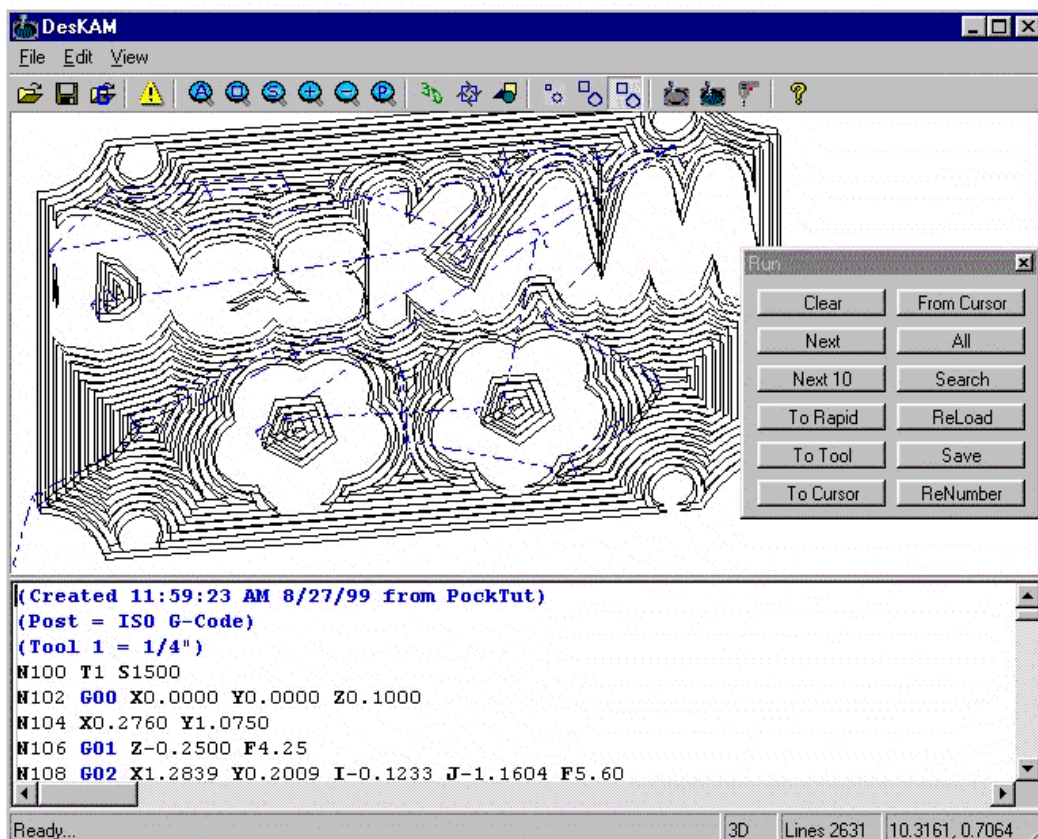


DesKAM

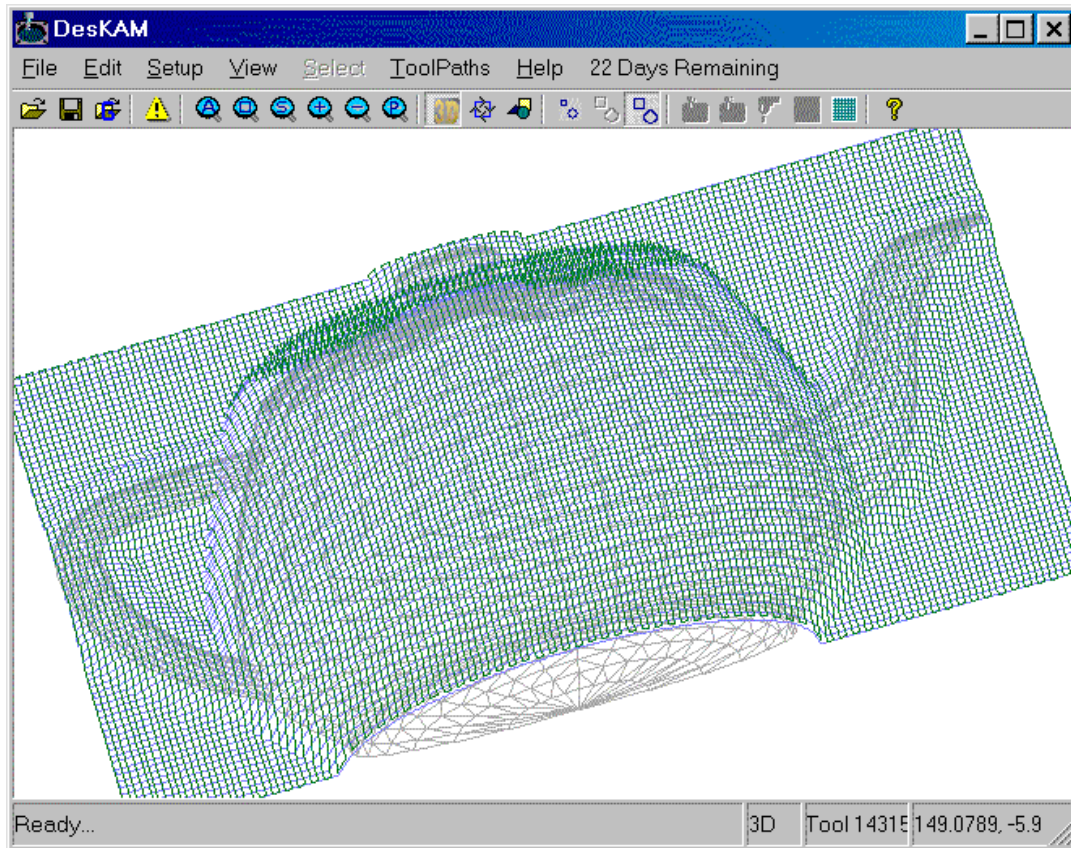
DesKAM 2000 is a Windows 95/98/NT based CAM program that inputs a 2-1/2D DXF or 3D STL file and creates a post processed output file.

DesKAM is one of the easiest CAM programs to learn and use ! It can be used for milling, routing, drilling or anywhere requiring 2 1/2 D toolpath creation. It performs spiral pockets, contouring, drilling, and prototyping functions.

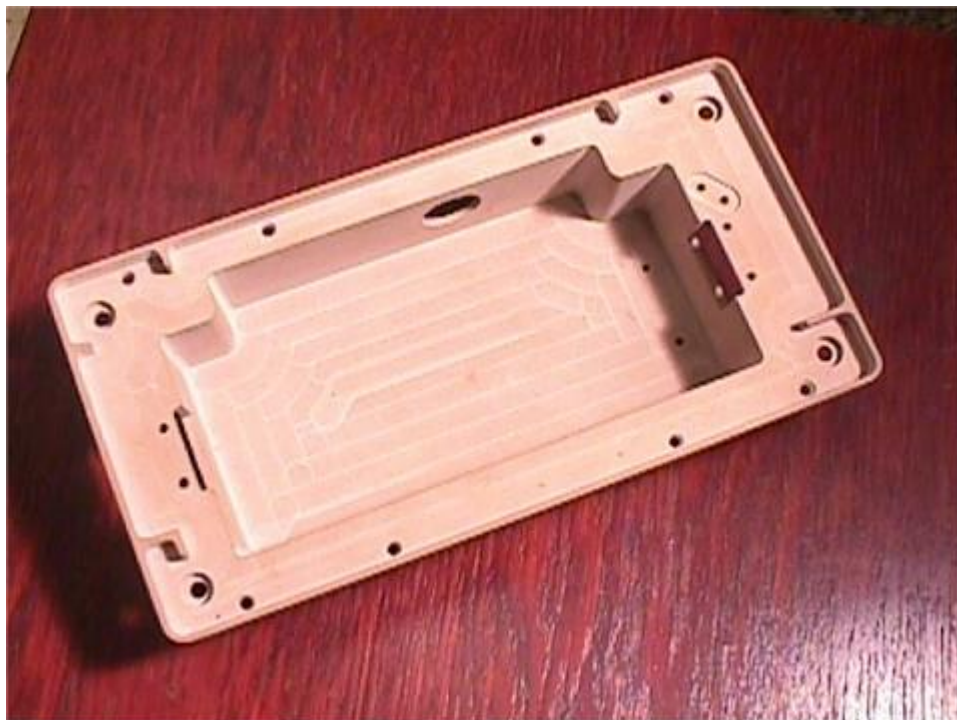
- Automatically detects islands and nested islands .
- Multiple Depths - Islands on top of islands inside boundaries...etc.
- Wall Profiles - Any region can have any wall profile.
- Automatically chains adjoining entities into regions to give an immediate visual representation of whether any regions are open or closed.
- Machining order if automatically determined, with manual override.
- Integrated 3D Editor with real time zoom, pan, and rotate.
- Syntax highlighted editor.
- User defined configurations.
- Programmable Post Processor.
- Automatically vectorizes DXF Text.
- Easy routing of PC Boards.
- Prototyping of STL solids.



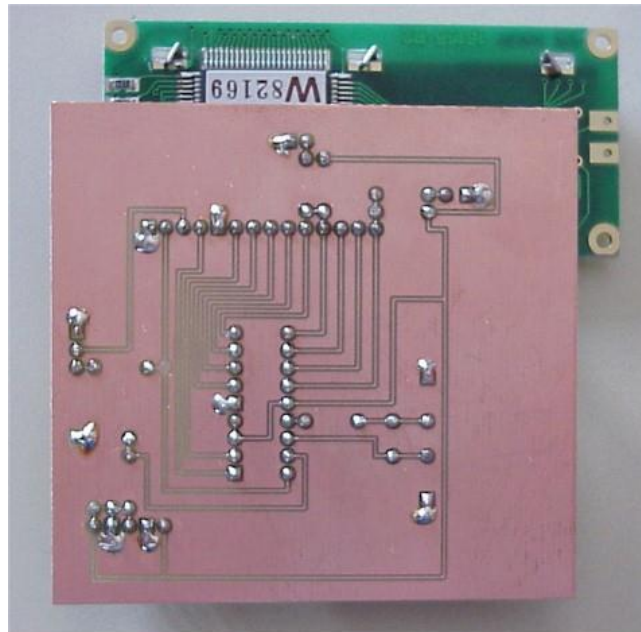
This is an actual screen capture showing the finished tool paths and the **G-code** ready to be loaded into **DeskNC**. With this program you can process 2D **DXF** files produced by your CAD software using various CAM functions such as Spiral Pocketing, Contouring, and PC Board Routing.



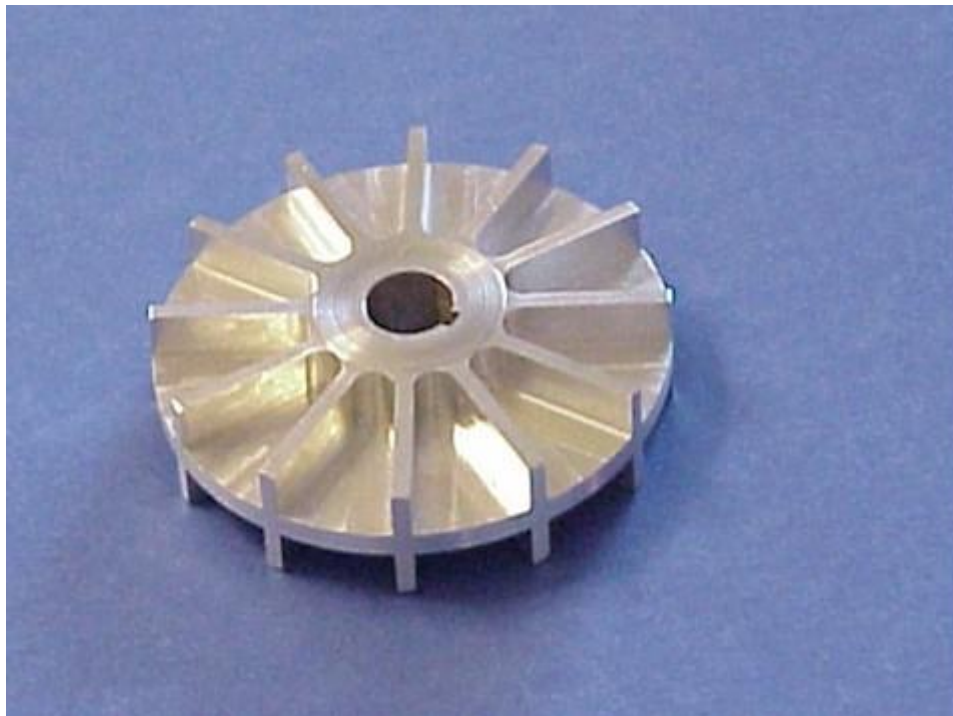
Here is a screen capture of a 3D teapot imported from an **STL** file. The toolpaths are seen around the STL geometry. You now have the ability to create parts of very high complexity... all from your computer!



This plastic mold has two degree sides and was created from **DesKAM 2000**



LCD display printed circuit board
Created by **DesKAM** - Routed by **DeskNCrt** in Closed Loop Mode



Impeller for a Model Engine Water Brake
Created with **DesKAM** - Machined with **DeskNC**

Summary

1. Glossary.....	6
2. File	7
2.1 Open DXF	7
2.2 Open NC File	7
2.3 3D editor	8
2.4 Add Text	8
2.5 Save Toolpaths.....	9
2.6 Save Toolpaths to DXF File	9
2.7 Close	9
2.8 Exit.....	9
3 Edit.....	9
3.1 Delete Selected Regions	9
3.2 Information	9
3.3 Reverse Dir of Region	10
3.4 Change First Entity	10
3.5 Set Machining Order.....	10
3.6 Explode/Join Selected.....	10
3.7 Combine Selected.....	11
4 Setup	11
4.1 Options	11
4.2 Post Processor Setup.....	13
5 View.....	15
5.1 Zoom Previous	15
5.2 Zoom In.....	15
5.3 Zoom Out.....	15
5.4 Zoom Window.....	15
5.5 Zoom All	15
5.6 Zoom Selected	15
5.7 Show Directions	15
5.8 3D	15
5.9 View All	15
5.10 View Selected.....	16

5.11 View Toolpaths.....	16
5.12 Show Tool Bar.....	16
6 Select	16
6.1 Select All.....	16
6.2 Select All Open.....	16
6.3 Select All Closed	16
6.4 Select All as Islands.....	17
6.5 Select Window.....	17
6.6 Select Region	17
6.7 Select Single.....	17
6.8 Select Empty Circles	17
6.9 Single Boundary.....	17
6.10 Single Island	17
6.11 Deselect All.....	18
6.12 Deselect All Open	18
6.13 Deselect All Closed	18
6.14 Deselect Window	18
6.15 Deselect Region.....	18
6.16 Deselect Single	18
6.17 Deselect Empty Circles.....	18
7 Toolpaths	19
7.1 Tooling parameters.....	19
7.2 Multiple Depths (All Toolpath Functions).....	19
7.3 Wall Profiles (Pocket and Contour Functions).....	20
7.4 Entry/Exits (Pocket and Contour functions)	20
7.5 Pocket.....	20
7.6 Contour	21
7.7 Drill	22
7.8 PC Board.....	22
7.9 Prototype	22
8 Registering	23
8.1 Days Remaining.....	23
8.2 Register	23

1. Glossary

DXF

Common CAD File Format. Most CAD Programs can output this format.
DesKAM only reads DXF Files (AutoCAD 10 to 14 format).

G-Code commands

G21 units = mm for coordinates and mm/min for feed rates
G90 Absolute coordinates

G00 Rapid motion
G01 Motion at the current feedrate
G02 Circular interpolation (Clockwise) in XY plane
G03 Circular interpolation (CounterClockwise) in XY plane
G04 Pause during a defined delay
G81 Drill cycle

X X axis position at the end of the move
Y Y axis position at the end of the move
Z Z axis position at the end of the move
I Arc center offset X (Signed)
J Arc center offset Y (Signed)

M00 Pause program
M02 End of program, stops spindle and motors
M03 Start spindle or laser
M04 Start spindle or laser with rpm or power proportional to feed rate during axis acceleration
M05 Stop spindle or laser
M06 Tool change
M07 Coolant on (Mist)
M08 Coolant on (Flood)
M09 Coolant Off

T Tool number
F Feedrate (mm/min)
S Spindle speed (0 to max rpm) or laser power (0 to 100)
N Line Number
P delay of pause in millisecond

GRBL

Implementation of G-CODE on DIY (Arduino, etc.) CNC machines

Island

A closed selected Region inside a selected Boundary.

Nested

DesKAM can determine if a region is within a boundary (an Island). If a region is then found inside this island it is defined as a nested boundary. If there is an island inside this nested boundary, it is an island to the nested boundary and not the outer boundary.

Region

An unselected string of connected entities. May be open or Closed.

Boundary

A closed selected Region that defines an outer boundary to a selection. It may stand alone or be inside an Island.

Contour

An Open set of connected entities.

2. File

2.1 Open DXF

The Open DXF Command is used to load a DXF file. Most CAD Programs allow you to export your work in the DXF file format. All part geometry is automatically converted to open and closed regions while loading and machining order is determined while loading.

To interrupt this command

Press Esc.

Tip

You can immediately determine if a region is closed or open by its color. If a region appears open and you think it should be closed you can either :

1. Press the Show Directions button with the Mark First Entity Only option selected under options.
2. Press the Information button.

Either of these will reveal the first entity in the region allowing you to decipher where the region is broken.

2.2 Open NC File

Viewing

When viewing toolpaths, the Zoom All, Zoom In, Zoom Out, and Zoom Window functions are available (See Zoom Comands). The display can be cleared by the Clear button. The toolpaths can be displayed in 3D by selecting the 3D option in the View menu. To rotate the toolpaths in real time, hold the left mouse button down and drag the toolpaths. To zoom in and out, hold the right mouse button and the Cntrl key down and move the mouse forward or backward. To Pan the toolpaths, hold the shift key and left mouse button down and drag the toolpaths.

Available command buttons :

All Button

Starts at the beginning and displays the entire file.

Next

Displays the next move. The line is highlighted in the edit box.

Next10

Displays the next 10 moves. The line is highlighted in the edit box.

To Rapid

Displays all moves up to and including the first G0 rapid move. Pressing this button again will display to the next G0 move. The line is highlighted in the edit box.

To Tool

Displays all moves up to and including the first Txx tool change code. Pressing this button again will display to the next Txx code. The line is highlighted in the edit box.

To Cursor

Displays all moves up to and including the line the cursor is currently on. The line is highlighted in the edit box.

From Cursor

Displays all moves from the line the cursor is currently on to the end of the file. The line is highlighted in the edit box.

Search

Selecting this button brings up a search dialog where you can search for and replace text.

ReLoad

After editing a file, you must reload the file for the edits to be seen on scree.

Save

Press this button to save the file after editing.

Renumber

Selecting this button brings up a ReNumber window. Line numbering follows the syntax of the 'Line Numbering Token' that is set in the current Post Processor. ReNumbering can only take place at the beginning of the current line. Enter the Start value and Increment. An option is available to add a space after the Line Number for easier reading.

Additional Right Click Options

By right clicking in the edit box two more options are available.

Font

Select this option to change the Font in the Edit Box.

Show Syntax

Select this option to highlight the Machine Commands in the color and style set under Options - editor.

2.3 3D editor**2.4 Add Text**

Opens a Dialog window for exploding True Type Font vectors.

Definitions and Guidelines

By Point/Enter Below

Select between entering the Origin directly or placing the origin by clicking in the workspace. When adding text around an Arc, this will be the center point.

Linear/Arc

Select between aligning the text linearly along the x axis or around an Arc.

Left/Cent/Right

Text can be justified to the left, center, or to the right of the origin. When adding text around an Arc, text can be justified to the left, center, or to the right of the Start Angle.

Center X/Y of Selected

Text can be justified to the center of any selected regions. If there aren't any regions selected, this feature will be unavailable.

Angle/Radius

Used for aligning text around a circle. Angle is the Start Angle of the text. 0 degrees is at the top and continues Clockwise. Radius is the distance from the origin to the text.

Origin X/Y

Origin used to align the text. When adding text around a circle, this will be the center point.

Font

True Type Font used to add the text. Double clicking this box will bring up a font dialog window which allows the selection of any True Type Font.

Height

Size of the added text. This is the maximum height of each character.

Text

Actual text to add.

2.5 Save Toolpaths

If the toolpaths were not saved during a toolpath command, you may save them at any time by selecting this command. If the toolpaths are not saved you will be prompted to do so when you exit DesKAM.

2.6 Save Toolpaths to DXF File

This command allows saving of the Toolpaths in the DXF file format.

2.7 Close

This command closes either the loaded DXF file or NC file and releases system resources.

2.8 Exit

This command ends the current session of DesKAM 2000. You will be prompted for confirmation if the toolpaths or DXF file have not been saved.

3 Edit

3.1 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 DesKAM 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.

3.2 Information

This command allows you to view the ID of each region. It has a slightly different definition for each view.

When the View is set to All and there is nothing selected

When the view is set to all (View All in the View Menu) and no regions are selected, a label will be displayed at the beginning of the first entity in each region. The number in this box is the region number. This command in this view can be useful in determining where a Region starts. It also defines the Machining Order when toolpaths are created.

When the View is set to All or Selected and there is a selection

A label will be displayed at the beginning of the first entity in each region. The first number in the box will be the region number DesKAM associates with each region. The (Optional) second number will be displayed if this region is an Island. The second number will be the number of the boundary that this region is an Island to. This command is useful after your selections are made to verify that indeed an Island is an Island to its boundary. Sometimes an Island can become an Island to a more distant boundary if you use many different selection commands (Select All, Open, Close, Window, Region, and Single) for one file. A third number may be present in parenthesis. This is the Wall Profile number

and the DeltaX associated with this wall.

When the View is set to Toolpath

A box is displayed at the beginning of the first entity of each Toolpath region. This number corresponds to the order at which the toolpaths will be processed. Useful to see the order of the toolpath segments.

Tip

- When selecting regions, it is sometimes better to Select All regions (or a larger area than required) and then Deselect what you don't need.
- If a Region is Open and you think it should be closed, using this command will mark the point where the Region is separated.

3.3 Reverse Dir of Region

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.

3.4 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.

3.5 Set Machining Order

Machining Order

With this function the order the Regions are machined can be controlled. DesKAM 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.

3.6 Explode/Join Selected

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. DesKAM 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.

3.7 Combine Selected

Combines two or more intersecting regions into 1 region. Used mainly for routing PC Boards.

This feature is used mainly for combining 'traces' and 'pads' found in PC Boards. It can be used to reduce the number of Islands and speed up toolpath calculations when routing PC Boards. Select the regions and then select this command. The Status Bar will display the number of new regions and the number of old regions.

4 Setup

4.1 Options

Here you can change the Options in DesKAM and set viewing preferences.

Tolerances

Min Arc Length

Enter the minimum length of an Arc segment. Arcs less than this length will be converted to line segments.

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

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

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

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.

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.

Try to correct bad DXF geometry

This feature will try to correct 'overlapping' regions usually caused by Raster to Vector converters such as Corel Draw.

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).

Post Processor

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

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.

Text

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 TrueType Font glyph will be converted to a series of line segments. The number of line segments per curve is set in the 'Percision' parameter.

STL

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 diamter in the machining parameters.

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 toolpath.

4.2 Post Processor Setup

DeskNC 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 'DeskNC' is a standard G-Code Post Processor that the program DeskNC 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

DesKAM 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

Misc 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 Non-Modal Token (default) will be written to the output file even it is identical to previous value.
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 (N---)
- Press the Add Command button (G---)
- Select the 'Xt' Token and press the Add button (X---)
- Select the 'Yt' Token and press the Add button (Y---)
- Select the 'Zt' Token and press the Add button (Z---)
- Select 'Ft' and press the Add Modal button (<F--->)

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 (modal command).

```
Ex. N0026 G01 X15.3457 Y16.3457 Z17.3457< F27.35>
```

Result in POST.CFG file :

Scripts0=F* G01*6*F*-1*F*0*F*1*F*2*F*7*T

- * : text separator
- F* : start of script
- 6*F : non modal command (line number)
- -1*F : put previous command (6*F here) at the beginning of the gcode line
- 0*F, 1*F, 2*F : non-modal command (X, Y, Z coordinates)
- 7*T : modal command with token7 (feed rate)

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.

5 View

5.1 Zoom Previous

Displays the previously displayed view before a zoom command was executed.

5.2 Zoom In

Zooms in the amount set in the Zoom In/Out Percent Option.

5.3 Zoom Out

Zooms out the amount set in the Zoom In/Out Percent Option.

5.4 Zoom Window

Hold the left mouse button down and drag a box around what you wish to magnify.

5.5 Zoom All

Fits the entire DXF File in the main window.

5.6 Zoom Selected

Fits only those regions that are selected to the main window.

5.7 Show Directions



If this item is checked, a small arrow will be displayed at the midpoint of an entity. If the Mark First Entity Only option is selected, only the first entity in a region will be marked. Likewise if the Mark All Entities option is selected, all entities will have an arrow displayed at their midpoint.

Tip

With Mark First Entity Only, the first segment of a toolpath is easily distinguished.

5.8 3D

Toggles 3D and 2D viewing of geometry and toolpaths. When in 3D mode, geometry can be dynamically zoomed and rotated in 3D space.

Redraw

Redraws the screen.

5.9 View All

Selecting View All will display all regions in the loaded DXF File. The second panel of the status bar will display All and the total number of segments.

5.10 View Selected

Selecting View Selected will display only those regions that are currently selected. The second panel of the status bar will display Selected and the total number of selected segments.

5.11 View Toolpaths

Selecting View Toolpaths will display only the toolpath segments that were created from a previous Toolpath command. The second panel of the status bar will display Toolpath and the total number of toolpath segments.

Tip

Use the Show Directions option, and the Information command to get a good visual representation of the loaded file.

5.12 Show Tool Bar

This menu item will show the toolbar at the top of the main DesKAM window when it is checked. It will likewise remove the toolbar when unselected. Removing the Tool Bar increases the viewing area.

6 Select

6.1 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.

6.2 Select All Open

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

6.3 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.

6.4 Select All as Islands

Selects the outermost region as a boundary and all other regions as islands to that boundary.

This feature is used mainly for the routing of PC boards. When creating PC board geometry, create a box surrounding all traces and pads. When 'Select All as Islands' is invoked, all islands and pads will be selected for use in the PC Board Routing function.

6.5 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.

6.6 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.

6.7 Select Single

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

6.8 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.

6.9 Single 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 DesKAM's automatic selection tools fails to select a region as a boundary and you **know** it should be!

6.10 Single 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 DesKAM'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.

6.11 Deselect All

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.

6.12 Deselect All Open

This command will deselect all selected contours.

6.13 Deselect All Closed

This command will deselect all selected closed boundaries and islands.

6.14 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.

6.15 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.

6.16 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.

6.17 Deselect Empty Circles

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

7 Toolpaths

7.1 Tooling parameters

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.

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 occurs. Enter a value the amount *above* the work piece (Z=0) to allow clearance for the tool when it is travelling 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.

Line Number starting value

The number corresponding to the first line in the Machine Code File that will be created. Default is 100.

Line Number increment

The amount to increment the line numbers by. Default is 2. Line numbering can be added and removed in the editor.

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.

7.2 Multiple Depths (All Toolpath Functions)

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 than 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, DesKAM will first remove material around an island until it reaches a depth greater than the depth of the island. DesKAM will then remove material from the top of the island according to that islands depth.

7.3 Wall Profiles (Pocket and Contour Functions)

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.

7.4 Entry/Exits (Pocket and Contour functions)

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.

7.5 Pocket



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

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. $X-Y \text{ Step Size} = \text{Tool Overlap} \times \text{Diameter}$. 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 rate 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.

7.6 Contour

For Definitions and Guidelines common to all Toolpath functions go to [Common tooling parameters](#).

Definitions and Guidelines**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. DesKAM 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.

7.7 Drill



For Definitions and Guidelines common to all Toolpath functions go to [Common tooling parameters](#).

7.8 PC Board

For Definitions and Guidelines common to all Toolpath functions go to [Common tooling parameters](#).

The PC Board Routing function differs from the [Contour](#) function in the way that [Islands](#) are avoided. The PC Board function avoids all Islands as a selection where as the Contour function will Contour each region separately. The PC Board will only route closed Islands. It is therefore recommended to encase all traces and pads with a closed box and use the Select All as Islands command.

Definitions and Guidelines

Routing Depth

The depth to route the Islands. Enter a positive value. Only Islands are routed.

Feedrate

The rate to perform the routing operation.

7.9 Prototype

This toolpath function is 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 be 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.

8 Registering

8.1 Days Remaining

This menu item will display the Days remaining under the trial period. DesKAM will run for 30 days before the Toolpath functions will cease to work. After 30 days you must Purchase and Register DesKAM to continue using the Toolpath functions. The demo is limited to 1,000 lines of machine code that can be saved at a time.

8.2 Register

Simply enter your user name and registration code in the appropriate fields. You may need to restart DesKAM for the registration to take effect.