DeskCNC

Setup & Operating Manual

DeskCNC Manual

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 DeskNC and DeskCNC

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

Assuming you now have your controller hooked up to your computer 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. Common settings for Sherline machines can be found in the displayed windows.

If your machine is using anything other than 3 axes you will first need to open the Desk***.ini file which you will find in the same folder as the main program. Open this file in a text editor and scroll down to the **[Tradj]** label. The first line after the label has *"Axes = 3"* , change this value to either 2 or 4 to describe your machine and save and close the file. You can now start DeskNC (DeskCNC) to continue the configuration

To start the software configuration you will need to go to the Menu bar at the top of the screen and click on Settings/Machine Settings.

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 not required when no Controller is available.

In this window there are 4 Tabs, these are:

- *Machine Velocities Tab*
- *Axes Setup Tab*
- *DeskCNC Setup Tab*
- *Tool Sensor/Probe*

Before you can use DeskCNC you will need to configure it to work with your machine and computer. The first part of this procedure is to define your machine to DeskCNC, to do this you will first need to click on the **DeskCNC Setup** Tab

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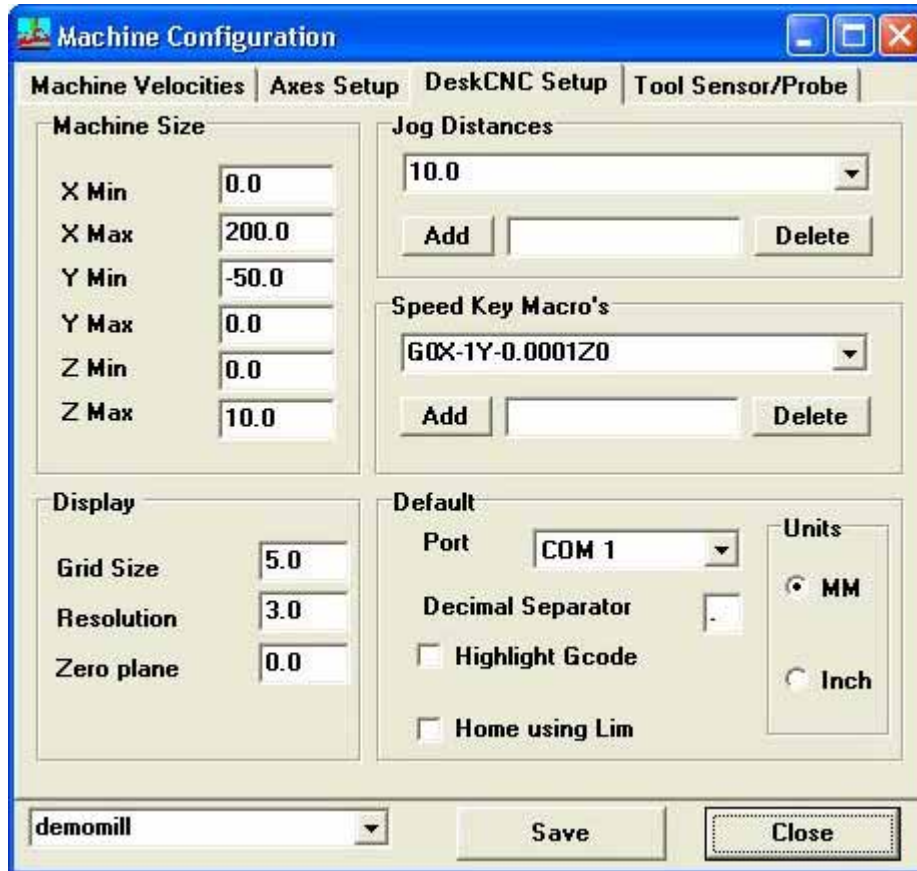
This page is used to define your machine workspace, the screen display, the connection to the controller and your base unit, metric or imperial.

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
Enter 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 axis setup and resolution, this can be done under the **Axis Setup** tab.

Machine 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 Co-ordinate's of the Axis is 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: A 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 [aprox. Resolution * (distance moved/measured distance)]

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.

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

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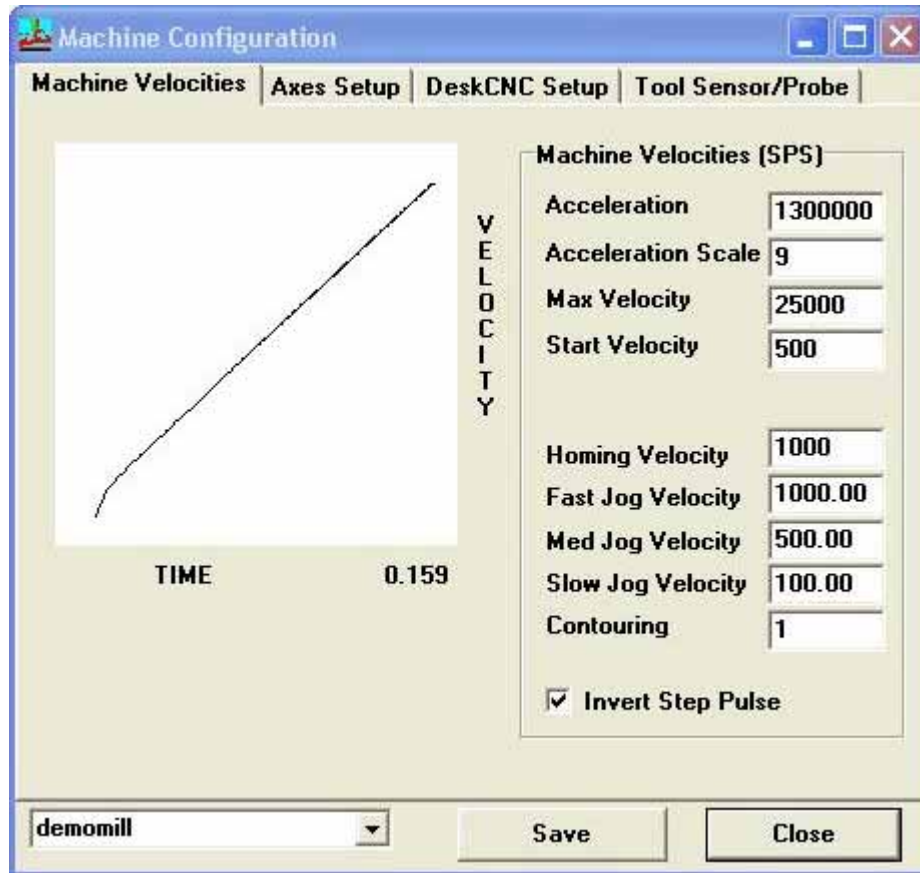
Setting up your Machine Velocities
DeskCNC utilizes a linear acceleration profile throughout its 40 - 40,000 SPS range. To help set the velocity and acceleration parameters, a graph of the acceleration profile is made available.

Parameters for viewing the acceleration profile

- **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 a 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
Set the velocity for all Homing functions in Steps per Second.

Fast, Medium, and Slow Jog Velocities
Set the velocities for Continuous Jogging. Values are in Units per Minute.
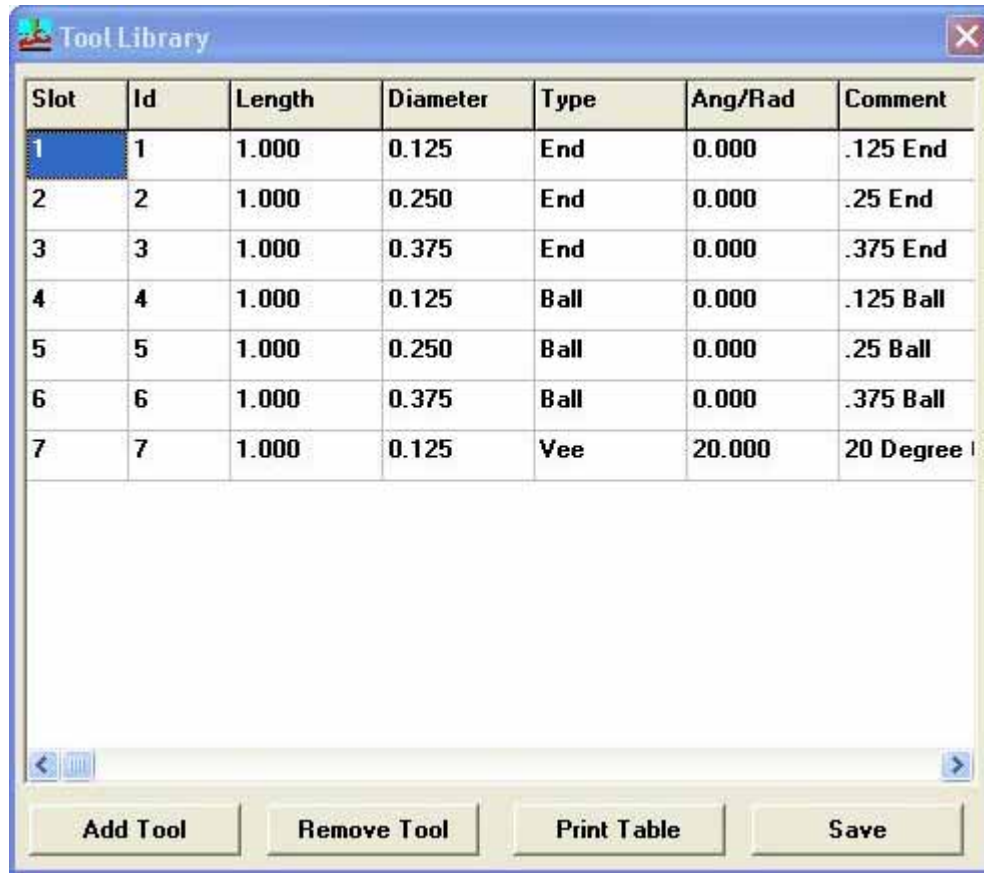
Contouring
Turns Continuous Motion Contouring (CMC) On/Off and sets the threshold. When Contouring = 0, CMC is off and all moves will decelerate to the Machine's Start Velocity. When Contouring = 1, CMC is set to Automatic. In Automatic CMC, each move will decelerate only the amount necessary to make 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

Slot	Id	Length	Diameter	Type	Ang/Rad	Comment
1	1	1.000	0.125	End	0.000	.125 End
2	2	1.000	0.250	End	0.000	.25 End
3	3	1.000	0.375	End	0.000	.375 End
4	4	1.000	0.125	Ball	0.000	.125 Ball
5	5	1.000	0.250	Ball	0.000	.25 Ball
6	6	1.000	0.375	Ball	0.000	.375 Ball
7	7	1.000	0.125	Vee	20.000	20 Degree

Add Tool   Remove Tool   Print Table   Save

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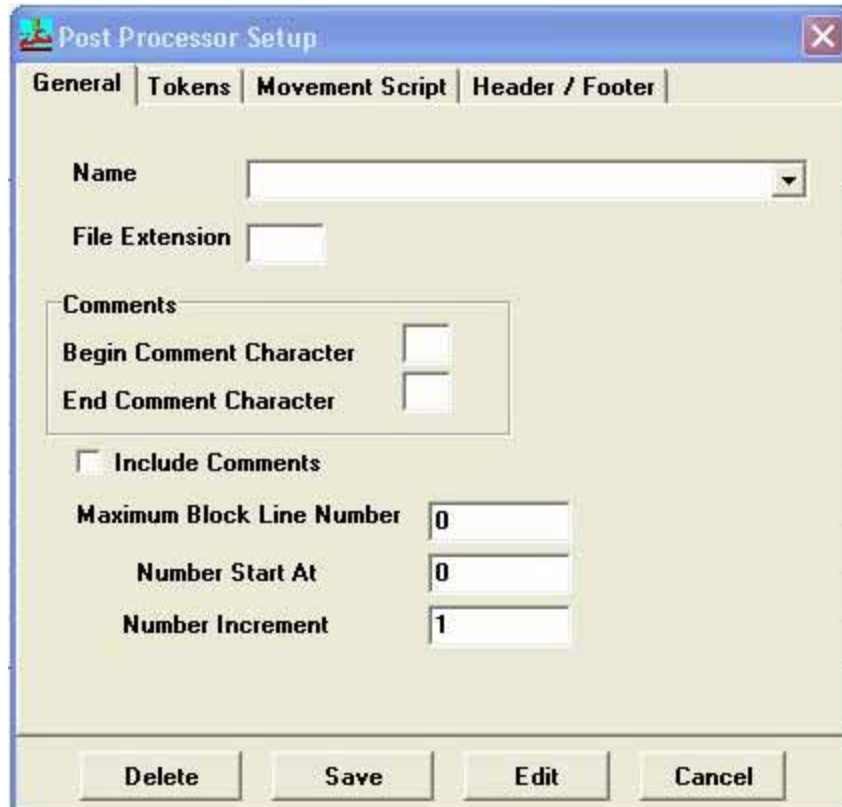
Add Tool
Click this button and fill in the tools parameters. For tool Type, use End, Ball, or Vee only. These must be spelled exactly.

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.
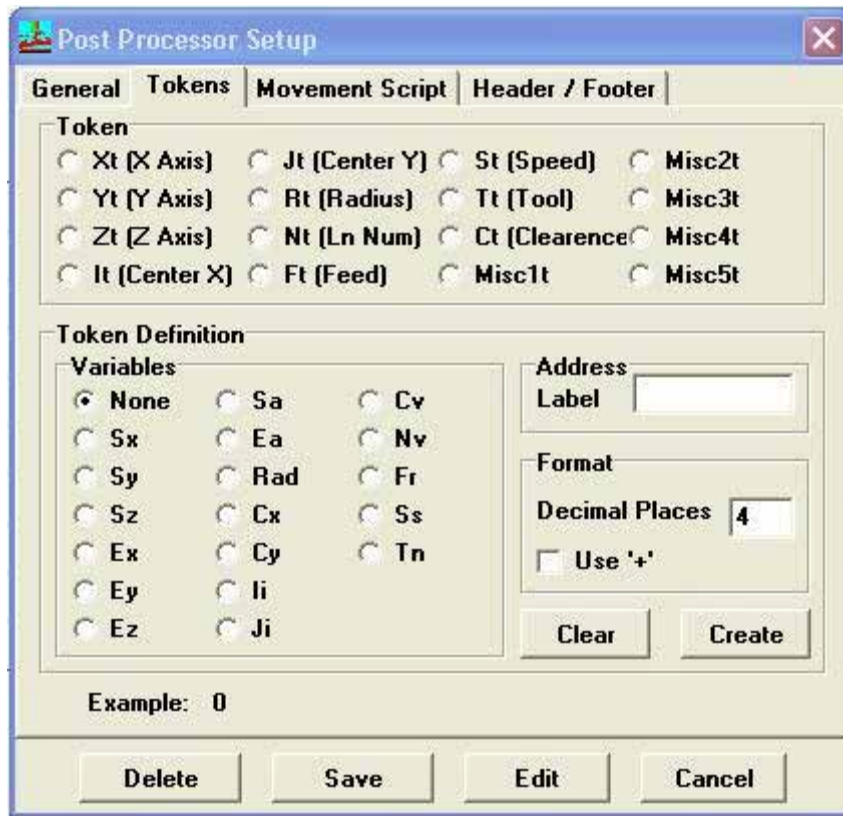
Tokens
Commands are broken down into 'Tokens'. A Token consists of an Address Label and a Value. The Token 'X2.5' begins with the Address of 'X' (in this case representing the X axis) and its value of 2.5. Tokens can begin with a space to make the output file more readable. All Tokens must be created 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 Feedrate
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 toolpath
Sy for the Start Point Y of a toolpath
Sz for the Start Point Z of a toolpath
Ex for the End Point X of a toolpath
Ey for the End Point Y of a toolpath
Ez for the End Point Z of a toolpath
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.
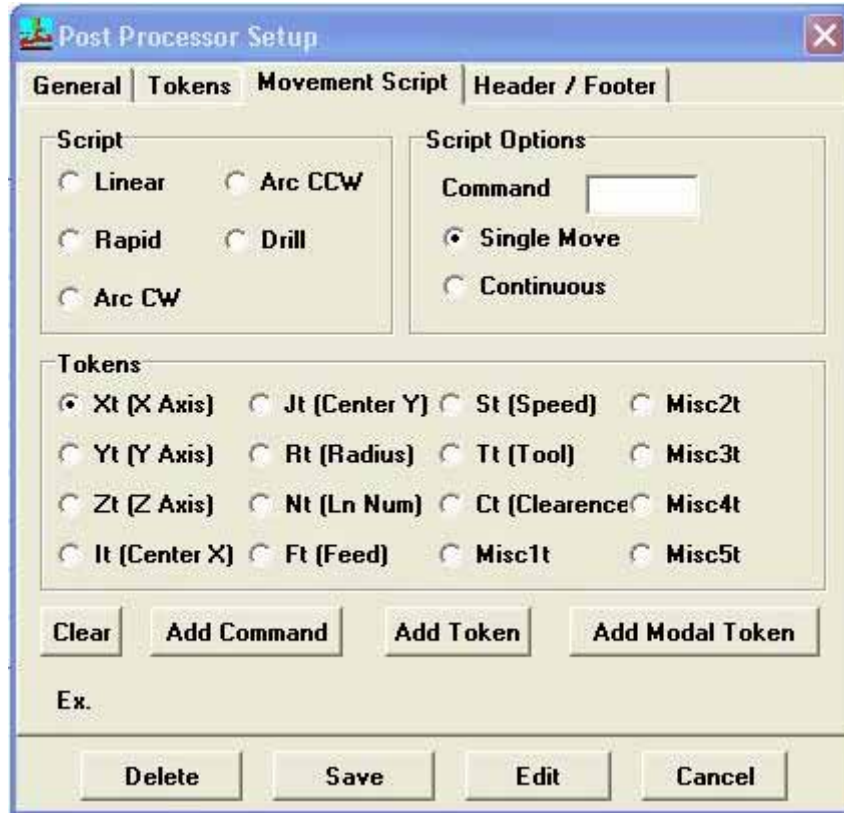
Movement Script

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 feedrate Token (Ft) is displayed in brackets to show that it will only be written if it differs from a previous value.
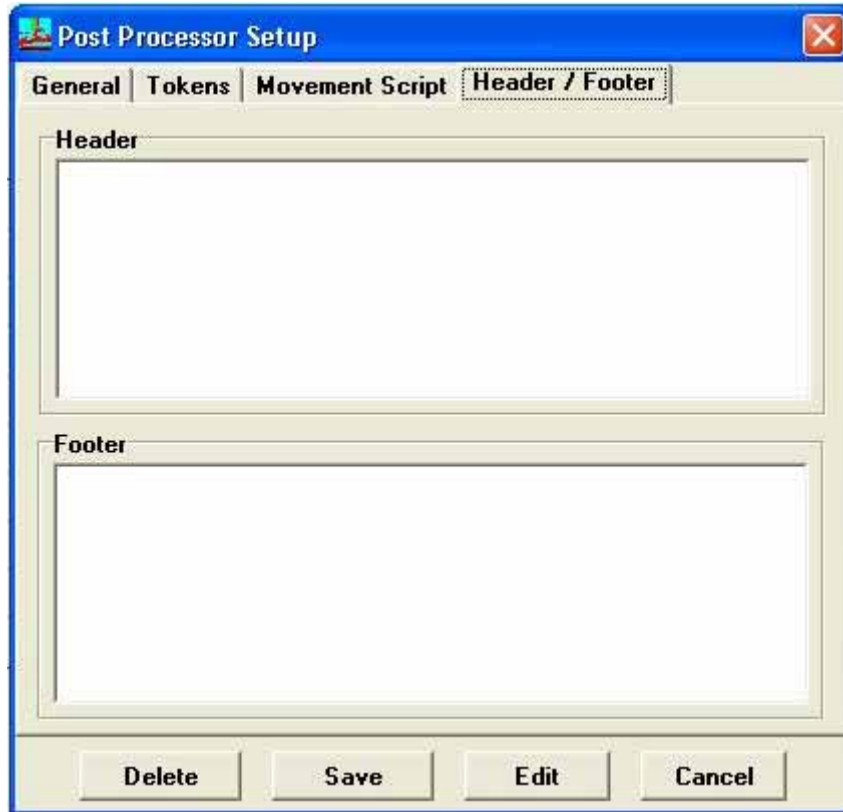
Header / Footer
Here you can add any miscellaneous start up or end commands that will be written to the output file. you can also add your own comments.

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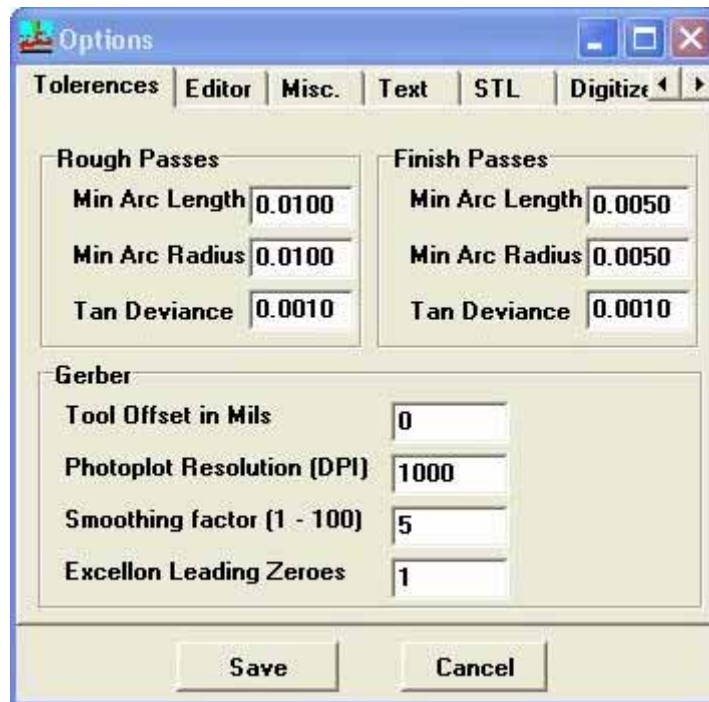
Min Arc Length
Enter the minimum length of an Arc segment. Arcs less than this length will be converted to line segments. The length of an Arc should be less than the length of a few 'steps' as defined by your CNC machine.

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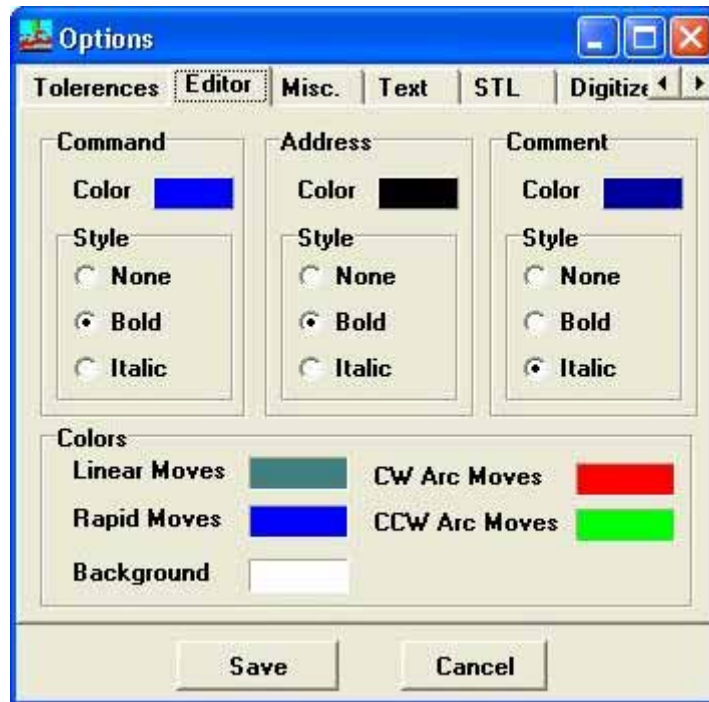
Color
Enter the color of the Command, Address, or Comment that is displayed in the Edit Box when Show Syntax is enabled. Show Syntax allows easier reading of the Machine Code file by 'Highlighting' key words based on the currently loaded Post Processor.

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. Toolpaths 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.

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

Show Boundaries when Redrawing Toolpaths
If View Toolpaths is selected in the View menu, you have the choice of viewing just the toolpath segments or the toolpath 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
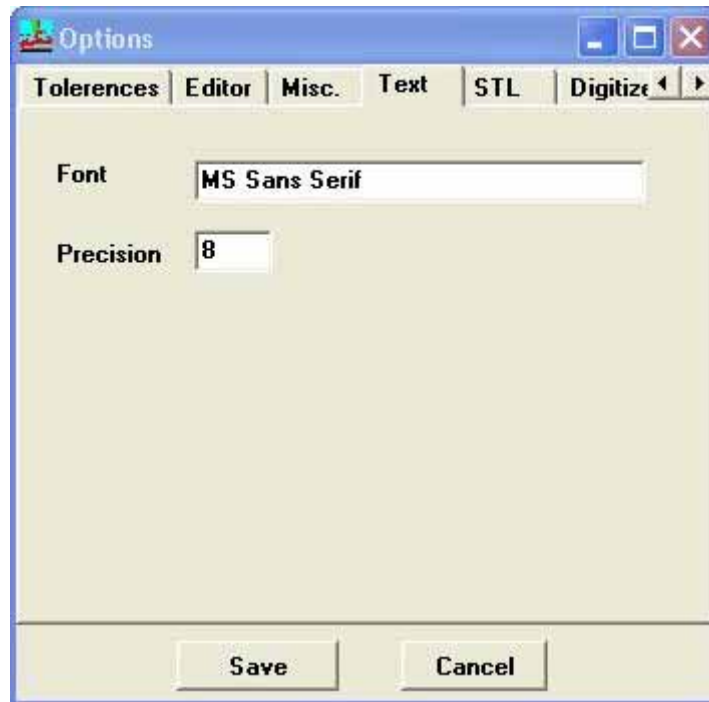
Font
Select a default True Type Font. All text references in the DXF file will be converted to vectors while loading.

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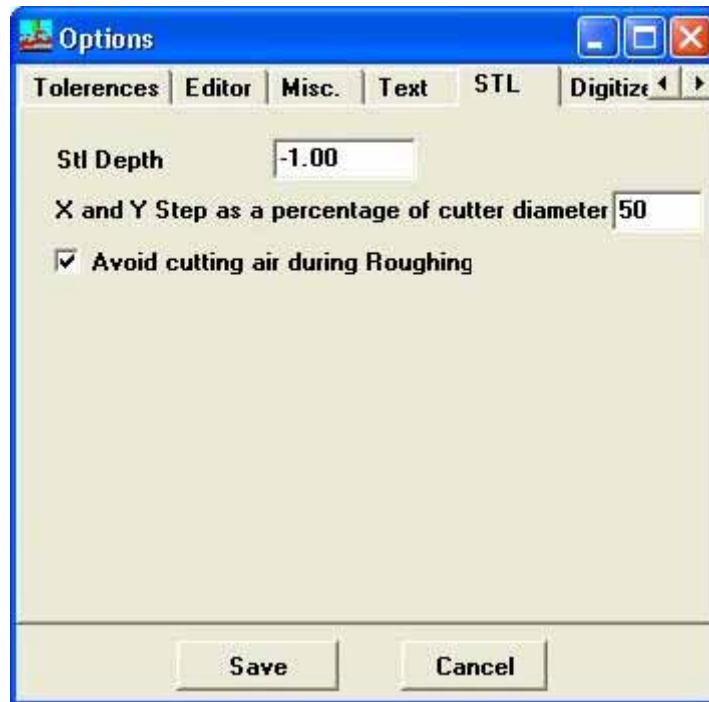
STL Depth
The depth at which to set the top of the STL geometry on load. All geometry will be translated in the Z direction so that the Maximum Z value of the geometry is located at this depth. The Blank's Z2 value will be set to this depth.

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
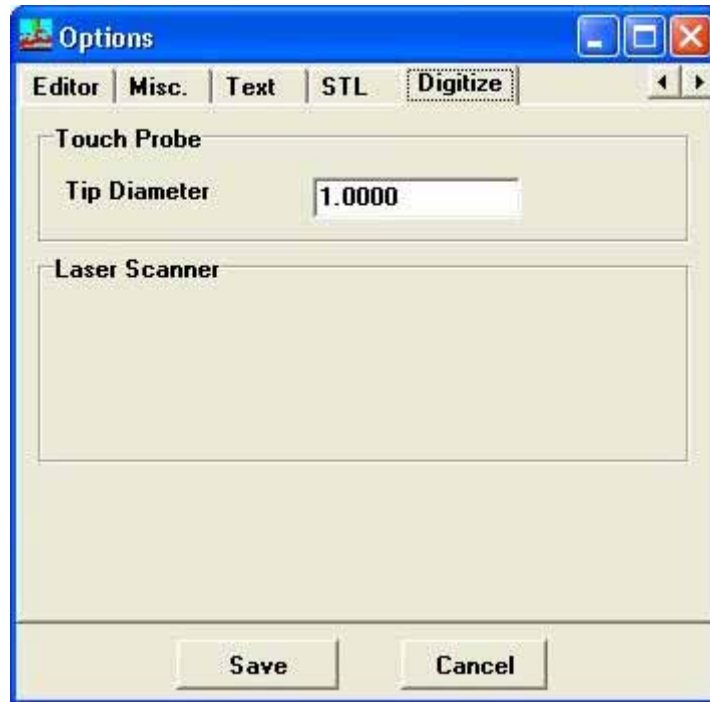
Tip Diameter

If you are using a digitizing probe (5 way switch) you can enter here 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 toolpath 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 toolpath 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 Toolpath 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 toolpath loop.

Operation

Start this command by pressing the Change First Entity button or 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 Toolpath 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 toolpath 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
Delta 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.

Creating Toolpaths

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 toolpath segments that trace the selected regions exactly.

Cutter Compensation (Left, Right)
Setting cutter compensation to Left will produce toolpath 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.

Common Tooling Parameters
<u>These definitions and guidelines are common to all toolpath functions. For help on an individual toolpath function see the help for Pocket, Contour, Drill, or Route.</u>

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 Toolpaths 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 Toolpaths 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

Entries and exits 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.

Definitions and guidelines

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.

Working with STL Files

These toolpath functions are used to create toolpaths from 3D STL files.

For Definitions and Guidelines common to all Toolpath functions go to Common tooling parameters.

Definitions and Guidelines

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
Toolpaths 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 toolpath. The Y Step size is the distance along the Y axis for each segment of the toolpath. 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 Toolpaths-
After selecting Toolpaths - 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 Toolpaths 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 Toolpaths - 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 Toolpath 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 feedrate for the Toolpath 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 Toolpaths
Clicking this button prompts you to save the Toolpaths 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

The following options are available

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 Toolpath 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.

Single:

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.

Topography:

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 Toolpaths 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 Toolpaths
Clicking this button prompts you to save the Toolpaths 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.
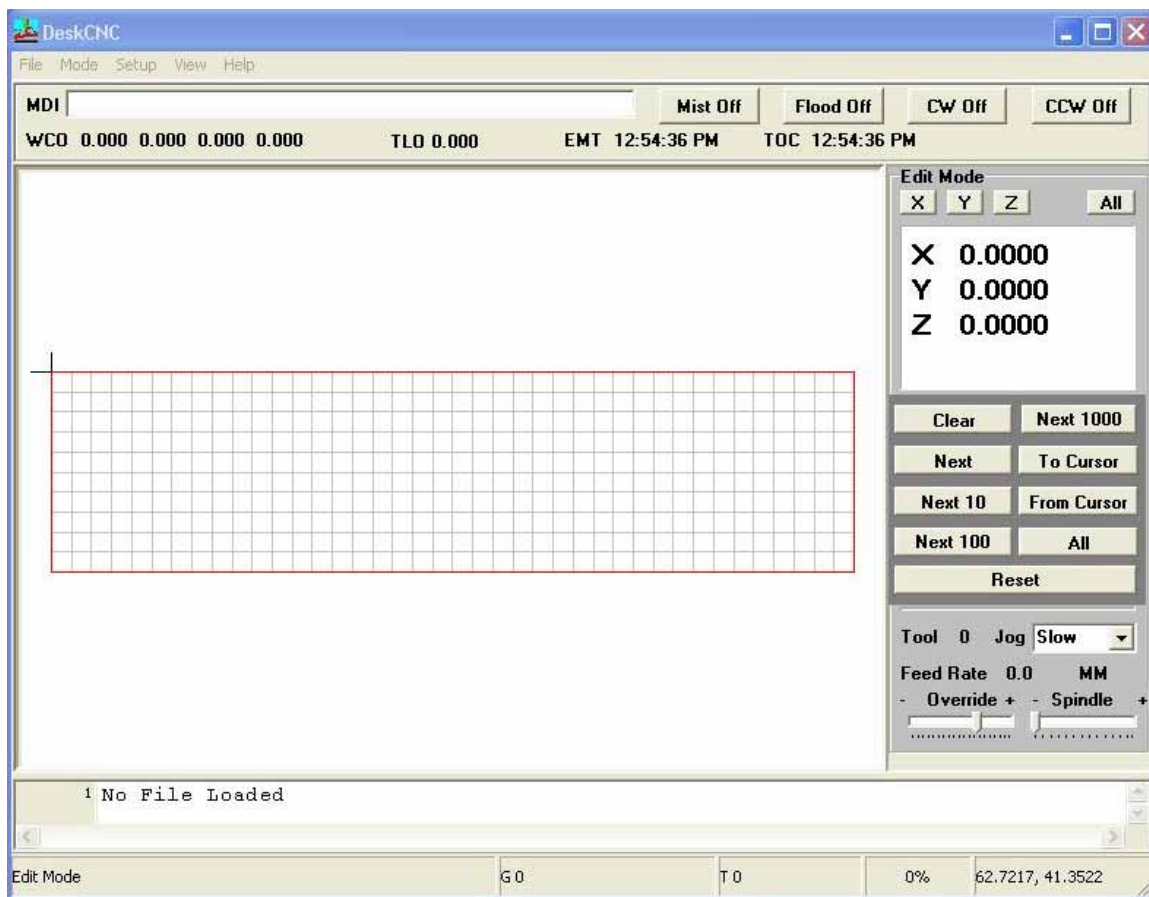
Mouse Movements

- **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.
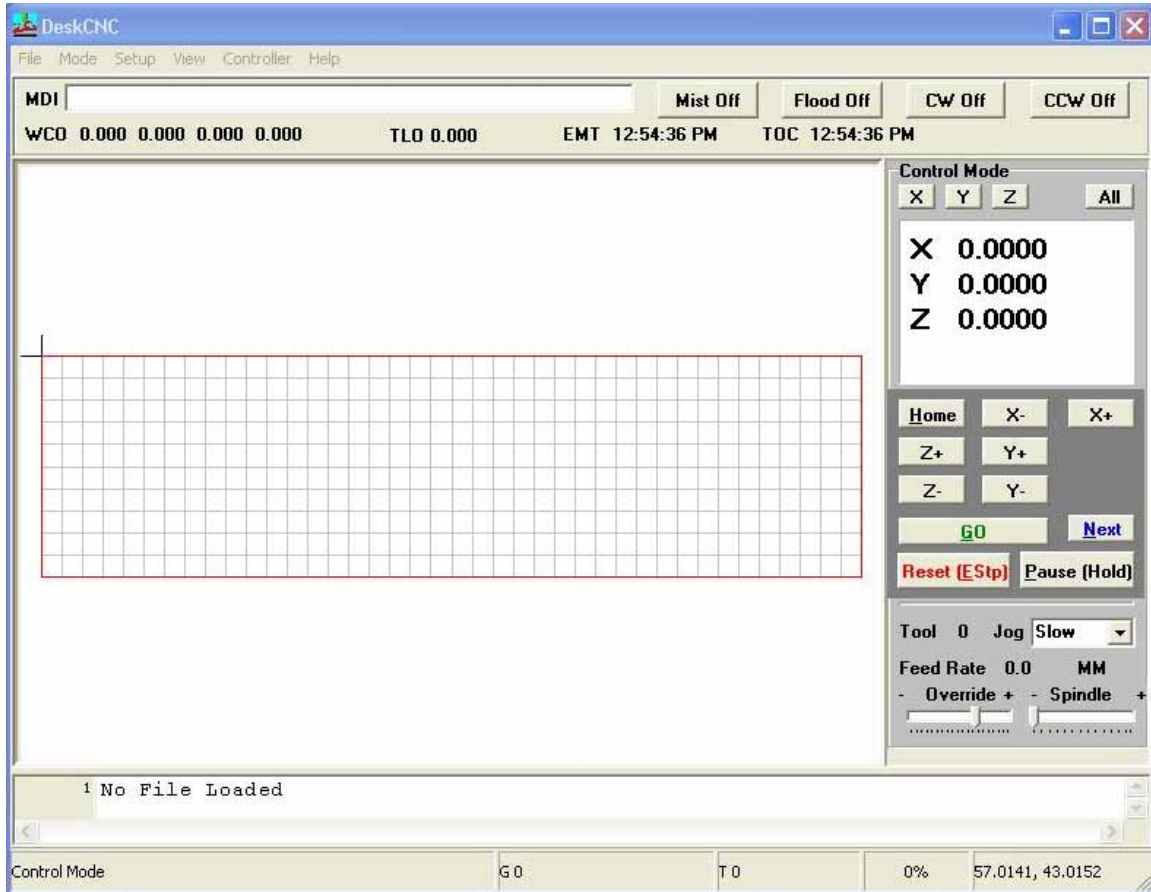
The MDI Command Box

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 of 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 axis. 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 axis 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 feedrate. The amount of increase in feedrate is not a percentage of the actual feedrate 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 feedrate in a similar way.
- **Spindle** - Sliding this control to the right of center will increase the feedrate. The amount of increase in feedrate is not a percentage of the actual feedrate 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 feedrate 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 Toolpath Display area.

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

DXF 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)
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.

Select Toolpaths - 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 toolpath segment of each loop. If the Mark All Entities option is selected then an arrow will be displayed at the midpoint of all

the toolpath segments.

Select Toolpaths - Run Machine.
Accept the default filename of PockTut.dnc and press Enter. DeskCNC will go into
Machining Mode and automatically load the PockTut.dnc toolpath 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 Toolpaths - 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 toolpath 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 toolpath segment of each loop. If the
Mark All Entities option is selected then an arrow will be displayed at the midpoint of all
the toolpath segments.

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

Select Toolpaths - Run Machine.
Accept the default filename of PockTut.dnc and press Enter.
DeskCNC will go into Machining Mode and automatically load the PockTut.dnc toolpath
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
Right click the mouse button and select "Select Empty Circles" (Select - Select Empty Circles). All the circles will turn to blue boundaries.

Select Toolpaths - 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 toolpath segments are created.

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

Select Toolpaths - Run Machine.
Accept the default filename of DrillTut.dnc and press Enter.
DeskCNC will go into Machining Mode and automatically load the DrillTut.dnc toolpath
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 Toolpaths - 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 toolpath segments are created.

Select Toolpaths - Run Machine.
 Accept the default filename of TeaPot.dnc and press Enter.
DeskCNC will go into Machining Mode and automatically load the TeaPot.dnc toolpath 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 Toolpaths - 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 Toolpaths - 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 Toolpaths - Run Machine.
Accept the default filename of Advanced.dnc and press Enter.
DeskCNC will go into Machining Mode and automatically load the Advanced.dnc
toolpath 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