

Star CAM CNC Control for Windows

USER GUIDE

Larken Automation Inc.

© Sept 2005

REV 1.3 (Apr 2007)

Table of Contents

Welcome to Star CAM
A Note From the Larken
Contact Larken
Copyright Notice

Star CAM Installation

- System Requirement
- Beginning the installation
- Shut down other applications
- License agreement
- Choose destination location
- Set program shortcuts
- Confirm Setup Settings

Notes on Using Star CAM

- Keyboard Control
- Selecting Objects
- Gravity Snap
- Typing in Coordinates
- Layers
- Screen Coordinates
- Routing Process Overview

Star CAM Menu Commands

File Menu

Edit Menu

Contour Menu

Draw Menu

Tool Menu

Route Menu

Using the Layer Screen

- Two types of depths within objects
- Changing layers and depths
- Tool number
- XY Feed and Z Feed
- Rotation Angle
- Change button
- Clear Layer button
- Swapping layers

StarCNC Spooler

- Initializing the controller
- Configuring the machine settings

Controller overview
Calibration factors

Importing Design Files From Popular Programs

AutoCAD
Corel Draw
Design Cad 2D for Dos
Design Cad 2D for Windows
Design Cad 3D
Rhino 3D
Surfer 3D
CasMate Pro
EnRoute
Sign Lab

Selecting Cutting Tools

Wood router bits
Milling cutters
Carbide vs., HSS
RPM of spindle
Feed rate
Machining metals
Routing and milling chart
Wood router bit suppliers
Milling cutters and coolant suppliers

Digitizer Probe

Connecting the probe
Using the probe menu
Procedure to probe a part
Saving the probe file

Troubleshooting and Help

Helpful tips and tricks
Troubleshooting questions and answers

A Note From Larken.

Welcome to *Larken automation's* Star CAM CNC control system. The system consists of a front-end layout and design program called Star CAM and a CNC controller called StarCNC.

Use Star CAM to import your designs, layout and edit, and do simple drafting. Tools are assigned to cut the objects and then the tool path is saved for the StarCNC control popup to spool to the controller box.

This easy to use system allows you to be designing and laying out your next job in Star CAM while the last one is cutting. Full multitasking control the way Windows was designed to be.

Please note that the StarCAM program is still being developed and growing ,so please check our website www.LarkenCNC.com for the latest free updated version.

Larry Kenny
President, Larken Automation Inc.

Contact Larken Automations

By Mail:

Larken Automation
13-5350 Canotek
Ottawa, ON K1J 9E1

By Phone/Fax:

Toll-Free: (800) 591-0049
Ph: (613) 746-5685
Fx: (613) 746-6451

By E-Mail:

Larken@storm.ca

Website:

www.LarkenCNC.com

Copyright Notice

No part of this publication may be reproduced mechanically or in any form without the prior written permission of Larken Automation Inc. The software described in this manual is furnished under license and may only be used or copied in accordance with the terms of such license. The information in this manual is for informational use only, is subject to change without notice and should not be construed as a commitment by Larken. Larken assumes no responsibility or liability for any errors or inaccuracies that may appear in this document. StarCAM and StarCNC are registered trademarks of Larken Automation Inc.

© October 2003, [Larken Automation Inc.](http://www.LarkenCNC.com)

Computer Requirements

Pentium 266+ recommended
Windows 95 ,Windows 98 , ME., XP ,2000,Vista
32 Meg ram minimum recommended
SVGA Color monitor (1024 x 768 min)
Parallel (centronics) port (2 ports if used with a printer)

Note: Starcam updates are available at www.LarkenCNC.com under the tech-support page

Installation

Download the latest Starcam and StarCNC programs from the LarkenCNC.com website. They are in Zip files so you will need WinZip on your computer to Unzip them. Create directories (folders) on your C: drive called StarCam and StarCNC.

EG: C:\StarCam
and C:\StarCNC

Copy the appropriate Zip file to each directory and unzip them, putting the files in that directory.

Creating the Icon

Right-Click the mouse on the blank desktop, select from the menu NEW > SHORTCUT. Use the Browse button to find the StarCam1.exe file in the C:\Starcam directory and select it and click OPEN. Click. Next, Next and Finish. You should now have the StarCam Icon on your desktop.

Do the same for the Lspooler.exe file in the C:\StarCNC directory.

Note: Do not drag the file to the desktop to create an Icon, since this doesn't properly set up the working directory for the program. You can do this, but you will need to edit the Icon's properties and set the working directory to the correct directory location .



Configuring the Spooler

Run the StarCNC program (Lspooler.exe) and goto Setup>Environment_Setup. You will need to set the location of the 'StarCam Spooler Files' using the Browse button, to the directory where StarCam is located (normally C:\StarCam) and then click the save button.

This tells StarCNC where to look for the 'Temp0.LKS' file (temporary spooler file) that Starcam creates each time that you select and output a job.

Keyboard Control

- Star Cam is designed to use the mouse or keyboard to control cursor movement. To use the cursor arrow keys, hold down the **Alt** key at the same time. The cursor will move in the *cursor-increment* which is displayed at the bottom right of the screen. To change the *cursor-increment*, press **Alt-PgDn** or **Alt-PgUp**
- You can also enter the cursor increment using Ctrl-PgDn. Very handy for drawing lines, drilling holes in a pattern and pasting objects in a pattern.
- If objects are selected, you can move them with the cursor keys by adding the **Shift** key to the Alt-arrow keys.

Selecting Objects

Objects are *selected* by clicking on them or windowing them with the mouse. To *unselect* objects, hold down Shift while clicking or windowing. To unselect all, double click on a blank part of the screen or press ESC.

After objects are selected, they can be dragged with the mouse by clicking and holding the mouse on the center select-node. Double-Clicking on a blank part of the screen unselects all objects.

Gravity Snap(*)

- The * asterisk key is a gravity snap key. It will snap the cursor to the nearest line segment end or node. It is useful when drawing or selecting objects.

Repeat Drawing command (F3)

Repeats the Last drawing or Create tool-path object command.

Measure Function (F5)

Press F5 and you can measure distance and angle.

Object Properties (F6)

The properties popup displays information about a single object you have selected. Properties doesn't display info when more than 1 object is selected.

Typing in coordinates

Typing in Absolute, relative, or Polar coordinates can move the cursor for very accurate cursor placement. While using these functions, don't move the mouse. Use the Tab key to move between data items to enter, then tab to OK and press enter.

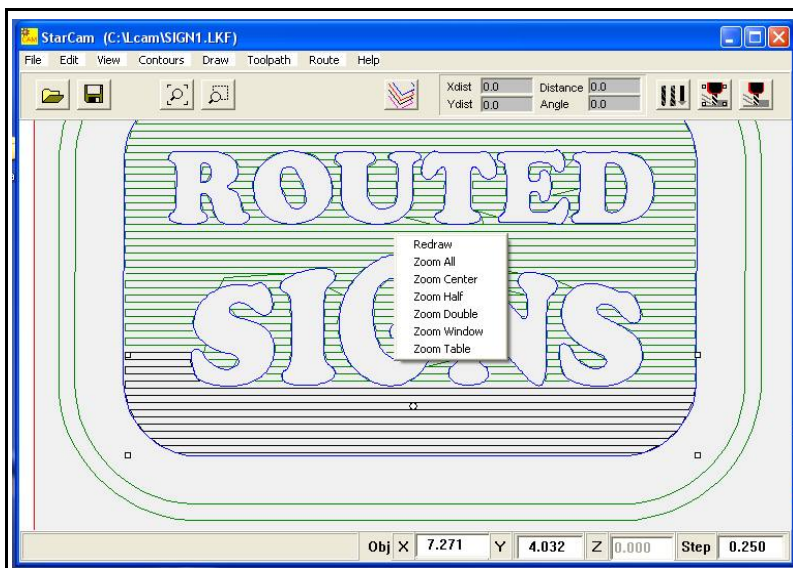
Press ; (Semicolon) to enter absolute coordinates

Press ' (Apostrophe) to enter relative coordinates

Press [(Left Square bracket) to enter polar coordinates

Zoom Submenu

Right clicking the mouse on the screen displays the **Zoom submenu**. The 'Z' Zoom command is almost always active, even while running other commands. This is very useful for moving objects positioned them to .001" accuracy where you can zoom in and positioned them to .001" accuracy.



Zoom Hotkeys

F7 Zoom All F8 Zoom half (centered on cursor) F9 Zoom double (centered on cursor)

Screen Coordinates

- The screen coordinates of Star cam are exactly the same as on the xy table. The Origin (0,0)* point on the screen corresponds to the 0,0 point on the table. The origin can be set anywhere in the drawing as the controller can run in full Plus and minus coordinates. On the Z axis negative is down and positive is up.

Routing Process Overview

The process flow of running a job on the router is to:

Create your Design

1 - Create the design on your design program Corel-Draw, AutoCAD, Sign Lab, Master Cam, Surf-Cam etc. and export the data to a HPGL, EPS, DXF or G-Code file. The file must be a vector file, in an HPGL, DXF or G-code format. (Not a bitmap file such as BMP or JPEG etc.)

Import to Star Cam

2 - Run 'Star Cam' and import your design using the File/ import commands. There are Import commands for standard HPGL, DXF and GCODE files.

Use the Layer menu to set depths and routing sequence

3 - You can organize the drawing into layers for better control of the order of cutting. Layers cut in order from 1 to 50. Also you can set the depth and other parameters for each layer separately. **Note:** The depth setting only applies to non 3d objects that are designated 'depth by layer'.

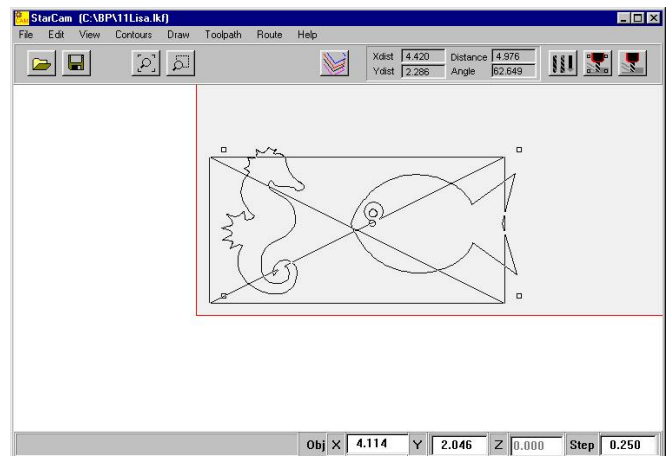
You can also turn off layers and protect layers from editing using the Status setting.



Surf Name	Stat	Depth	Tool#	XY Feed	Z F
LAYER 1	Live	-0.250	7	0	0
LAYER 2	Hot	-0.200	1	0	0
LAYER 3	Hot	-0.200	1	0	0
LAYER 4	Hot	-0.200	2	0	0
LAYER 5	Hot	-0.200	3	0	0
LAYER 6	Hot	-0.200	1	0	0
LAYER 7	Hot	-0.200	5	0	14
LAYER 8	Hot	-0.200	1	0	0

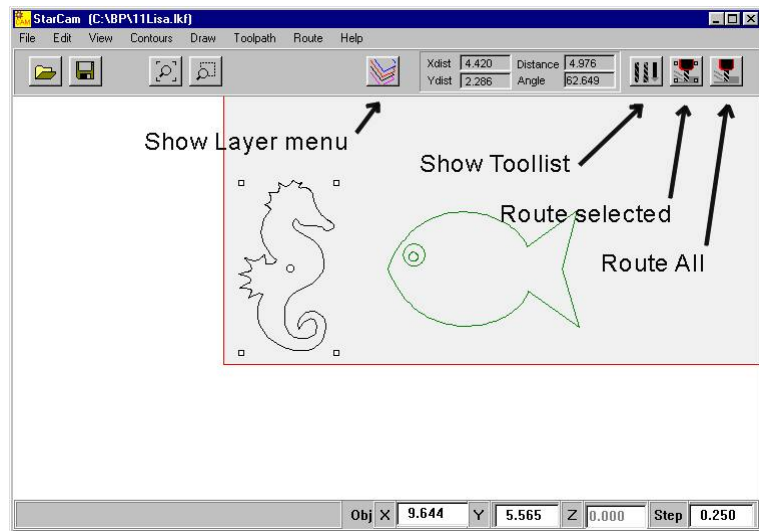
Layout your drawing

4 - You can then set up the file better for controlling a router by set/ting direction of travel, setting depths of cut etc. Star Cam can also do scaling, copying, moving, rotating and has many other editing features. It will also allow you to position drawings on the screen, and can sort your file to remove unnecessary tool lifts.



Select Objects to Route

5 - In the Route menu you can select **Router All** or **Route Selected** depending on what you want to cut. Select one of these commands to send the router file to the Star Cam Spooler program.



Run the StarCNC Spooler

6 - Run StarCNC from an Icon on the desktop. This runs the *StarCNC Control Spooler* that controls the router table. The spooler program sends commands to the Motion Controller in the Black box to move the 3-axis router table.

- You then switch to StarCNC, which runs your file and controls the XYZ table. For information on using the StarCNC program, see that section in this manual.



LAYERS

The Layer menu is an important part of the Star CAM program.

Layers allow you to separate the drawing into easily controllable sections.

Each (section) Layer can have a different depth, speed or tool#. You can turn off layers that you don't want to cut or protect layers from being edited. Also cutting order is easily controlled as it cuts from top (layer1) down in the Layers Menu, a maximum of 50 layers are available

Two types of depths within objects

There are two types of contours depth control within Star Cam. Layer controlled depth and 3D contours.

When flat 2d objects are imported into Star Cam such as from CorelDraw using the HPGL or EPS format or flat objects drawn in Star Cam, there is no Z-axis value. Star Cam puts a *Flag number* into the Z coordinate of each vector of the contour that is substituted with the *Layers depth setting* at time of routing. This allows you to change the depth for cutting easily by just changing one number for each layer at the machine control. This works well for a lot of 2 ½ D cutting and multiplayer woodworking and sign making.

(Note: The flag number is the value +100. since +100 is 100" above the table this number is never used for cutting)

When objects are imported using Gcode there is usually a Z coordinate that is imported from the file. This file is then a true 3d file and the Z value is fixed within the vectors of the contour. The Layer depth setting does not affect the depth of this type of contour. (Use the Properties command to display the contour type.)

When importing DXF files, you will be prompted to select whether you want use the Z value within the file or convert it to a Layer controlled depth.

The **Set_Zdepth** command in the Edit menu lets you change objects to use Layer depths or a fixed Z depth setting.

Changing layers and depths

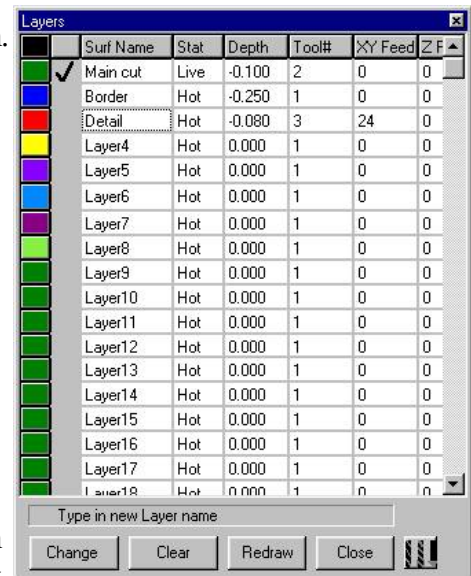
- Select the Layers menu. To set the cutting depth of objects in a layer, move the box with the cursor keys or mouse to the 'Depth' column and to the layer you want to set (set by color). Type the depth (negative in inches).

Star Cam uses the words **Off**, **Cold**, **Hot** and **Live** to describe the status of a layer

Off - the layer does not show and cannot be edited or referenced.

Cold - the layer shows and can be referenced but cannot be edited.

Hot - the layer shows and can be referenced and edited.



Live - Only one layer can be 'live'. Live indicates the layer that you want newly created objects to be added to or the layer that you want to move objects to.

-You can select the live layer by clicking the mouse on the word 'Layer' of that layer, or by moving the cursor to the status column and pressing L. Also clicking in the Live Check column will select the layer live

- Clicking in the **Stat** column changes the layer stat to the next type
- You can edit the name for easier Layer reference and clarity.

Tool number

The tool number is used for using manual or automatic tool change routers. This can be set to prompt the operator to change the tool at run time.



Clicking on the tool list Icon on the bottom of the Layer menu displays a list of the available tools. To define new tools select the Tool List command in the Tool Path menu

When objects are routed, parameters for selected tool such as xy feed , z feed and RPM etc, are sent to the controller for that layer.

The tool number is also used when displaying a simulated tool path using the View_ToolFill mode for that tool on that layer.

XY Feed and Z feed

The XY and Z feed settings are an override of the setting in the tool list. If this setting is not Zero (0.0) then Starcam will use it instead of the tools feed setting. This is useful if you need to have one layer run at a slower or faster speed than normal, such as a final light trimming of a part. If these settings are zero, then they will be ignored.

(Note: *Z feed* is not used when cutting 3d contours, only on Layer controlled depth contours)

Rotation Angle

On systems with a 4th rotary axis, you can set the angle for the rotary head to turn to before the start of each layer. This angle is 0 - 360 degrees.

Change Button

To change objects from one layer to another, first select the object on the screen to change, then set the layer that you want to change them to, to **Live**. Then click the change button .

Clear Layer Button

Press the Clear button to erase all objects in a layer.

Swapping Layers

You swap Layers order between two layers by dragging the colored square (on the left side) to another colored layer square. This will change their cutting order.

Menu Commands

FILE MENU

NEW This command lets you erase the entire drawing so you can start new. Caution data cannot be recovered after this command.

LOADJOB This command lets you load a previously saved job. It loads the data along you all layer information. If there is a drawing in memory be sure to erase it first.

SAVEJOB This command lets you save the entire state of the job that is in memory. All layer settings; depths and cutting speeds are saved with the job file. Do not enter a file extension, as the program will automatically add the 'LKF' extension. Note: LKF files are Larken Format only and not compatible with any other software. (E.g.: Corel has its native CDR format 'similar concept')

Import HPGL Import an HPGL file at the current cursor position. The sort option should normally be enabled to sort drawings when they are first imported. This removes unnecessary tool lifts and negative tool up movements.

For your reference the data in HPGL file looks similar to this: These are 'Select pen' 'Pen Absolute xy' 'Pen down xy' and 'Absolute Arc' HPGL commands.

```
SP2; PA0, 0; PD 2543,123;PA1000, 100;AA, 1000,100,180,3;
```

Import DXF Import a DXF format file. Lcam's DXF import loads only simple entities. Blocks, text and curve fit planes are ignored. DXF is useful though for importing 3D lines from Cad programs such as Design Cad and AutoCAD since Z coordinates are read in.

Note: Use Revision 12 dxf files. R13, R14 ,2000 and above are not compatible.

Note: If you want to erase the Z coordinate values that the DXF has read in to Starcam, use the Set Z depth command in the Tool menu and set the new depth to Layer controlled Z depth

Import GCODE Import a Gcode 'NC' file type. Some programs designed to interface with industrial machine controllers use Gcode. The default file extension is '. NC'. The default format is decimal, absolute and supports arcs. G00, G01, G02, G03 are supported.

For your reference the data in a Gcode file should look similar to this:

```
G00 X1.000 Y1.000
```

```
G01 X2.250 Y4.500 Z-1.125
```

```
M30
```

Export DXF Export's all the drawing that is visible to a DXF file in 2D format.

EDIT MENU COMMANDS

SELECTING OBJECTS You can window objects with the mouse or click on them to pick them. To **Unselect** objects, hold down **Shift** while picking or windowing them. Its best to window arcs, because currently StarCam can't select an Arc, by picking it.

MOVING OBJECTS There are a number of ways to move objects.

- Dragging them with the mouse while selecting the center of the group.
- Moving them with the keyboard, by holding down ALT and SHIFT while using the Cursor keys
- Using the Move command in the Modify Dialog box to enter a move distance.
- You can use the + - keys to rotate the objects as well. The rotate increment is a setting in the Configuration menu.

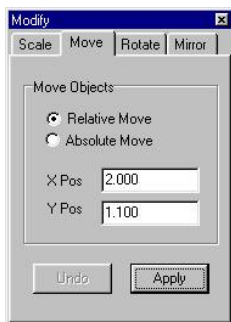
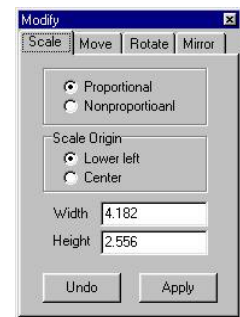
COPY The Copy command copies the selected objects to the Star cam clipboard. You can then use the Paste and Array Paste commands to place the objects.

PASTE Use the mouse or cursor keys to position the object where you want it. You can use the zoom command while pasting objects to get very accurate placement.

Before using the paste command you must have used the copy command.

ERASE Erases selected objects. *Delete* is the hotkey.

SCALE You can scale the object proportionally or non proportionally. If non-proportionally is selected you can enter separate width and height. Note the numbers you enter are the actual size you want it to be, not a scale factor. You can also specify the origin for scale on center or lower left.



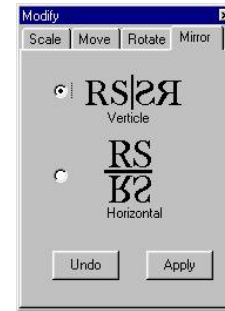
MOVE The move command is in the modify toolbox. You can select absolute or relative positioning.

ROTATE The Rotate command allows an accurate rotation of the objects selected. Enter + - 360 Degrees



MIRROR

The mirror command mirrors on center along the Horizontal or Vertical axis. Use the Undo button to reverse the command



3D commands and Depth control

Layer controlled depth and Zdepth by Setting.

Within Starcam, there are **two** ways to control the Z depth within a contour. Each contour can either be a 3D contour, or a 2 ½ D contour with its depth controlled by the Z value in the Layers menu

3 D contours have a specified Z number value for every vector. 3d is used for carving contoured surfaces and 3d paths that

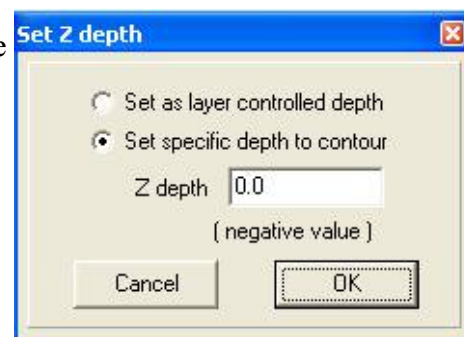
2 ½ D contours have a 'Flag' in the Z value that tells Starcam to use the Layer menu's depth setting for that layer. The advantage of this is easy control of the depth for cutting flat material such as sheet wood and plastics. Also Starcam can output automatic multi-pass cutting depths for cutting thick material.

Automatic Multi-Pass for 2 ½ d Objects

To create multipass tool paths, specify in the Layer menu the tool # that you would like to use for that layer. The '*maximum depth of cut*' setting for each tool in the *Tool Menu* specifies how deep the cutter will cut in one pass of a Layer controlled object. EG: If the *Max depth per cut* is 0.25" and the Depth in the Layer menu is 1", Starcam will go around the object 4 times each time going deeper by 0.25".

SET Z DEPTH As well as using the layers to set the cutting depth of an object, you can also assign a specific depth to an object. This allows an unlimited number of depths in a drawing for very complex work. Select the object(s) and then enter the depth.

To reset an object so the depth is controlled by the layer depth setting, click the 'Set as layer controlled depth'



LAYOUT MENU

PROPERTIES (F6)

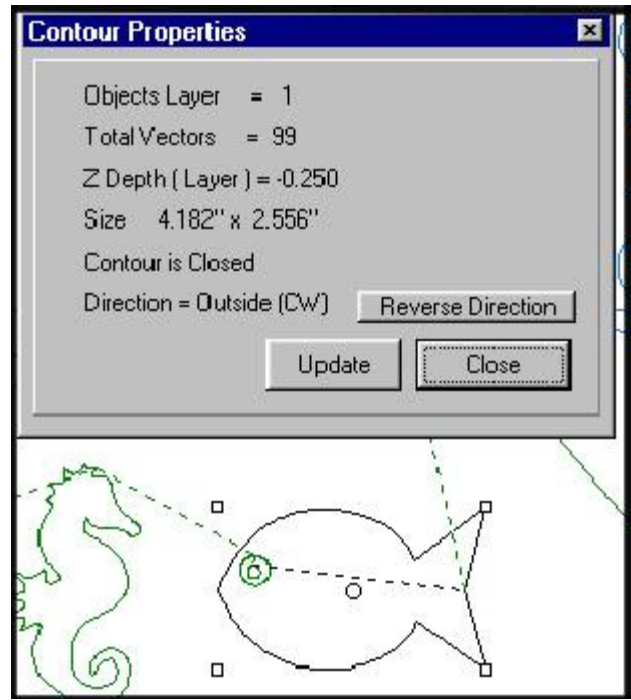
Select an entity and this command displays the path direction, number of segments and whether the entity is open or closed. Select the Properties command in the Layout menu or by the hotkey **F6**. Click Update to get info.

The properties shows information about the object such as the Layer its in, the number of vectors, and its XY size.

The Z depth will show (Layer) if the object uses the layer depth setting for its depth. If the object has had a specific z depth assigned to it with the Depth command then (setting) will appear at the z depth parameter.

The direction can be reversed by clicking with the direction button. It will show if the contour is CW or CCW.

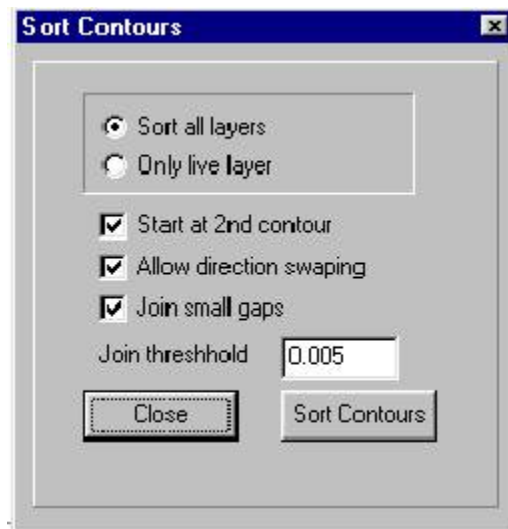
It may show ??? if it can't determine the direction, but the Reverse Direction will reverse the direction anyway.



Sort Path This command sorts the displayed layers and connects lines for the fastest routing time. It also reduces unnecessary tool raises by joining lines that are not connected but are within the parameter *join-threshold* setting.

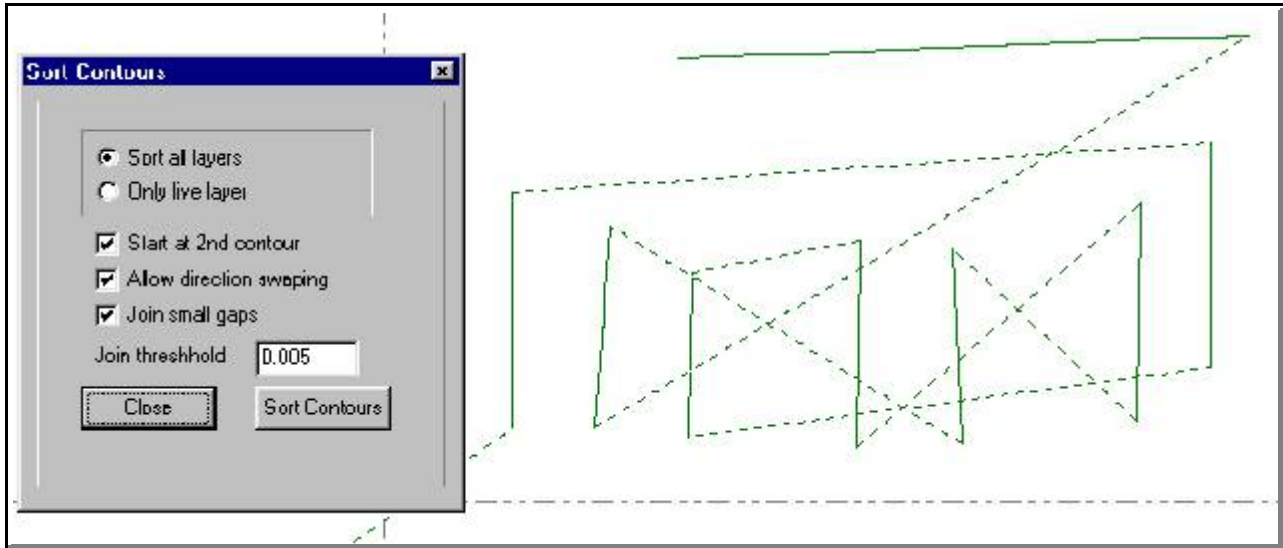
You can use the Analyze command in the tool menu to see the number of tool lifts in the drawing before and after sorting.

The 'start at 2nd vector' setting allows you to manually set the first vector with the **Set_first** command so you have control of where the router goes first. The 'Allow Direction Swapping' setting allows you to prevent the sort command from changing cutting direction of contours.



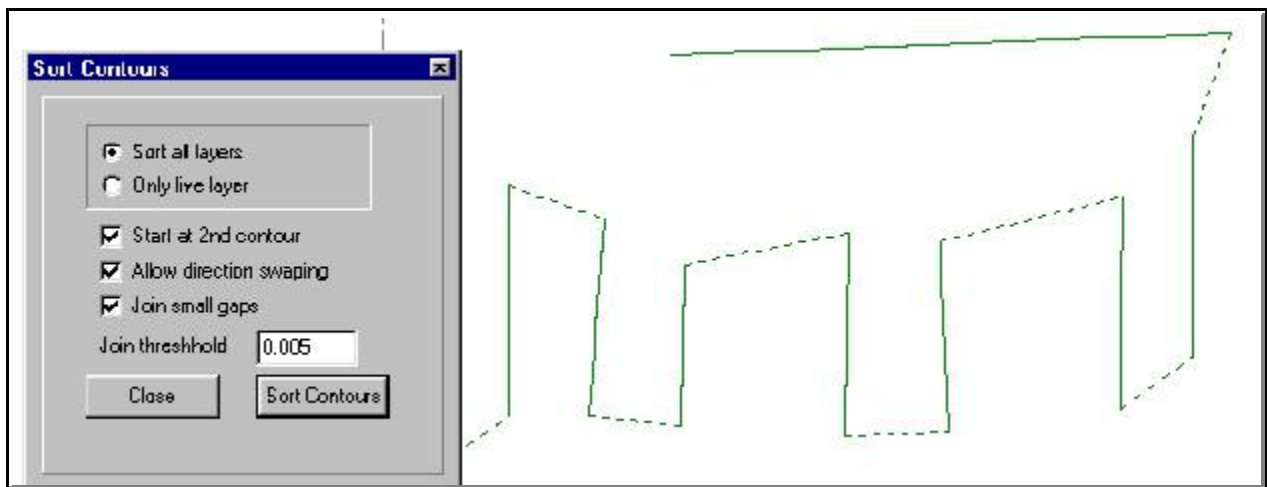
Sorting Examples -

Below is a simple line drawing before sorting. After sorting the Rapid tool movement distance had been greatly reduced.



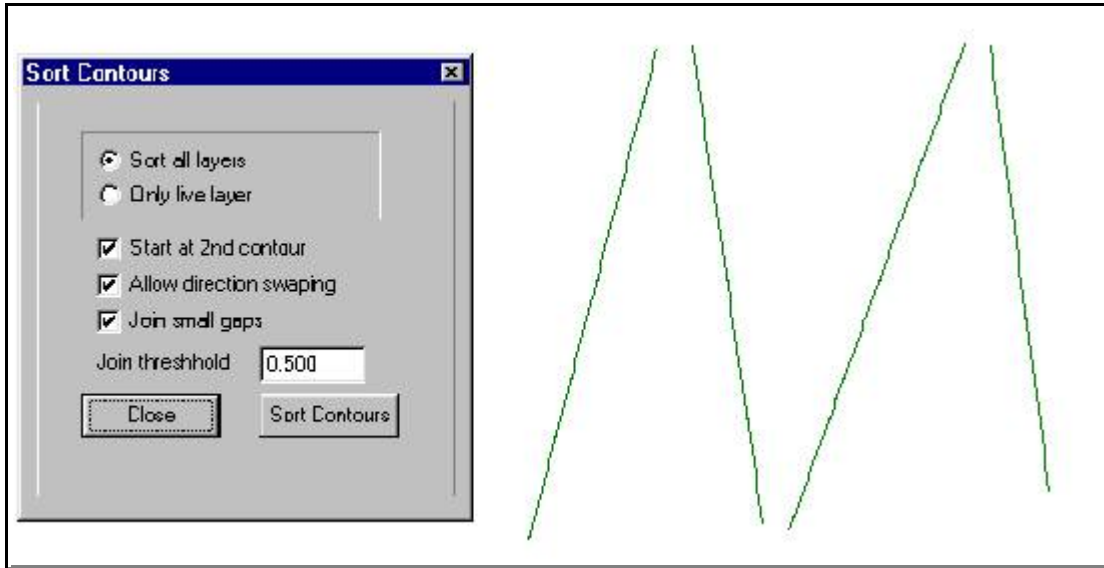
Note that the sorting started after the 1st vector, and then looked for the closest vector. It checks each end of the vector to see which is closer and reverses the vector (if Allow Dir Swapping is enabled).

Use the **Set First** command before sorting

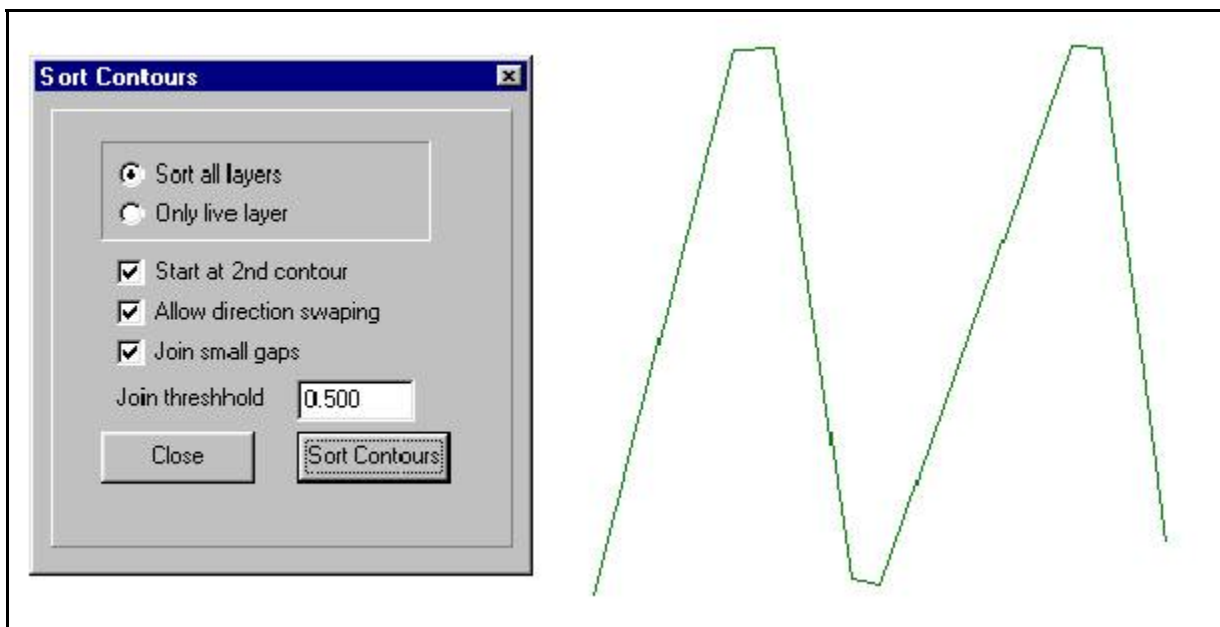


Sorting Examples - Joining gaps

Below is a simple line drawing before sorting. The lines are not connected.



The join threshold setting is increased to larger than the gap distance. (This can be measured with F5). After the sort command, the lines are now joined and are 1 continuous entity (a polyline).

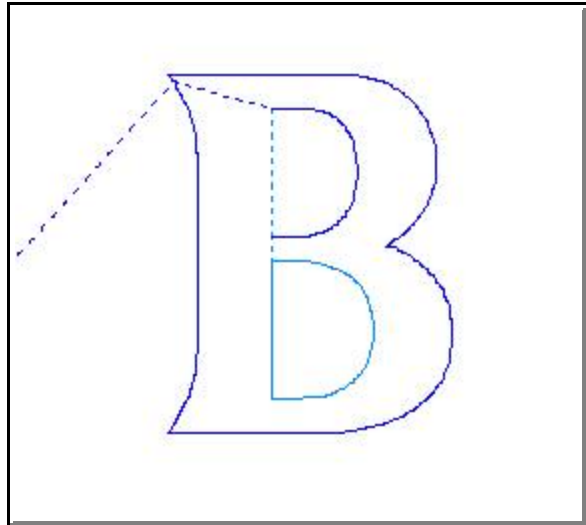


SET FIRST CONTOUR

This command lets you set the order that the entities of a layer will be cut in. Each time an entity is selected, it will be set to the beginning of the other entities and the entity that was first will now be second. Therefore to set the order of a group of entities, you have to set them in reverse order.

Use the mouse to Click on a contour to set first since you can only select one contour. (Don't use the window select).

In the example, the rapid moves to the outer part of the letter B first. This cut order would be wrong, since the letter would be loose when it went to cut the middle.

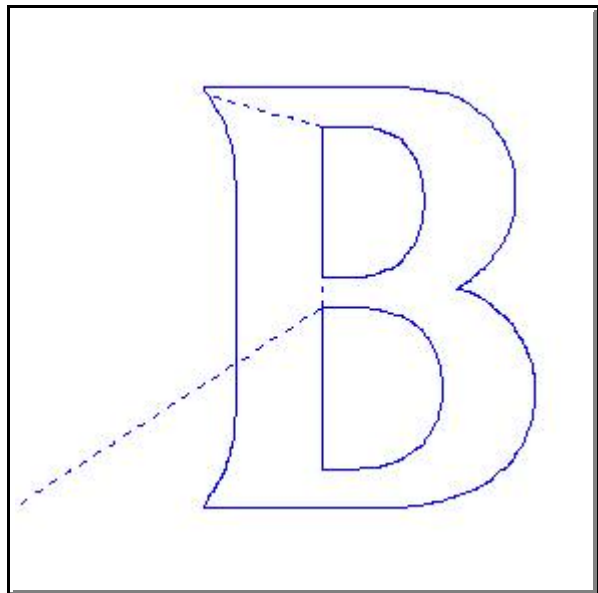


Using the **Set first** command, the inner cuts were selected, which moved them to the front of the layer list. This would require you to use the Set First command twice, each time selecting one of the inner contours.

After you use Set First, you can use the **Sort Contours** command to re-order the cutting order starting after the contour you have set first.

The Set first command only works within the layer, and layers are cut in order from 1-50.

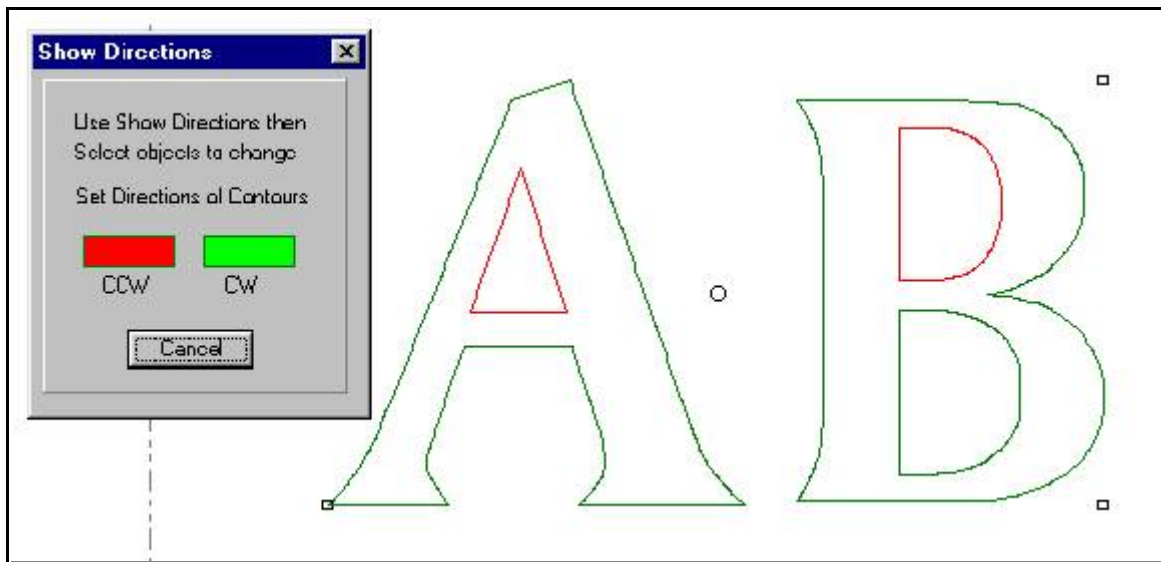
Separating your drawing into Layers is also a good way to control cut-order.



SHOW DIRECTIONS

This command shows you the cutting direction of selected object, so you can see if the cut CW or CCW. (CW is Climb-Mill, and CCW is Backmill or Conventional mill)

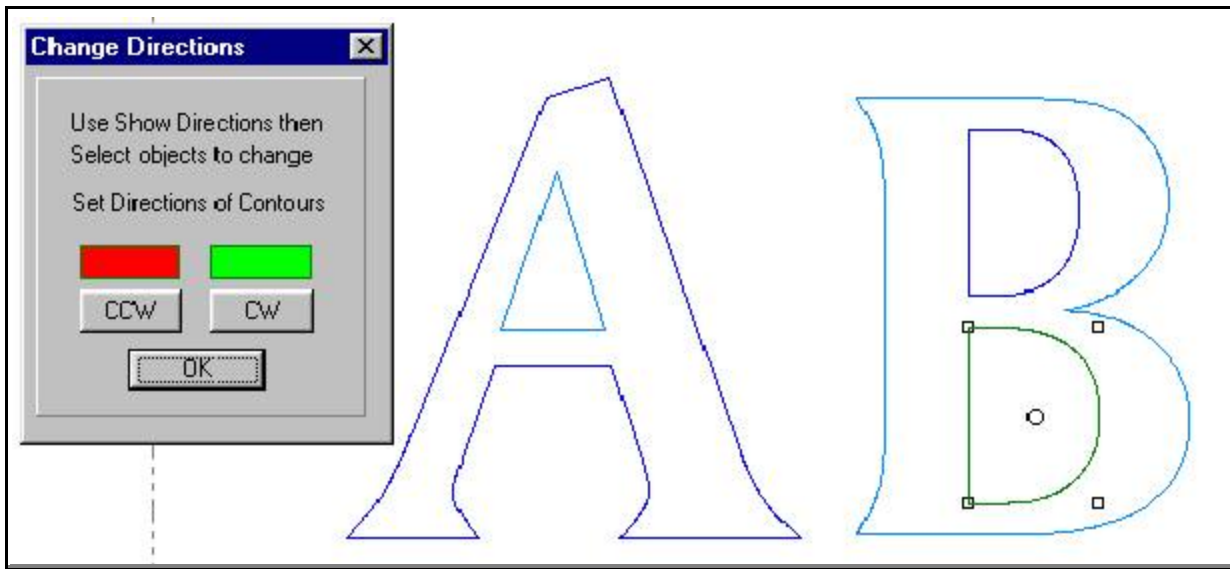
Select the objects first that you want to check and then use this command.



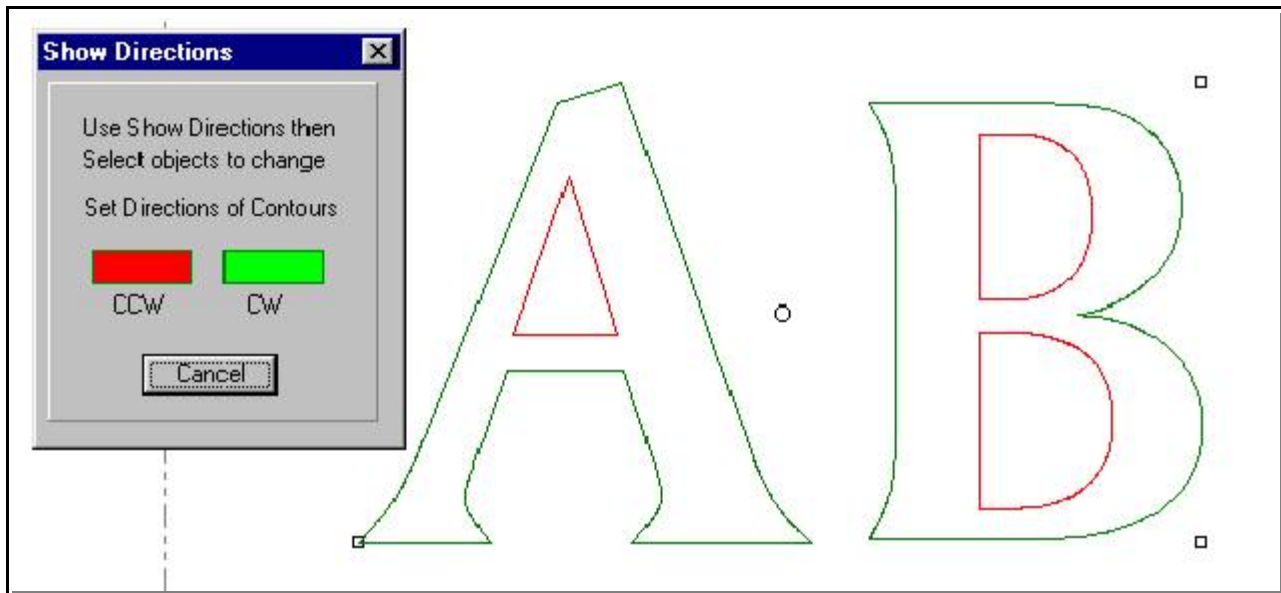
If you need to change the object direction, use the Change Directions command.

CHANGE DIRECTIONS

In the example above, only the lower inside of the letter B need to be changed, so just it is selected, then the Change Direction command is run. Then the CCW button is clicked to change it to CCW. Then OK closes the command.



To check the result, select all the objects and use the Show Directions command again.



Now all the insides cut CCW (climb-mill for inside) and the outsides cut CW (climb-mill for outside)

Note - Some objects are too complex for Starcam to determine if they are CW or CCW., which Starcam displays as Black when using the Show Directions command.

Also to reverse a contour's direction you can also use the **Reverse Direction** button in the **Properties** command (F6)

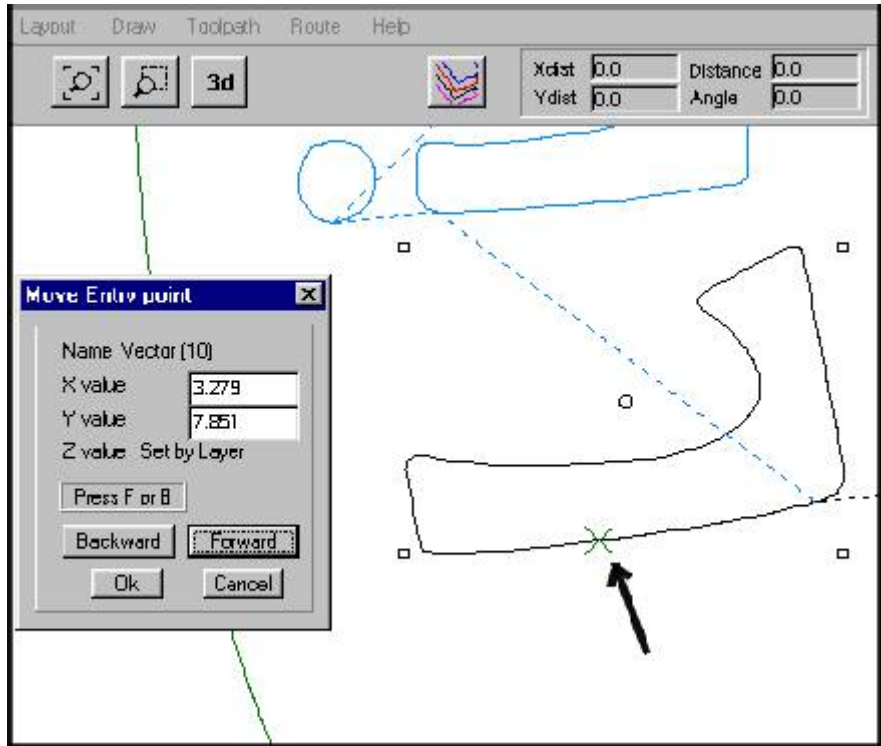
ENTRY-POINT

This command lets you change the position where the tool enters a entity. Select the contour, by clicking on it with the mouse and an 'X' will appear where the current entry point is. (Note: **do not window select as you want to select only one object**)

Closed object

A closed object is one where the last point in the contour is exactly in the same XY position as the starting point.

Use the 'F' forward and 'B' back keys to move the entry point along the contour. Press Ok when you are satisfied or press cancel to abort the command.



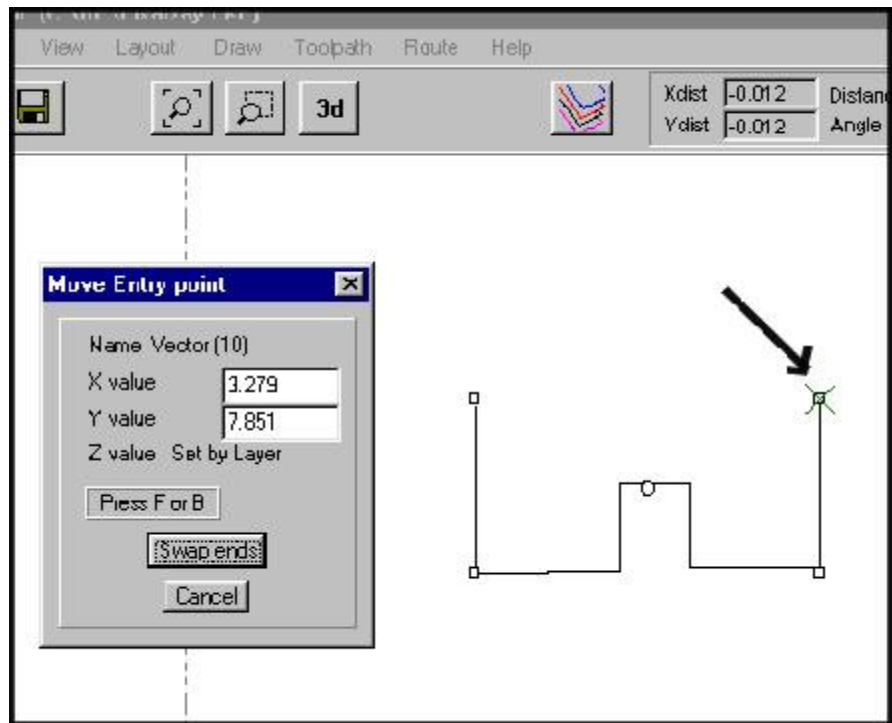
Open Contour

An open object is one where the last point in the contour is **not** in the same XY position as the starting point.

Use the 'Swap ends' button to reverse the contours end point. Press Cancel when you are satisfied.

If your contour looks closed but comes up with the swap ends box when you select it, then its not exactly closed.

Use the Close contour command in the Layout menu to close it.



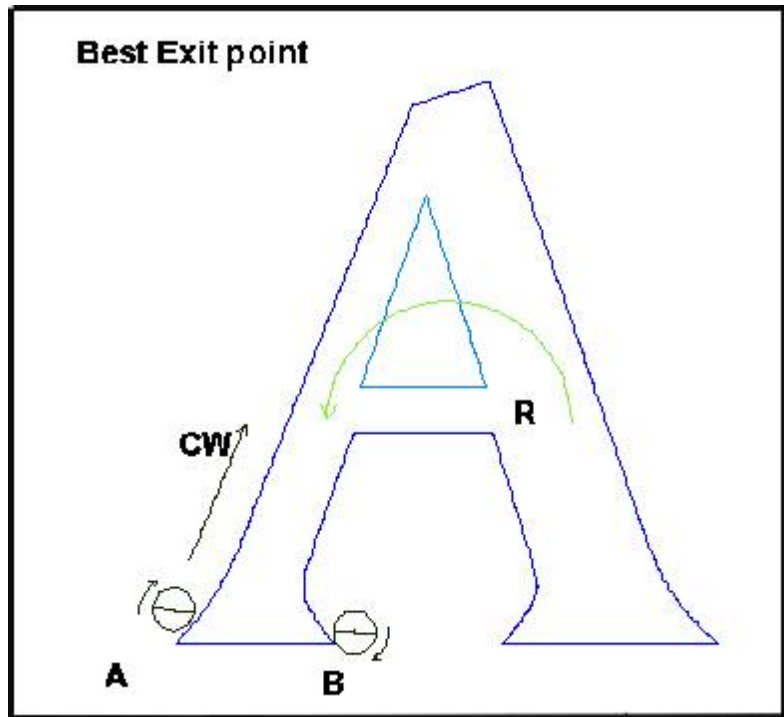
Setting Tool Exit Point

When cutting objects on the router, setting the best entry and exit point will keep the piece from being damaged and nicked when the object comes free.

It's a good idea to take at least 2 passes. The final cut should just take a small amount fo there is little force when the object is finally cut free.

EG for 1/4" plastic, cut 0.2" for the 1st pass and then .06"-.075" for the final pass.

Since the cutter is turning CW (clockwise) if the exit point in the diagram was **A** then when the letter was cut free, the letter would want to move CCW as shown in **R**. It would move away from the cutter and not be disturbed.



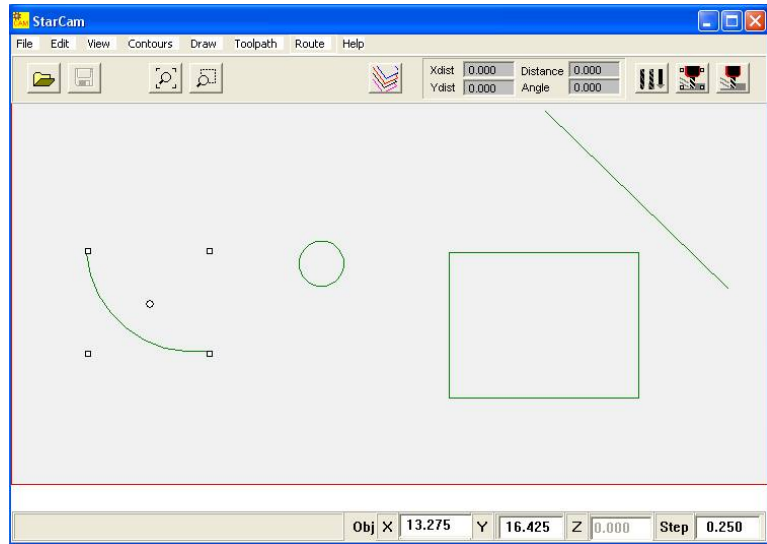
If the exit point was **B** when the letter came loose, the letter would want to rotate CCW and would be self fed into the cutter and be damaged and thrown.

DRAW MENU

In the Draw menu of Star CAM they are some basic drawing commands that allow you to do basic CAD. Although we recommend that you do your drawing with other Cad and Graphic programs, Star CAM has some good drawing features itself.

Lcam has very good cursor control with the Keyboard and an active zoom command that allows zooming within commands.

There are a few features that help drawing such as Snap to intersection and definable cursor step. Also you can move the cursor with the mouse, cursor keys or enter the absolute or relative coordinates



Here is a list of function keys useful in drawing.

- | | |
|-----------------|-----------------------------------|
| Alt-Cursor Keys | Move Cursor |
| Alt-Enter | Enter points |
| Alt-PgUp | Double Cursor Step |
| Alt-PgDn | Half Cursor Step |
| ; | Enter absolute coordinates |
| ' | Enter relative coordinate |
| * | Snap to intersection |

also

- | | |
|-----------|---|
| F5 | Measure distance |
| F6 | Properties for changing directions |

There are commands for drawing Lines, Arcs, Circles and box.

When drawing, pressing ESC cancels the command. You can snap to existing geometry while drawing to get exact connections. Also press 'Z' for zoom and then zoom anywhere while drawing.

Drawing using the Keyboard (no mouse)

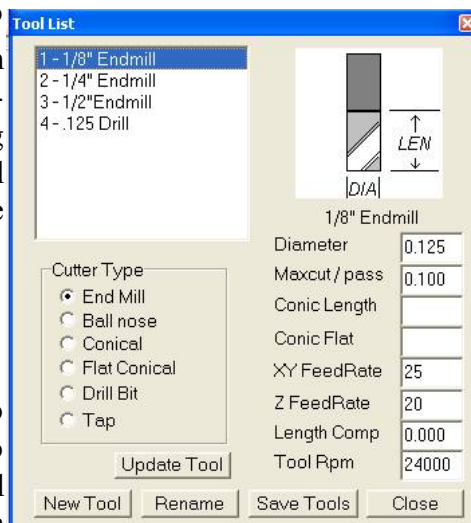
You can draw without the mouse (for high accuracy) using the Cursor keys (holding Alt) and use Alt-Enter to enter points.

TOOL MENU COMMANDS

TOOL LIST The tool list is a user-definable list of up to 30 tools. You will be asked the tool type, diameter, length (if it is a Vee type), maximum cut per pass, and a 10-character tool name. In the layer menu the tool setting indicates the tool in the tool list that the layer uses for tool offset, hatching and tool path simulation. When you route the job, you will be prompted to change tools.

To add a new tool:

Click on the NEW TOOL button, you will be prompted to enter a name for the tool (ex. 1/4" End mill), click Ok to add your tool name. Choose the Cutter Type and fill in all information that is applicable to that tool than press then update Tool to save.



To edit existing tool:

To edit a tool click on it's name, after you made the changes click on Update Tool to save changes.

The Maxcut/Pass parameter is used by Starcam when you are cutting objects with Layer controlled depths. This setting is always positive. This is used to multi-pass cut flat 2d objects.

Peck Drill

If the tool is a drill, the Maxcut/Pass parameter is used set the **Peck-Drill** depth.

The RPM setting will be passed to the controller and will control spindle RPM if the spindle drive is a variable speed inverter-drive.

ROUTE MENU COMMANDS

Enable StarCNC This command launches the StarCNC spooler program to run the router file (temp0.LKS) that Star Cam creates using the Route commands.

(See the StarCNC section in this manual for more information)

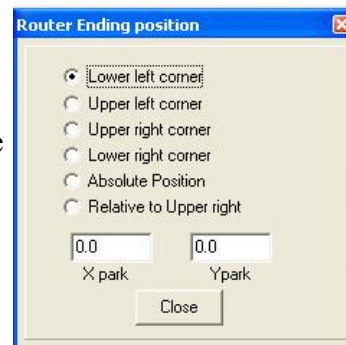
Route All This selects all visible data on the screen to be sent to the router controller. (All layers except layers turned Off will be routed)

Route Selected This lets you select just parts of a drawing to be routed. This is very useful if you need to re-cut a section because of a broken bit or wrong depth etc. Select the parts you want with the select menu, and then select 'done'. While selecting you can use the zoom command to move around and refresh the screen.

Route item at Origin This command effectively moves the selective objects to the origin and lets you route them there. This command is great when you have a lot of assorted letters to cut on scraps. Put all the letters in one file and then use this command to route each letter right at origin on the scrap piece without having to move the letter on the drawing screen.

Rotate all 90 Deg You can rotate the entire job 90 deg at a time with this command to let you position the job on your workpiece. This command also does some cleanup of the drawing and can be used to remove negative move problems.

Set Ending position Select which corner of the object you would like the router to return to when it's done cutting. Use the 'save' in the Parameters command to save this setting.

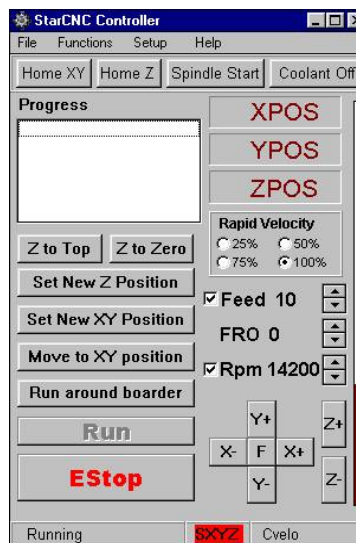


The StarCNC Spooler

The StarCNC system is comprised of a 'motion control processor' (MCP) circuit board in the control box and a windows spooler program. The spooler reads either a Starcam file or a Gcode file and sends it to the MCP buffer. The spooler also continuously displays coordinates in real time as well as velocity and status of the hardware. It can spool huge files to the (MCP) so file size is unlimited.

The PC is connected to the control box by the printer port. The printer port was chosen over the serial port for its ease of configuration and its highspeed data transfer.

Note: *Be sure to remove any software that directly talks to the printer port such as Zip drives, File transfer programs, scanner drivers etc., as these can interfere with communication to the controller.*

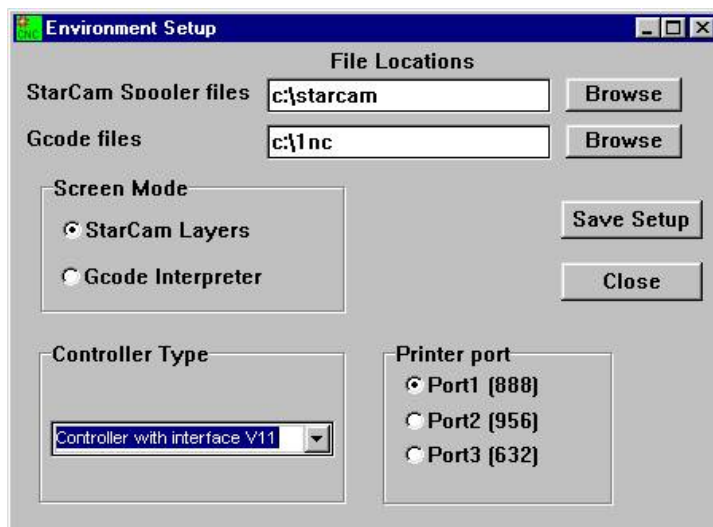


Environment Setup

You may need to change the printer port setting in the spooler by selecting the Environment settings in the setup menu. As you change the port setting you may need to reset the controller. If the coordinates in the spooler display 0.000 then the port is correct and you have communication. Be sure to click "Save Setup" when the correct port is found.

You can preset the default directories that you will load the Gcode and Starcam spooler files from.

Selecting Screen mode for Gcode or StarCam mode selects the appearance of the StarCNC program. In the Starcam mode the displaybox displays the Layer table to allow you to turn layers on and off and to set layer depths at cut time. In Gcode mode, this box displays the Gcode as it is read.



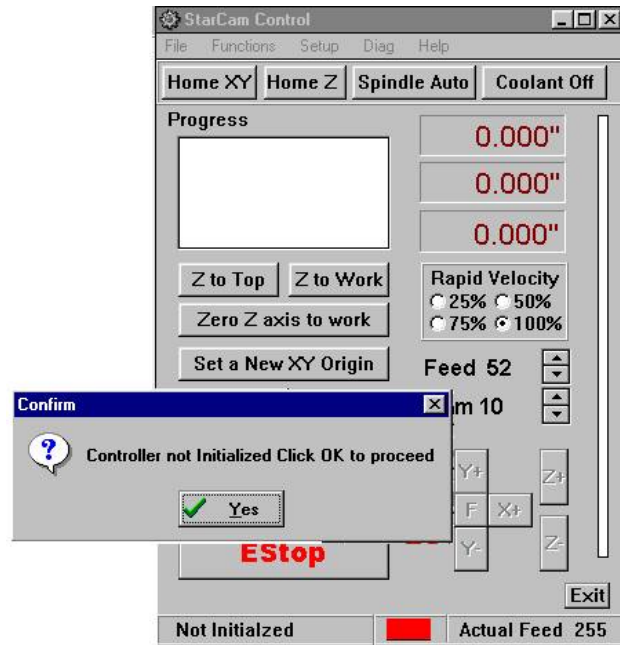
The Controller type can be 3 types : Controller board without interface, with interface type 11 or with interface type 5-10. If step/direction wires connect directly between the controller MCP and the Stepper drivers, then you do not have an interface. The interface will be connected to the MCP with a 25pin ribbon cable. If you have a green interface with plugon connectors, then it is a type 11 interface. If the interface has mostly soldered-on wires, it is a type 5-10 interface. *Contact Larken for the correct setting if you are unsure.*

Initializing the Controller

The cable used between the PC and the controller should be a male to female straight though extension cable, preferably a new IEEE shielded type for high speed printers.

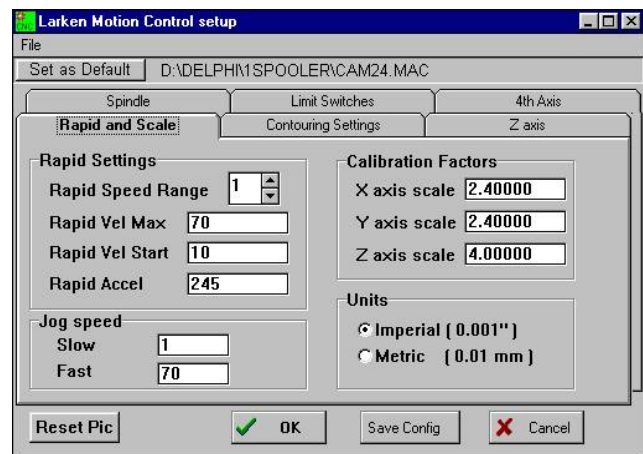
With all cables properly connected the next step is to check that the PC is communicating with the MCP.

The procedure to start the system is first to turn the control box on. The Led on the front of the controller should flash. Then run the spooler and a message 'Controller not Initialized' should appear. Click OK and wait about 10 seconds while the spooler initializes the controller and downloads settings to it. (You should notice the 'Feed' display at the bottom right run from 1-100)



Configuring the Machine settings

Select the Configure Machine command in the setup menu to change the motion control parameters for controlling the CNC router of 3 axis machine that you are running. A list of preconfigured setups for Larken machines is supplied and can be loaded from the File menu. Goto the File menu and 'Open machine Config' then select the correct file for your machine. Then use the 'Set as Default' button to have it load automatically



Exit the program and rest the controller and reload after changing the config file.

For custom machines you need to modify settings to make your machine run correctly. If you have any questions please contact Larken Automation.

Controller Overview

The 'motion control processor' circuit board (or MCP) outputs pulses to the 3 (X,Y,Z) motor drivers to move the machine. The timing between the pulses is controlled in a very precision way using a digital timer in the MCP. Stored inside the MCP is a table of 100 timer values from slow to fast. This table is used in acceleration and deceleration to change velocities. The controller

may start moving at table position 10 and gradually move up the table to setting 100 causing acceleration . These are the motion profile settings seen above (EG: Rapid Vel Max, Rapid Vel Start)

There are separate settings for Rapid traverse (when the tool is up and the machine is moving to its next cutting position) and settings for cutting. Also the time the controller waits on each step as it moves up the table determines the rate of acceleration.

In addition to the 1-100 velocity table, 4 *speed ranges* (like a 4 speed transmission) can be set. The master clock signal to the timer can be set to different speeds to select 4 different speed ranges of pulses. Typically you would set the *speed range* to where your machine could reliably move with Rapid vel max set at its fastest rate (100). The speed range can be set to a slower range for cutting moves if required as it has a separate setting. The *speed range* is fastest when set to 0 and slowest when set to 3.

The Accel parameter sets the rate of acceleration. It is a setting between 0-255 with 255 setting the fastest acceleration. Generally numbers between 210-250 are used. The higher the number the faster the acceleration. This setting is depends on the weight of the machine and the performance of the motors etc.

Calibration Factors

To calibrate you xyz table to travel the correct distance, the X,Y and Z scale factors need to be set.

The setting is defined as - The number of motor steps required to move the 0.001” or (0.01mm for metric).

EG: if your x axis has a stepper driver that has 1600 pulses per turn, and you are using a lead screw that has 2 turns/inch then it takes 3200 steps to move an inch. Or to move 0.001” it takes 3.2 steps so you would set the X scale factor to 3.20.

Note: To convert an inch calibration to metric, divide the inch scale factor by 2.54

Using the StarCNC spooler

Jogging

Jogging the machine can be done using the mouse, or keyboard. To use the keyboard, hold down **Alt** while using the arrow keys for X,Y and use **PgUp** and **PgDn** for the Z axis. **Home** and **End** select fast and slow jog.

Homeing XY and Z axes

Use the Home XY or Home Z buttons to make the machine seek its home switches. It will set the position to Zero (0) after it homes. Homeing XY and Z speed are parameters in the Machine configuration .

Setting Origin

To position the tool at a starting point of origin to cut, jog the machine to the required position and select the *Set New XY Position* or *Set New Z Position* buttons.

Selecting the spooler file.

To cut a file you need to select then file first. Using the *Open Spool file* command in the file menu and select the file named 'Temp0.LKS'. Now you will notice that the Run button is enabled. Note: The 'Temp0.LKS' file is created by the Starcam program using the Route objects command.

Running

Click the Run button to start the job. The bottom prompt will display 'Running' instead of 'Local' .The spindle will start and the machine should start cutting. Press the Estop button or the Stop button on the controller to stop the job. The controller will automatically stop the spindle at the end of the job.

Feed Rate Override (FRO)

While the machine is cutting you can speed up or slow down the cutting speed by adjusting the FRO setting. The change takes place on the next vector , so if you are cutting a long line you will not see the speed change until the end of that line.

GCODES and MCODES

Gcodes supported

G00 Rapid move EG: G00X2.5Y10.025

G01 Linear Feed move EG: G01X3.5Y6Z1.25

G02 Arc X,Y,I J XY are absolute and IJ are relative.

G03 Arc X,Y I,J

G04 Dwell P 0.25

G83

G84

G90 Absolute coordinates

MCODES Supported

M03 Spindle Start S - set spindle rpm EG: M03 S16000

M05 Spindle Stop

M06 Pause for tool change T tool# EG: M06T5

M08 Auxiliary output 1 ON

M09 Auxiliary output 1 OFF

M10 Auxiliary output 2 ON

M11 Auxiliary output 2 OFF

The 3D Digitizer Probe Option

The 3D Digitizer probe option is available on all Larken routers. To install the probe, remove router plate and mount the probe instead. The small plug connects to the 2 pin connector next to the spindle AC power connector.

To test the probe, run the StarCNC program and notice that when you move the tip of the probe the W axis home indicator on the bottom display bar of the StarCNC should show.

The probe menu is located under the Functions menu. If there is no probe menu, you may need to contact Larken Automation for an updated security password.

In the probe menu, you can set the size of area you would like to probe. Also the sample pitch is the probing resolution or the distance between probe samples. Setting the sample pitch too low will result in very large file sizes and long probe times, since every time you reduce the pitch by 2, the probing time will take 4 times longer. Also the accuracy of the probe is approximately 0.010”

The probing speed cannot be too high or loss of steps will occur since no acceleration is used when probing. Generally speed is 15 - 25 %.

Setup for probing : Make sure the object to be probed is securely mounted on the table. The sharp probe point may mark the surface, so don't probe delicate items.

Jog the XY axis to the lower left (most negative) corner of the object to probe and set both X and Y coordinates to 0,0. Set the Z axis to the lowest point that you want probed and set the Z axis coordinates to zero. Set the X and Y size to the size of area that you want probed. Ensure that the Z axis can raise up enough to clear the top of the object with out hitting the Z limit switch . You can now press the *Start Probe* button. The pause button allows you to pause the probing without stopping the job, to make adjustments or fix a bad contact situation.

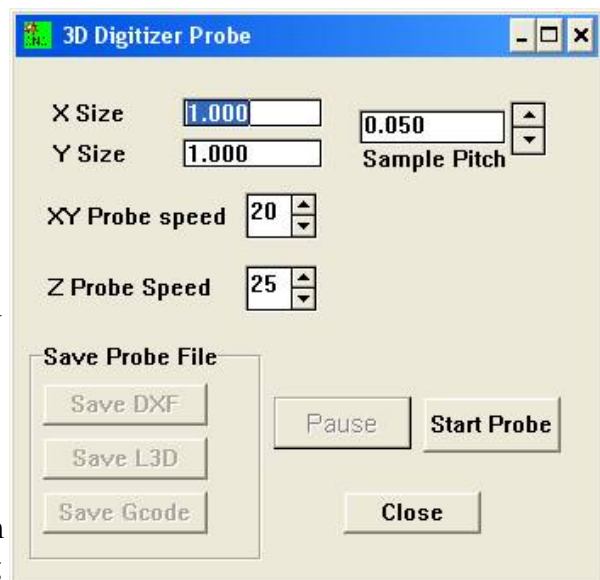
At any time you can click the Stop button to stop the system and abort the job. Do not close the probe window unless the job is stopped or Starcnc cannot close properly.

When the probing is finished you can save the 3D probed information in a number of formats.

DXF format is a very common format for Cad programs. The surface is saved in 3D Face entities (4 sided polygons).

Larken Surfer (L3D) format can be read by Surfer 3D.

The Gcode option saves the data as a tool path just as it was digitized in a serpentine contour using G01 commands. You can put a small plastic ball in



the end of the probe the same size as the cutter that you will use to route it for direct 'digitize to cut' duplication.

Importing Files from other programs

AutoCAD- DXF import is the autocad standard, but be sure to export the DXF file as a Revision 12 type. DXF versions of R13, R14 and greater are not recommended.

You can also use the HPGL format. You should configure AutoCAD to use a Hewlett Packard 7585 plotter and set the scale to 1=1. Set 'plot to file' and don't calibrate. You can do a detailed setup and set pencolors for all 8 pens (8 maximum in AutoCAD). AutoCAD can only support plots up to 65 inches (at .001" resolution) because the plot variable in AutoCAD is a 2-byte value. Use the Plot command in AutoCAD to export the design and give the file a '.PLT' extension.

Corel Draw! Corel Draw is an excellent companion program for StarCam. Version 4 or later is recommended since it has excellent contour (offset) and weld commands command. You can use the contour commands for cutter offset and object fills. In the example a letter 'B' was made inside an oval then the 'to center' contour was selected. Generally the objects should be 'welded' before contouring to reduce errors. If you need to access the contours individually afterwards you must separate and break apart the contours. Use the Export command and select the HPGL format. The EPS and AI formats also work well especially if you need to enlarge them afterwards in Starcam since the curves are stored as beziers.

Design Cad 2D for dos Design cad is a very good and quick to use program, and can plot files any size using its plot command. You should configure the plotter as a HP 7585 plotter and plot to file. It has some nice single line fonts for small simple engraving and a very good spline curve command.

Design Cad 2D for windows Unfortunately the windows version doesn't have a plotter driver so you are supposed to use the windows HP plotter driver and print to file. We have had limited success with the Microsoft windows driver so we don't recommend using DC2 for windows.

Design Cad 3D With DC3 you can draw 3d lines and curves for carving with the router. The Z coordinate should be positive to carve downward. You will then need to transfer the file to DXF format with the DCFiles utility and use the DXF import command in Starcam to read it in.

Also with our **Surfer 3d**. 3d tool path program for Windows, you can have your router carve them out.

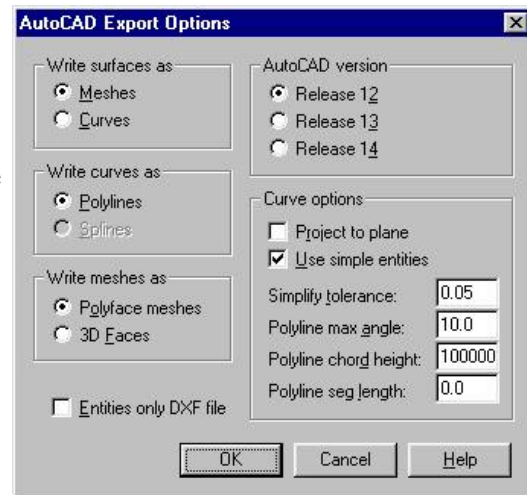
Rhino 3D Rhino 3d is an excellent 3d modeling program that is also affordably priced. It makes it easy to draw complex 3d objects for prototypes and signs etc.

3D Lines , 2d Arcs, circles and other non-surface entities can be directly imported into StarCam using a DXF file.

To mill the 3d surfaces with your router you would need a tool path program such as our Surfer 3D. Export your drawing in DXF format and import into Surfer 3D.

The Surfer 3d manual describes the procedure.

For more info on rhino, check out their website at www.rhino3d.com



Surfer3D

Surfer3D by Larken Automation is a program that creates full 3d tool paths from Cad drawings done in a 3d cad program. The drawing should be made of surfaces and shadeable within the Cad program. Surfer can import DXF files and generate roughing and the finish planar tool paths with 3d cutter compensation for ball nose, Vee and flat cutters. For more info on Surfer check out our website at www.LarkenCNC.com

Dedicated Sign making Programs That output Gcode files

The following sign making programs have many features that are more powerful than Starcam such as tool offsets and tool fills. You may wish to do all the tool path setup in these programs and just use StarCNC to run the Gcode file. It is a good idea, however to import the file into StarCam to display it if you have any problems.

CasMate Pro You will need to install the Larken driver for CasMate for windows. (contact CasMate or us if it is not on the system disks) There are 2 files 'Larken3d.ENG', Larken3d.CFG. These should be placed in the caswin directory. - - Go to the 'miscellaneous' menu and select 'setup' to install the drivers. - Pick the Larken driver and select 'Add'.

EnRoute Use the same drivers as for CasMate. EnRoute allows you to do a number of complex tool path types such as 3d carving using a small V cutter and multipassing it to cut a wide

letter. Also you can specify a rough, regular and fine cutter size. It separates the tools into different layers when you export to Starcam.

Sign Lab You will need to install the Larken driver for Sign Lab (contact Larken if you don't have the driver on your system disks). Use install driver from the plotter menu or you can add it to SIGNALAB2.INI manually in the plotter section.

Use the plot command to create the Gcode file. Set the tool as 'router'. In the color bar you can double click on a color and set the tool # from 1-20. This will let you transfer different colored entities to different Layers in Starcam. If you want to use Starcam layer system for depths instead of setting depths in Sign Lab you can check the setting in the Starcam Gcode import menu.

Turning Arc support 'off' in the sign lab plotter setup command lets signlab have more control of arc resolution but makes files larger.

Profile Lab You will need to install the Larken driver for Profile Lab (go to www.LarkenCNC.com, if you don't have the driver on your system disks).

During Profile Lab setup you will be asked if you would like to install drivers (make sure the box is checked) then click next. This will bring you to the driver installation screen select Larken Automation, click next.

Note: You can also access the Driver Installation dialog through your Windows Start button: Program Files/Profile Lab/Driver Installation.

Types of cutting tools:

Wood Router bits - Wood router bits generally are for high rpm cutting of woods and plastics. They can either be a straight flute or a slow spiral flute; usually they have 1 or 2 flutes. They are available in Carbide or HSS (high speed steel)

Milling cutters - Milling cutters are for machining metals and are only available as a spiral cutter. The spiral is a faster spiral (like a finer thread) than a woodcutter to prevent chatter when cutting. They are also available in HSS and Carbide. Milling cutters are available in 2 - 4 flute. The 2 flutes are better for clearing chips and are recommended over the 4 flutes. Be sure all cutters you use can plunge (drill) for tool entry.

Carbide vs. HSS - For most cutting on the Larken systems we recommend using carbide cutters. They will last many times longer than HSS and produce a better finish; also they can run at higher rpm and feed rates. The only time you may prefer a HSS cutter is when you need to use a long very narrow cutter because carbide is more brittle, or where you are machining a very rough metal where there is a high shock to the cutter.

RPM of Spindle - The Rpm of the spindle is a very important factor when cutting different materials such as metal. Wood and soft materials can usually be cut at full rpm although plastics may melt if the feed rate is too slow.

Generally the harder the material, the slower the cutting speed in feet per minute of the cutting edge of the cutter. If you try to cut a hard material such as brass running the spindle at full rpm, you will burn and dull the cutting edge of the bit immediately. The larger the diameter of the cutter the slower you must run it because of its larger circumference.

Feed Rate - The feed rate (how fast the xy table moves the spindle) depends on the finish required and how much load you can put on the router or spindle. A slower feed rate will usually give a better finish.

Machining Metals - Metals require much more operator attention than other materials. You must use a coolant and keep chips clear of the cutter. Also the rpm of the cutter should be 10000 or less. Do not use cutters with a diameter greater than 1/4" or chattering may cause xy table and spindle damage.

Set the maximum cut per pass to .05" - 0.1" maximum per pass and set xy cutting speed to less than 10%.

The z plunge speed should also be set to between 1 and 20%.

Routing and Milling Chart

<i>Material Type</i>	<i>Spindle Rpm</i>	<i>Feed rate</i>	<i>Comments</i>
Wood/ Composites	24000	20 -100%	Use spiral Carbide woodcutters, up or down spiral. Adjust feed rate to required finish quality
Plastics	10000-20000	20 - 100%	Use spiral or straight flute wood Router bits. If feed is too slow, plastic will melt. Adjust Rpm to feed rate
Foam board	Med to high	50-100%	Use spiral or straight flute wood router bits. Too slow a feed, and foam will melt. Adjust rpm to feed rate
Aluminum	16000 for 1/4"	2-15 %	Use High-Helix spiral 2 flute carbide milling cutters. Use only 6061 or 7075 Grade Aluminum. (not utility grade 3003 used in sheet metal shops) Use coolant, clear away chips.
Brass	12000-16000	2 - 15%	Use only very sharp spiral carbide milling cutters. Use a machinable or engraving brass. Copper doesn't machine well.
Stainless steel	10000	2 - 10%	Use only very sharp spiral carbide milling cutters. Stainless will wear-harden if the cut is too light or cutter is dull.

Wood router bit Supplier :

Onsrud Cutter Inc.

800 Liberty drive, Libertyville, ILL 60048
(708)-362-5028 (They have special cutters for plastic also)
1-(800)-234-1560

Milling cutters and coolant:

DoAll Industrial Supply

USA - 2817 N, 37th Ave Phoenix AZ. 85009-1341 (602)-269-7231
Canada - 2655 Rue de Miniac, St-Laurent QC H4S1E5 (514) 337-3631

KBC Tools

USA - 6300 18 Mile Rd., Sterling Heights MI. 48314 (800)-521-1740 or (313)-979-0500
Canada - 6200 Kennedy Rd, Unit 1 Mississauga, ON L5T2Z1 (905) 564-6600

Helpful Tips and Tricks for your Larken Router System

- Separate your designs to different layers for best control of complex jobs. You can give the layers names for easy reference. Running complex jobs by layer helps reduce human error and lets you use different tools for different areas.

- The best place for the router to finish cutting out a letter or shape is after a corner if you are climb milling or just before a corner if you are backwards milling. This pushes the loose part away from the cutter instead of making the part self-feed into the cutter.

- Using a reverse spiral (down shear) cutter is a way to prevent parts from coming loose after they have been cut out. The reverse spiral packs the routed path with material and helps to hold it in place. This works well with MDF and wood products, not plastics.

-- You can use the 'Rotate Job' command in the router menu to move jobs to the origin easily. Rotate it 4 times. This positions the work 'cleanly' at 0,0

- For engraving you need a very level work surface to be able to engrave a constant depth without a floating spindle. You can screw or double-sided tape a piece of acrylic or fiberboard down and then machine its entire surface making it perfectly flat. You can use the Box command in the Draw menu and then the tool fill command to create the surface-milling pattern.

- When making a sign where there is raised letters and a background is to be lowered, it is wise to run the whole job once to let the tension in the material adjust, and then run the job again with the tool removing a small amount (0.020" - 0.060") to get a perfect finish.

Troubleshooting Questions and Answers

Q. Parts cut are wrong size, compared to drawing on screen.

- A.**
1. The wrong configuration (*.mac) file is loaded for your machine. Load the correct config file
 2. The parts you are cutting weren't drawn with the tool offset drawn in.

Q. Z Axis seems to cut deeper and deeper?

- A.**
- 1 Setscrews in Z coupler are loose. Turn controller on and try to turn screw with fingers. If you can turn the screw, remove setscrew to locate flat on shaft. Retighten.
 - 2 Speed settings in Z maybe set too high. Settings to check 'Z_raise vel', 'Z_first drop, Z Start vel.
 - 3 Z screw and plastic nut may need cleaning if screw doesn't turn easily between two fingers. Use Teflon spray periodically

Q. Machine not accurate or circles are not cut round?

- A.** - There may be Play or loosening in setscrews between motor and/or screws on gears. Turn controller on (to lock motors), push axis back and forth to find play.

Q. Machine won't respond.

- A.**
1. Parallel port cable not on properly. Check and secure cable. Zip tie to frame of equipment if necessary
 2. StarCNC is set for wrong parallel port. Try ports 1, 2 or 3.
 3. Does LED flash when controller is turned on ? (turn controller off for 30 sec)
 4. Did controller work on this computer previously ?
 - 5 Does the StarCNC spooler display the S when the stop button is pressed ?
 - 6 Are limit switch indicators (XYZ) on? Click the limit switches on the machine and see if the program reads them. Is an axis stuck on a limit switch?
 7. Call Larken automation Tech support