for the Tollbooth file.

Rapid Height Enter the clearance height that all rapid moves will go to while moving to the next line.

Gouge Control Check this box if you want gouge control to be applied to the toolpaths. When gouge control is active, the cutter will not gouge a single black pixel in an area of white. Three tool types are available- Ball, End, and Vee cutters. It is always recommended to apply gouge control while carving images.

Gouge Control - Tools Selecting a Tool number from the drop down list will automatically set the tool parameters from the tool database. These may be changed or altered for user defined tools. Tool definition parameters must be set for Gouge Control and for viewing a rendered image of the tool paths. Different parameters are needed for each tool style. For Vee style cutters, the included angle must be set but for Ball cutters the ball radius must be set.

Machining Direction Both X and Y machining directions are available. A roughing pass in one direction followed by a finishing pass in the other direction yields best results.

Cut Margin Selecting this option will remove unnecessary machining 'around' the 'interesting' part of an image. For this option to be effective, the image of interest should be surrounded by pure Black. For example, a white circle in a pure black background will give two different results depending on whether the Cut Margin selection is checked. **Create** Creates the Carving based on the current settings. The View automatically changes to wireframe. **Machine the Tool paths** Clicking this button prompts you to save the Tool paths to a DNC file and loads the resulting file into the DeskCNC Machine Controller mode. From here, you can immediately run this file on your machine from Windows (optional Controller required).

Engraving Images

Width Enter the Width of the Engraving. The vector toolpaths that are created will be scaled based on this value. Aspect ratio is preserved and can not be changed. The default width is the Image width based on its Dots per Inch property.

Cutting Depth Enter the Depth of the Engraving. The vectors that are created will be based on this depth. When using the Topography feature and Region Depths are 'Based on Color', this depth represents the cutting depth of pure Black regions. The depth of lighter regions will be scaled accordingly under these conditions.

Z Level of Image When creating an Engraving at a single depth, this represents the depth of color to operate on. Valid range for this parameter is 1-99 and denotes the percent of the image depth. 0 % equates to pure White while 100 % equals pure Black. Therefore, the default value of 50% will vectorize those portions of the image that are 'pure' gray (half way between black and white).

Z Step The Z Step parameter is used when making an Engraving under the Topography option. The Z Step is used to calculate the next depth to operate on. For example, if the Cutting Depth is 1 and the Z Step is 0.1 then toolpaths will be created in 0.1 increments from 0 to 1.0. There will be 10 'layers' of vectorised toolpaths. If the 'Based on Color' option is checked, then each 'layer' will have its cutting depth scaled accordingly. Otherwise, all 'layers' will have the same depth set to the Cutting Depth.

Feedrate Sets the current Feedrate that will be used in the Tollbooth File.

Rapid Sets the current Rapid or Clearance height that the cutter will travel at while going to the next region.

Style Sets the style of the created vector regions. <u>Single:</u> Creates vector regions at a single depth. The depth (in terms of Image Color) at which the vectors are determined is set in the Z Level field. The actual Cutting Depth of the resultant toolpaths is set in the Cutting Depth field. <u>Topography:</u> Creates regions at multiple depths thereby creating a Topographical Map of the Image. Each level of the Topography Map is created from the Z Step field. The created regions can be set to the same Machining depth by setting the Region Depths to Single Depth and entering a Cutting Depth. If Region Depths is set to Based on Color then the depth of the created Regions will be scaled to the value set in the Cutting Depth of the Image.

Reduce Removes points from the created regions. Helps to reduce the overall number of line segments created. Values can be from 1 to infinity. Example - If set to 2 then DeskCNC will keep a point, remove the next 2 points, keep the next point, remove the next 2 points etc. of a region.

Curve Fit When checked, each region created will be smoothed by fitting the regions' points to a B-Spline. The value entered is the number of segments that will be added

between each point in the region. By using Reduce and Curve Fit together, regions can be smoothed out considerably and the number of toolpaths reduced.

Filter Filters out or removes smaller regions. All regions that have fewer than the number of Tool paths entered will be removed.

Create Creates vector regions based on the current settings. The View automatically changes to wireframe.

Clear Clears all Vector Regions. If regions are not Cleared, then the created toolpaths will be added to any existing toolpaths.

Undo Removes the last set of created regions.

Machine the Tool paths Clicking this button prompts you to save the Tool paths to a DNC file and loads the resulting file into the DeskCNC Machine Controller mode. From here, you can immediately run this file on your machine from Windows (optional Controller required).

Image Viewing Options

Image The Original image is displayed and any toolpaths are hidden. Left clicking on the Image will zoom in while right clicking will zoom out.

Wireframe Displays any toolpaths that have been created using the Carving or Engraving features of DeskCNC. While viewing the Wireframe mode, dragging the left mouse button will orbit the toolpaths in dynamic 3D fashion. Holding the Shift button down and dragging the left mouse button will Pan the toolpaths while holding the Ctrl key down and dragging the left mouse button will Zoom in/out.

Rendered Displays a 3D Solid based view of the toolpaths. For proper display, the tool number must be defined in the tool library. The quality of the rendered display is set by the OpenGl Resolution as set in Options. A lower value will give a better display at the expense of display speed.

Machine Mode
The G-Code Editor and Display

Overview

The G-Code editor is capable of interpreting and plotting over 50,000 lines per second in 3D on a P600 Mhz computer system. It has optional syntax highlighting for displaying the g-code file. It can display the entire g-code file or step through portions of the file in 3D using different colors to represent the different types of interpolation (rapid, linear, CW, and CCW). The finished part can be displayed as a rendered solid if your system supports OpenGL.

Mouse Movements

• **Panning** - While holding down the Shift button, click and drag the left mouse button to dynamically pan the currently loaded file. 'Autoredraw' must be set to 1 in Options for the file to automatically be redrawn. When g-code geometry is moved, the extents of the display will change accordingly.

• **Zooming** - While holding down the Control button, click and drag the left mouse button to dynamically zoom the currently loaded file. 'Autoredraw' must be set to 1 in Options for the file to automatically be redrawn. When g-code geometry is zoomed, the extents of the display will change accordingly.

• **Rotating in 3D** - If the 3D Isometric view is selected under the View menu then dragging the displayed geometry with the left mouse button will dynamically rotate the interpreted g-code file. 'Autoredraw' must be set to 1 in Options for the file to automatically be redrawn. Otherwise, just the machine extents and the display grid will rotate initially. Clicking on one of the run buttons will update the display.

The Display Views under the View Menu

• **Rendered** - Sets the display mode to 3D OpenGl rendered solid part mode. In this mode, the part is seen machined on screen as a 3D solid. Only available on systems that support OpenGL. This mode supports 'Gouge Checking'. Any move that gouges the finished part will be highlighted in bright yellow. It is necessary to define each tool used in the Tool Library. If the tool is undefined a default .25" end mill is used. It is not recommended to leave DeskCNC in this mode while machining a part (use wireframe mode).

• **Wireframe** - Sets the display mode to the default wireframe mode. This mode shows the toolpaths as single wires.

• **Isometric 3D** - Sets the display mode to 3D. When in 3D mode, the geometry can be rotated in 3D by dragging the left mouse button. Not available in Rendered mode.

• **Top** - 2D viewing mode looking down the Z axis.

• **Front** - 2D viewing mode looking down the Y axis. Not available in rendered mode.

• **Zoom Window** - Click and drag a box around the geometry to zoom in on specific geometry. Works in 2D and 3D mode but not in Rendered mode.

• **Clear Screen** - Clears the display screen.

The Editor Buttons

• **Clear** - Clears the display area.

• **Next** - Interprets and displays the next line of g-code in the loaded g-code file.

• **Next 10** - Interprets and displays the next 10 lines of g-code in the loaded g-code file.

• **Next 100** - Interprets and displays the next 100 lines of g-code in the loaded g-code file. • **Next 1000** - Interprets and displays the next 1000 lines of g-code in the loaded g-code file.

• **To Cursor** - Interprets and displays up to the position of the cursor in the loaded g-code file.

• **From cursor** - Interprets and displays from the current cursor position to the end of the loaded g-code file.

• **All** - Interprets and displays the entire loaded g-code file.
• **Reset** - Resets to the beginning of the loaded g-code file.

The Coordinate Display

While in Edit Mode, the coordinate display will change to the last point plotted on the display. To reset any or all coordinates, click on the appropriate x, y, z, all buttons in the coordinate display area. When in Machine Mode, the displayed coordinates represent the absolute position of the controller board. Under extreme use, the coordinate display may not update in real time.

The G-Code edit area

Here you can edit the actual g-code as loaded from a file. Several commands are available by right clicking anywhere inside this edit box and selecting from the pop-up menu.

• **Undo** - Undoes the last edit.

• **Font** - Brings up a dialog window to select the Font and Font Size used in the editor.

• **Search** - Brings up a dialog window to search and replace text in the editor.

• **Select All** - Selects the entire file for cut and paste operations into other programs.

• **Renumber** - Brings up a dialog window to Add/Remove or Renumber the g-code file. Add will add block line numbering starting at the value in Start At and incrementing by the value in Increment. Remove will remove line numbering while Renumber will renumber existing line numbering by the Start and Increment values. An

option is available to add a space after the line number. The line number takes the form of N<number>.

• **Show Syntax** - When selected, Show Syntax will highlight portions of the loaded g-code to make it easier to read. G and M codes along with other commands are set in bold colorful text to make them stand out better. Turning Show Syntax off will display the file in standard black and white text fashion.

> • **Save** - Brings up a dialog box to save the currently loaded g-code file.

Menu Items

Home

The Homing functions will take place at the Homing Velocity and Homing direction as specified in the Options.

• **Home X** - Homes the X axis only.

• **Home Y** - Homes the Y axis only.

• **Home Z** - Homes the Z axis only.

• **Home All** - Homes All axis at once.

Controller

• **Start From Line** - Enter the absolute line number of the loaded G-Code file to start from when running the file. This is the Line Number at the far left and Not the Block Line Number (Nxxx). The GO button will display the Starting Line in brackets.

Control Mode Commands

The DeskCNC software can be placed in control mode by selecting the Mode - Control Mode menu item. This mode of operation is only available when the external controller board is attached and communicating with the software.

The MDI Command Box

The MDI Command box allows executing a single line of g-code text. The text is executed immediately when the Enter key is pressed while the cursor is inside the MDI Command box. If the SPEEDKEYS are set in Options then pressing F1 - F9 will automatically enter commonly used g-code text into the MDI Command box.

The Auxiliary Output Buttons

Four auxiliary outputs are available to toggle on/off solid state relays for additional functions. The Controller will output a logic high (+5v) or a logic low (0v) when a button is pressed. Look at the pinout diagram and the supported g-codes for more information.

The Coordinate Reset Buttons

Above the coordinate display are four buttons to zero the x, y, z, or all axes. Pressing any of these buttons will zero the display coordinates and send a command to zero the absolute position stored in the external controller.

The Jog Buttons

The Jog Distance can be changed by selecting a different distance from the drop down Jog box. These jog distance values can be changed from Options. You may also enter any valid Jog Distance directly in the drop down Jog box.

• **Home** - Homes all axes at once. It is possible to Home an individual axis by selecting the Home - Home - <axis> menu item. Homing Velocity, Offset, and Direction are set in Options.

• **Z+** - Jogs up (Z+) the specified Jog Distance. Also available by pressing the Page Up key.

• **Z-** - Jogs down (Z-) the specified jog distance. Also available by pressing the Page Down key.

• **X-** - Jogs left (X-) the specified jog distance. Also available by pressing the Left Arrow key.

• **X+** - Jogs right (X+) the specified jog distance. Also available by pressing the Right Arrow key.

• **Y-** - Jogs forward (Y-) the specified jog distance. Also available by pressing the Up Arrow key.

• **Y+** - Jogs back (Y+) the specified jog distance. Also available by pressing the Left Arrow key.

• **Go** - Start running the loaded g-code file. Also used to continue motion after a Pause. Any key press will pause motion.

• **Reset/EStop** - Will immediately stop processing a loaded g-code file and send a command to the external controller to execute an EStop. The controller will then act as if a Limit switch has been triggered - an immediate stop with no ramp down.

• **Pause** - Will pause motion. Any key press will force a Pause. Continue motion by clicking on the Go button.

Information Area

• **Tool** - Displays the current tool number as being read by the interpreter. This is not necessarily the current tool. The interpreter will be 50 moves ahead under normal operation.

• **Jog** - This is the drop down Jog Distance box. The value in this box will be the distance to move an up, down, left, right, back, or forward jog.

• **Feed Rate** - Displays the current Feed Rate as being read by the interpreter. This is not necessarily the current tool. The interpreter will be 50 moves ahead under normal operation.

• **Override** - Sliding this control to the right of center will increase the feed rate. The amount of increase in feed rate is not a percentage of the actual feed rate but is a constant timing factor sent to the external controller. Selecting Controller - Zero Feed Override will reset this control to its center position. Sliding the control to the left will decrease the feed rate in a similar way.

• **Spindle** - Sliding this control to the right of center will increase the feed rate. The amount of increase in feed rate is not a percentage of the actual feed rate but is a constant timing factor sent to the external controller. Selecting Controller - Zero Feed Override will reset this control to its center position. Sliding the control to the left will decrease the feed rate in a similar way.

The Status Bar

• **First Panel** - Displays comments for your information.

• **Second Panel** - Displays the number of lines of the currently loaded g-code file.

• **Third Panel** - Displays the filename of the currently loaded g-code file.

• **Fourth Panel** - Displays the percentage complete of running a g-code file. This is the percentage of the file interpreted by the interpreter and does not reflect any moves currently executing in the external controller.

• **Fifth Panel** - Displays the coordinates of the cursor in the Tollbooth Display area.

DXF Pocket Tutorial

This short tutorial will introduce you to the most used commands in DeskCNC and will only take a few minutes. Select the menu command **File - Open DXF** From the list choose PockTut.dxf PockTut.dxf will be loaded and all closed regions will be displayed in a light gray color. These color choices may be changed in Options. The second panel of the status bar will read All 265. This indicates you are in the View All screen and there are 265 total segments in PockTut.dxf. The last panel will display the coordinates of the mouse cursor.

Right click the mouse button and select "Select All"

Select - Select All The Boundaries will be displayed in blue and all Islands will be displayed in red.

Select Edit - Information A small information box is displayed at the beginning of the first entity in each region. The first number denotes the region number and the optional second number indicates the region that this island belongs to.

Select Tool paths - Pocket The Pocket dialog window is displayed.

Enter the following parameters. Tool No - Enter 1 Tool - Enter 3/8" End Mill Tool Diameter Enter .375 Pocket Depth - Enter .5 X-Y Step Size Automatically calculated from Options - Tool Overlap. Z-Step Size - Enter .25. Stock to Leave - Enter .05 5 Hundredths will be left for the finishing pass. Rapid Height - Enter .1 Rapid moves will occur .1 inches above the workpiece. Spindle Speed - Enter 1500. Feedrate - Enter 10. Plungerate - Enter 7.5.

Now press the Pocket button.

Select View - Show Directions An arrow is displayed at the midpoint of the first tool path segment of each loop. If the Mark All Entities option is selected then an arrow will be displayed at the midpoint of all the tool path segments.

Select Tool paths - Run Machine.

Accept the default filename of PockTut.dnc and press Enter. DeskCNC will go into Machining Mode and automatically load the PockTut.dnc tool path file.

Select Mode

Select Edit to plot the file on screen only. Select Machine mode to run the machine

Click the GO button

DeskCNC will start machining the file

DXF Contour Tutorial

This short tutorial will introduce you to the most used commands for Contouring in DeskCNC and will only take a few minutes. Select the menu command **File - Open DXF** . From the list choose PockTut.dxf PockTut.dxf will be loaded and all closed regions will be displayed in a light gray color. These color choices may be changed in Options. The second panel of the status bar will read All 265. This indicates you are in the View All screen and there are 265 total segments in PockTut.dxf. The last panel will display the coordinates of the mouse cursor.

Right click the mouse button Right click the mouse button and select "Select Region"

Select - Select Region. Click on the first clover towards the bottom. The clover will turn to a blue boundary and the diamond inside of it will change to a red island. Click on the second clover.

Select Edit - Information A small information box is displayed at the beginning of the first entity in each region. The first number denotes the region number and the optional second number indicates the region that this island belongs to.

Select Tool paths - Contour The Contour dialog window is displayed.

Enter the following parameters. Tool 0 Tool Name- 1/4" 2 Flute HSS Standard End Mill Contouring Depth - Enter .5 Z-Step Size - Enter .25. Rapid Height - Enter .1 Rapid moves will occur .1 inches above the workpiece. Spindle Speed - Enter 2500. Feedrate - Enter 8. Plungerate - Enter 6. Select the Cut Out option in the cutter compensation frame.

Now press the Contour button.

 The tool path segments are created. Islands are contoured in the inside before boundaries are cut out on the outside Leaving the two Clovers effectively Cut Out.

Select View - Show Directions An arrow is displayed at the midpoint of the first tool path segment of each loop. If the Mark All Entities option is selected then an arrow will be displayed at the midpoint of all the tool path segments.

Select Edit - Information A small box is displayed at the beginning of each tool path loop displaying the loop number.

Select Tool paths - Run Machine.

Accept the default filename of PockTut.dnc and press Enter. DeskCNC will go into Machining Mode and automatically load the PockTut.dnc tool path file.

Select Mode

Select Edit to plot the file on screen only. Select Machine mode to run the machine

Click the GO button

DeskCNC will start machining the file

DXF Drill Tutorial

This short tutorial will introduce you to the most used commands for Drilling in DeskCNC and will only take a few minutes. Select the menu command **File - Open DXF** From the list choose DrillTut.dxf DrillTut.dxf will be loaded and all closed regions will be displayed in a light gray color and all open regions will be displayed in an even lighter gray color. These color choices may be changed in Options. The second panel of the status bar will read All 15. This indicates you are in the View All screen and there are 15 total segments in DrillTut.dxf. The last panel will display the coordinates of the mouse cursor. Right click the mouse button and select "Select Empty Circles"

Select - Select Empty Circles. All the circles will turn to blue boundaries.

Select Tool paths - Drill The Drill dialog window is displayed.

Enter the following parameters. Tool - 1 Tool - 5/16" Drill Bit Drilling Depth - Enter .375 The holes will be .375" Deep. Rapid Height - Enter .1 Rapid moves will occur .1 inches above the workpiece. Spindle Speed Enter 600 Plungerate - Enter 10.

Now press the Drill button. The Save As dialog window will be displayed with the default filename of DrillTut.Dnc. It will be saved in the DNC sub directory of the DeskCNC folder. Enter a different filename if desired or press the Save button. If the file already exists from a previous tutorial you will be prompted to overwrite it. If so, click Yes. The tool path segments are created.

Select Edit - Information A small box is displayed at the beginning of each drill point displaying the sequence number.

Select Tool paths - Run Machine.

Accept the default filename of DrillTut.dnc and press Enter. DeskCNC will go into Machining Mode and automatically load the DrillTut.dnc tool path file.

Select Mode Select Edit to plot the file on screen only. Select Machine mode to run the machine

Click the GO button

DeskCNC will start machining the file

DXF Advanced Tutorial

This short tutorial will introduce you to some of the advanced commands in DeskCNC and will only take a few minutes. Select the menu command File - Open DXF From the list choose Advanced.dxf Right click the mouse button and Select All

Select - Select All All Boundaries will be displayed in blue and all Islands will be displayed in red.

Select Edit - Information A small information box is displayed at the beginning of the first entity in each region. The first number denotes the region number and the optional second number indicates the region that this island belongs to. In parenthesis will be advanced information such as Wall profile.

Select Edit - Change Depths The Depth dialog window will pop up. Enter a Depth of 1. Select 'Boundaries' and Press 'Apply'. All Boundaries (in Blue) will have a depth of 1. The information marker for the outer most boundary will now show a depth of 1.

Enter a Depth of 0.5. Select Islands and Press 'Apply'. All Islands (in Red) will now have a depth of .5. The information marker for each Island will now show a depth of 0.5.

Enter a Depth of 2.0. Select 'Single Region' and Enter the region number of the inner circle (most likely 3). Press Apply. The information marker for the inner circle will now show a depth of 2.0. The inner circle will change color to the Boundary color (Blue) indicating that it is no longer an Island. Its depth was changed to a deeper depth (Its depth is 2 compared with the outer region's depth of 1.0) than its parent region and thus becomes a boundary to be machined and not an Island to be avoided. Press the cancel button to leave the Depths dialog window. Select Edit - Information to turn off the information markers.

Select Tool paths - Pocket The Pocket dialog window is displayed.

Enter the following parameters. Tool No - Enter 1 Tool - Enter 1/4" End Mill Tool Diameter Enter .25 Pocket Depth - LEAVE THIS AT 0. Depths have been previously set. X-Y Step Size Automatically calculated from Options - Tool Overlap. Z-Step Size - Enter 1.0. Stock to Leave - Enter 0 Rapid Height - Enter .1 Spindle Speed - Enter 1500. Feedrate - Enter 10. Plungerate - Enter 7.5. Now press the Pocket button.

Select Edit - Information Turn information markers back on.

Select Edit - Add Walls. The Walls dialog window will pop up. Enter 3 for the region (or the region number associated with the inner circle). Select the first profile (A circular profile). Enter a Delta of 1. This will give a full semicircular wall. It is equivalent to the difference in depths of the region (depth of the inner circle is 2) and its parent (its parent is the outer circle and has a depth of 1). Press the cancel button to leave the Walls dialog window. Select Edit - Information to turn off the information markers.

Select Tool paths - Pocket The Pocket dialog window is displayed. Leave all parameters the same except change the Z Step Size to 0.25

Now press the Pocket button.

Use the mouse to view the toolpaths at different angles. Notice the inner circle starting to take the shape of a semicircle or 'bowl'.

Select Tool paths - Run Machine.
Accept the default filename of Advanced.dnc and press Enter. DeskCNC will go into Machining Mode and automatically load the Advanced.dnc tool path file.

Select Mode
Select Edit to plot the file on screen only. Select Machine mode to run the machine
Click the GO button
DeskCNC will start machining the file

STL Tutorial

This short tutorial will introduce you to the most used commands for Prototyping in DeskCNC and will only take a few minutes. Select the menu command File - Open STL From the list choose Teapot.Stl.

Select Tool paths - Create The Prototype dialog window is displayed.
Enter the following machining parameters. Rapid Height - Enter 1 Roughing Direction Select the X axis Roughing Tool Name 6 mm Ball Mill Roughing Tool 1 Roughing Diameter Enter 6 Rough Tool Type Select 'Ball' X-Step Size Enter .6 (Automatically y Entered). Y-Step Size Enter .6 (Automatically y Entered). Z-Step Size Enter 10 (Each rough pass will be a maximum of 10 mm deep). Spindle Speed - Enter 5000. Feedrate - Enter 200. Plungerate - Enter 150. Finishing Direction Select the Y axis Finishing Tool Name 6mm Ball Mill (Same as Rough pass). Finishing Tool 1 Finishing Diameter Enter 6 Finish Tool Type Select 'Ball' X-Step Size Enter .6 (Automatically y Entered). Y-Step Size Enter .6 (Automatically y Entered). Z-Step Size Enter 50 (Finish in one pass). Spindle Speed - Enter 5000. Feedrate - Enter 200. Plungerate - Enter 150.
Now press the Machine button. The tool path segments are created.
Select Tool paths - Run Machine.
Accept the default filename of TeaPot.dnc and press Enter. DeskCNC will go into Machining Mode and automatically load the TeaPot.dnc tool path file.
Select Mode Select Edit to plot the file on screen only. Select Machine mode to run the machine
                    Click the GO button DeskCNC will start machining the file

The G-Code Interpreter

DeskCNC supports the following G and M Codes

G0 rapid positioning

G1 linear interpolation

G2 circular (clockwise XY Plane Only)

G3 circular (counterclockwise XY Plane Only)

G4 Dwell

G10 coordinate system origin setting

G17 xy plane selection

G20 inch system selection

G21 millimeter system selection

G40 cancel cutter diameter compensation

G41 start cutter diameter compensation left

G42 start cutter diameter compensation right

G43 tool length offset (plus)

G49 cancel tool length offset

G53 motion in machine coordinate system

G54 use preset work coordinate system 1

G55 use preset work coordinate system 2

G56 use preset work coordinate system 3

G57 use preset work coordinate system 4

G80 cancel motion mode (including any canned cycle)

G81 drilling canned cycle

G83 chip-breaking drilling canned cycle

G85 boring, no dwell, feed out canned cycle

G86 boring, spindle stop, rapid out canned cycle

G87 back boring canned cycle

G88 boring, spindle stop, manual out canned cycle

G90 absolute distance mode

G91 incremental distance mode

G92 offset coordinate systems

G92.2 cancel offset coordinate systems

G93 inverse time feed mode

G94 feed per minute mode

G98 initial level return in canned cycles

G99 R-point level return in canned cycles

M0 program stop

M1 optional program stop

M2 program end

M3 Spindle on Clockwise

M4 Spindle on Counter Clockwise

M5 Spindle Off

M6 tool change

M7 Mist Coolant On

M8 Flood Coolant On

M9 Coolant Off

It is not my intention to explain in detail the above codes, but hopefully I can give you enough information to get you over the initial stages of manually editing the g-code (.dnc) files that Deskcnc uses in its machining process. There is plenty of detailed information available on the Internet or elsewhere, and the best way for you to learn is by experimenting in altering your existing code or producing new.

A common way of producing g-codes within Deskcnc is by inputting a dxf or other drawing file. The resulting codes may need editing, in particular if you have appended a number of files, so it is essential that you have at least a basic understanding of g-codes.

Each line of code is to be considered as a block. This block contains at least one letter followed by a numeric field which refers to the value to be assigned to that letter or function. Comments can be added and are enclosed in brackets. You can relatively easily write correctly formatted g-codes, or edit existing g- codes with a standard ASCII text editor, such as windows notepad.

Spaces or tabs are ignored, but their use increases readability. Blank lines are also ignored. The code is case insensitive, upper or lower case characters are translated the same. If you append a number of files, you will find that the machining stops when it hits the first M2, so you will probably want to delete these from the files, (leave the last one, however) and possible the preceding and following G0 x0 y0 z0 blocks or whatever.

Line numbers begin with an N. Within deskcnc this line number is not referred to or otherwise used, but it helps you to trace errors or edit the code, or track the machining process (Note that the line highlighted in the small edit window at the bottom of the machine screen is not the line being machined at the moment.)

Many codes are modal, that means they are active until they are changed, even outside of their block. For example the line of code g01 Z-5.00 will set the z axis to -5.00. Subsequent moves such as g01 x5 y25.44 will move the cutter from where it was to x5, y25.44 at a depth of 5.00. (It is usual to set the top surface of the workpiece as Z0, thus any cutting into the work will be given by negative Z values.)

The feed rate is also modal, and like other commands it applies to its current block and subsequent ones. Thus G01 x5.0 f150 will move the cutter at a speed of 150 to a new position at x5.0 subsequent g01 moves will be at the same speed, until another 'f' value is entered. Feed rate does not apply to G00 moves, of course.

For circular moves, G2 and G3, as well as the X,Y end of arc coordinates there will either be an I and J value, or an R value. The example below demonstrates this use. I, J is the distance from the start of the arc to the centre, or R is the radius.
N10 G00 z2.00
N20 G00 x20 y20
N30 G01 z-5 f200
N40 G02 x30 y10 I0 J-10 f300
N50 G00 z2.00

This will move the cutter above the work surface, then go to x20, y20 at rapid speed, then feed down at a rate of 200, cut a clockwise arc of radius 10 (centred on x20,y10) to a position x30,y10 at a feed rate of 300 then raise the cutter to clear the work piece again. The line N40 could also be written as N40 G02 x30 y10 r10 f300.

M6 will normally stop the machine to allow a tool change. You can write your own macro code to move the tool to the appropriate position, (see the software help documentation). The results of M commands for coolant are brought out to terminals on the controller board, and you can use relays or whatever to switch coolant on or off, or use these commands to control other functions.

The modal drilling canned cycle is entered in the form
G81 x10 y10 z-15 r-10 f75
X20 y20
X30 y30
G00 z2

This will drill three holes, at x10,y10 x20,y20 x30,y30 to a depth of 15. The r value is the start depth of drilling, so the above code would apply to a surface which had already been milled out to a depth of more than 10 (assuming surface was at z0.) The feed rate (75) will only apply to the drill macro, and the machine will revert to the previously feed rate setting after the holes have been drilled

G83 is the chip breaking version, it will withdraw the drill to clear swarf, and is useful for deep holes. The code looks like
G83 x10 y10 z-25 r1 q4 f75
This is similar to code G81, except it will withdraw the cutter every 4 units of drilling, i.e. It will go to x10,y10,z1 then go to z-3, withdraw, go to z-6, withdraw, go to z-10, withdraw, and so on. This is often referred to pecking, for obvious birdlike reasons.

It is suggested that you experiment with the codes that you are likely to use, so that you gain a full understanding of how your machine behaves. It is simple to write fragments of code, or alter existing code with the editor, in the machine mode, and immediately test the effect by running the code. The results can be seen on screen, provided the controller board is powered up, you need waste no material in testing.

appendix 4

Fourth axis and lathe setup

The fourth axis is a recent addition to DeskCNC software, and will be discussed in greater detail at a later date.

The software author has stated (as of 2nd Jan 2004) "The 4th axis is working for 'Indexing' and 'Slaving'. It does not yet calculate a new feed rate based on the distance of the cutter from the Rotary Center. We will be releasing a 4 axis function to create GCode for Foam Cutting machines (basic 4 axis Linear)."

<div align="right">**appendix 5**</div>

Setting backlash

Backlash settings are not working properly,so set all values for backlash to 0

It is always best to eliminate backlash and general slackness at source i.e. ensure that gib strips and slides are adjusted correctly, and backlash in lead screw assemblies is eliminated. However, for various reasons it is not always possible to remove backlash completely.

Deskcnc has backlash values for each axis, which if set correctly can improve the accuracy of the system. The setting is applied for each change of direction of the particular axis. Basically, when the motor direction of rotation changes, it adds a preset number of steps to its next movement. It will not compensate for nonlinear errors, such as often occur on older lathe leadscrews close to the headstock.

Quick Setting. This method is not for the purists! Ensure that no power is applied to the stepper motors. Manually turn the leadscrew a few turns clockwise, then gently turn anti-clockwise, and note the angle that the handwheel is turned before you feel the movement taking up the weight of the table or slide. Repeat a few times, for both clockwise/anticlockwise directions, and take the average result. The value to be entered will be the number of steps required to turn the leadscrew that angle.

Example calculation - Assuming that the machine has a leadscrew of 5 mm pitch, and step size of 200 steps per mm. After testing you find that 'slack angle' is about 45 degrees. The steps per rev will therefore be 200 x 5 = 1000 steps/rev. Now, 360degrees/45degrees = 8, so the backlash setting will be 1000/8 steps =125 steps.

Precise setting. Use a dial gauge on a rigid stand. Attach the dial gauge to the cutter axis (usually the z axis), such that it can be used to measure movement in the table. You may need to fix a block to the table to probe against for determining the x any y values of backlash. Consider one axis at a time, of course. Set the jog value to something less than the range of the dial gauge, but more than the estimated backlash. 5 mm is usually OK. Set the gauge to point to the right towards the block fixed to the table. Jog left and adjust the gauge or block such that the gauge probe is depressed by 6 mm, say. Note the exact gauge reading. Now jog right, and note the gauge reading. If the backlash setting is correct, then the gauge reading will be 5 mm less, at 1 mm. Keep jogging left and right, adjusting the backlash step setting for minimum deviation from 5 mm. You will be unlikely to get the reading exactly and consistently constant, due in part to the step errors of the stepper motors. Repeat for the other axes.

Testing. Having set the backlash compensation, you can test the x/y accuracy by contouring out a circle, and pocketing a circle. The pocket diameter will need to be a thou or so larger than the contoured circle so that the contoured circle will just fit inside the pocket. If you hold the workpieces up to the light, then you should be able to see an even join between the two pieces around the edge of the circle. Rotating one piece inside the other should not vary this clearance gap.

Appendix 7
Software Updates

Build 50 Mar 9, 2004
Bug Fixes
1.   4 Axis Linear Control.
Features Added
1.   External Pause enabled in Setup (Firm V1.40).

Build 49 Feb 26, 2004
Bug Fixes
1.   Backlash Comp.

Build 47 Feb 19, 2004
Bug Fixes
1.   Backlash Comp.
Features Added
1.   8 Com Ports selectable.

Build 46 Feb 02, 2004
Bug Fixes
1.   Gouging of parts under extreme conditions.
Features Added
1.   Acquire from Video Camera.

Build 44 Jan 18, 2003
Bug Fixes
1.   DXF Export to ACAD.
2.   Language File hot keys for Zeroing the axes.
Features Added
1.   Visual Tool Location.

Build 43 Dec 30, 2003
Features Added

1.   Extended input to 4/5 decimal places.

Build 42 Dec 18, 2003
Bug Fixes
1.   DXF Export to ACAD.
Features Added
1.   XY or Z axis can be slaved to the A axis terminals (requires Firm V1.39).

Build 41 Nov 29, 2003
Bug Fixes
1.   Image Page Fault Crash.
2.   STL Machining.
3.   Single Step Direction on reversed axis.
Features Added

Build 40 Nov 26, 2003
Bug Fixes
1. Image Color Reduction activated.
2. WCO's reset before Homing.
3. Unwanted A axis rotation.
Features Added

Build 39 Nov 5, 2003
Bug Fixes
1. Language Files updated.
2. Added Soft Home, G28, G30 functions in menu Controller.
3. Added ZXY or ZYX Homing Sequence option in Machine Parameters.
4. Added Linear or Rotary A option in Machine Setup.
5. Added Rendering Color in Options;
Features Added
1. Set Coordinates default to current value.
2. Append File I/O Error;

Build 38 Oct 21, 2003
Bug Fixes
Features Added
1. Seperate Grid Size values for G20 and G21.

Build 37 Oct 15, 2003
Bug Fixes
1. Language Files Updated.
2. Cont Jog fixed for large machines (again).
Features Added
1. Slovenian, Russian Language added.
2. Probe Tip Compensation added.
3. Added separate move, rotate, translate buttons

Build 36 Sept 4, 2003
Bug Fixes
1. Language Files Updated.
2. Cont Jog fixed for large machines.
Features Added
1. Language Selectable in Options.
2. New Button Layout.

Build 34 Sept 4, 2003
Bug Fixes
1. Language Files Updated.
Features Added
1. Edit/View Excellon Drill Definitions.
2. Mirror Gerber along Y axis.
3. Open Point Cloud data file and auto convert to STL.

DeskCNC Manual

Build 32 Aug 29, 2003
Bug Fixes
1.   Language Files Updated.
2.   Z Axis Jogging Corrected.
3.   Tool Length Sensor and Tool Change Macro Corrected.

Build 31 Aug 15, 2003
Bug Fixes
1.   Will not close without Controller connected.
Features Added
1.   Image file can be saved after modification.
2.   Language file added.
3.   STL's scaled 1/25.4 on load when using Inch mode.

Build 28 July 23, 2003
Bug Fixes
1.   'r' in Stick40 TT Font.
2.   Edit no longer disabled when Add Text function invoked.
3.   Tool Display not updated when Tool > Defined Tools.
4.   Scan extents corrected when an XYZ axis is reversed.
Features Added
1.   Lim/Probe Terminal can be set for NC, NO, or Disabled for Digitizing.
2.   Max Z Speed settable for Digitizing.

Build 25 July 17, 2003
Bug Fixes
1.   Moved Tool Library to separate area in Setup.
2.   EMT more intuitive.
Features Added
1.   WCO's set in Setup.
2.   Tool Sensor input for dynamic TLO setting.

Build 23 June 17, 2003
Bug Fixes
1.   EStop gets stuck in endless loop (Stack Overflow).
2.   A Axis Homes by Home Switch only (Can no longer use Lim terminal for A
axis).
3.   Fixed Cutter Comp in GCode interpreter.
4.   Gerber Reads up to 1,000 Apertures.
5.   Switching post processors will now have an imediate effect (no need to restart).
6.   Jogging with arrow keys no longer moves cursor.
Features Added
1.   Zero Feed Override.
2.   Added Machine Time Estimate.
3.   Added Option to base depth of Carvings on 100% Black or the Darkest available

color.