# Larken Router Tutorials

Electrolab Training systems



Importing DXF or HPGL files Running Mastercam (gcode) files Using Corel Draw to create a Sign Using the Digitizing Probe Using the Signcutter Attachment Using the Floating Engraving Spindle Using the Diamond Drag Attachment

# **Cutting a Part on the Larken Camtool**

## Loading a Design File in StarCam

Running a DXF or HPGL, Gcode or EPS file on the Larken is a 2 step process:

- Import , layout and view the file using the **Starcam** layout software.
- Run the file using the Larken **StarCNC** spooler control software

## Step 1

Open the StarCam software

🐮 S	itarCa	m (unt	itled)												
File	Edit	View	Contours	Draw	Toolpath	Route	Help								
	<u>&gt;</u>			ā	3d				Xdist Ydist	0.0	Distanc Angle	e 0.0 0.0	§§1		3
								X -1.	190	Y	8.595	Z 0.00	0 St	ep 0.13	25

> Select File – New , if required to erase a previous drawing

Warning 🛛 🔀							
⚠	Clear Drawing from memory, Are you sure ?						
	<u>Y</u> es <u>N</u> o						

Select File – Import DXF ,HPGL or Gcode depending on your file type.

DXF is an Autocad Drawing Exchage Format this has to be R12 or less

HPGL usually has a file extension .PLT

Gcode is a format for controlling CNC machines. It is output by all CAM programs. There are many variations of Gcode

Import Dxf file			? 🔀
Look <u>i</u> n: 🗀 dxf		▼ ← €	) 💣 🎟 -
AK.DXF AL.DXF AR.DXF AR.DXF ARROWS.DXF ARROWS.DXF BAT.DXF	BATTER.DXF BIKE.DXF BLUEPRNT.DXF BUTTER.DXF CA.DXF CACTUS.DXF	CARTOON.DXF	DINO.DXF DIVIDRS.DXF ELF.DXF FL.DXF FLAKE.DXF FOOTBALL.DXF
			>
File <u>n</u> ame: DIN	0.DXF		<u>O</u> pen
Files of type: DXF	(autoCad)	•	Cancel

Locate your file and select open

號 Z Value selection	- <b>- ×</b>
Z coordinate from file C Use Z value from file • Use Layer Z value	
ОК	

For DXF a selection of Z value coordinate options will appear. Select Use Layer and click OK



- Right click and select zoom half
- Select your entire drawing by clicking and dragging a box around the drawing. The drawing should turn black and have a selection point at the center.



- Click and drag the selection point to move your drawing into the gray machine work area. Place it where you would like your finished part to be referenced to the lower left corner of the work area (Program Reference Zero). Right click and select zoom all.
- Select Contour > Sort Contours . This joins small gaps and re-sequences the drawing for faster cutting.

Sort Contours	×
Sort all layers	
C Only live layer	
🔽 Start at 2nd contour	
🔽 Allow direction swaping	
📕 Join small gaps	
Join threshhold 0.005	
Close Sort Conto	urs

 $\triangleright$ 

- Sort contours Close
- > Select the layer menu (center button at the top)

Laye	ers							×
		Surf Name	Stat	Depth	Tool#	XY Feed	ZF	
	$\checkmark$	7	Live	-0.030	1	0	0	
		Layer2 E	Hot	0.000	1	0	0	
		Layer3 E	Hot	0.000	1	0	0	
		Layer4 E	Hot	0.000	1	0	0	
		Layer5 E	Hot	0.000	1	0	0	
		Layer6 E	Hot	0.000	1	0	0	
		Layer7 E	Hot	0.000	1	0	0	
		. or	11.1	0.000	4	0	0	
Set Z depth (Negative down)								
Change Cle		Clear	Redrav		ose			

- Set the depth that you want your drawing to cut at . Note: All depths must be a negative value to cut downward.
- > Close
- Select Route > Route All

Output Tool Paths		
Total Vectors 797 Object width (X) 1.786'' Object Length (Y) 4.166''	Total Contours 80	
		Close

> Select output

The slider may not go completely to the end. This is normal

- Select Close
- > You are now done with StarCam, and next you will use StarCNC to cut the part.
- > The machine Output toolpath was saved to to a file named 'Temp0.LKS'

# Step 2 Cutting your design on the CNC Router

#### 1) Turning on the Control Box and StarCNC

- Power up the Larken Control Box. The green light on the controller should be flashing (this indicates the controller need to be initialized)
- > Open the Larken StarCNC software (the Spooler)
- 🗱 CNC
- The Controller Not Initialized box should appear

Confirm	ı 🔀
?	Controller not Initialized Click OK to proceed
	<u>Y</u> es <u>A</u> bort

**NOTE**: If the **Controller Not Initialized** window doesn't appear, close StarCNC control software and reopen it.. If the program won't close, use Ctrl-Alt-Delete keys to bring up task manager and end the program.

#### 2) Mount the stock

> Select yes

Mount your stock on the router table. When facing the front of the machine the positive x-axis direction is toward you and the positive y-axis is to the right. Position the stock with the PRZ (program reference zero) at the back left. **Check with the instructor if you are unsure.** 





Wait until controller is initialized and ready appears at the top

#### 3) – Set the PRZ

Using the jog buttons, position your cutter over the PRZ. The center of the cutter should be directly over the corner. This is normally the back left corner of the stock when facing the front of the machine. Jog the cutter down until the cutter just touches the top of the stock.

When jogging the machine always be aware of the direction that the cutter will move **before** clicking the button. Practice using the slow jog function (Toggle the fast and slow mode using the "F" button in the center of the x and y jog buttons)

Set New Z Po	sitior	)	×
Z pos	0.0		
Cancel		Ok	
Z pos Cancel	0.0	Ok	

Click on the Set new Z Position Button		
	StarCNC Controller	
	File Functions Setup H	Help
	Home XY Home Z Spin	idle Start Aux 1 Aux 2
	Ready	-0.857"
	GCL1 GCL2	-1.037"
Set New Z Position	GCL3 GCL4	-1.610"
	GCL5       Z to Top       Z to Zero	Rapid Velocity C 25% C 50% C 75% C 100%
	Set New Z Position	Feed 24
Z pos 0.0	Set New XY Position	FRO 0 🗦
Cancel Ok	Run around Boarder	<b>∝Rpm 120(</b> ‡
	Run Pause	Y+ Z+ X- F X+
≻Ok	EStop	Y- Z-
	Local	Actual Feed 5

≻ Click on Set New XY Position Button.

Set New XY Position							
☑ Xpos ☑ Ypos	0.0	(					
Cancel	Ok						



≻Ok

Note the Position readings for X,Y and Z, they all should read 0.0000 as shown below.



Select the Z to Top Button



## This will move the cutter to a safe height above the work.

Note: If Z axis it at the very top or has just been homed, Overtravel will occur if you tell it to move higher, So be sure to have set the New Z- Zero position to work before clicking this button.

## 4) - START ROUTER AND ADJUST THE SPINDLE SPEED

> To start the spindle click the Spindle Start button on the top bar

> Adjust spindle speed for the material being cut.

Note: RPM can only be adjusted on 220volt Colombo or Elte spindles.

Bosch routers have a manual Dial on the router to vary the RPM.

If the spindle does not start, check the on/off switch and the power cord.



The spindle is plugged in separately from the router itself. If it is unplugged or the spindle switch is off, it will not turn on and tool breakage may occur.

#### Opening the Spooler file ('Temp0.LKS')

This is the Toolpath file Starcam outputed and is always located in the StarCam directory.

Open StarCam file		? ×
File <u>n</u> ame: temp0.lks	Eolders: c:\starcam C:\ Starcam	OK Cancel N <u>etwork</u>
List files of <u>type:</u> StarCam Spooler	Dri <u>v</u> es: == c:	•

Select the temp0.lks file and select Ok The Run button should now be enabled

# Before hitting the RUN button double check

- 1. The correct program is loaded
- 2. The proper stock has been mounted securely for your program
- 3. The correct cutter is installed for your program
- 4. The PRZ has been correctly set
- 5. The spindle has been tested and the speed set
- 6. You are close the ESTOP button in case you need to stop the machine.

#### Select Run

If the Run button is grayed out select File – Open spooler file

You can run a dry run without cutting any material by jogging the z axis to a position above the stock and resetting the z axis ( select set new Z position ). Your program will run above the stock and any potential problems can be seen.

## \*\*\*\* Crash prevention tip \*\*\*\*

Do not confuse the Set new Z position button with the Z to Zero button.

Hitting the Z to Zero button causes the cutting head to immediately move to the zero position. If the cutter is positioned 1 inch above the surface of the stock yet the position indicator reads 2 inches (this can happen after a tool change) the cutter will attempt to move to z zero (1 inch **below** the surface of the stock)

Note: Be sure to reset the Set Z position after a tool change !

## RUNNING A GCODE FILE DIRECTLY WITH THE SPOOLER

If you have a Cam program that outputs Gcode like Mastercam, Visual mill etc, you can bypass using Starcam and directly run your Gcode (.NC) files with the StarCNC spooler.

To set the Spooler to run Gcode files directly you need to set to Gcode\_Mode in the Environment Setup. Then click Save\_Setup and close.

File Locations	
StarCam Spooler files c:\Starcam	Browse
Gcode files c:\ncfiles	Browse
Screen Mode       Machine Type         StarCam Layers       Router         Gcode Interpreter       Plasma         Controller Type       USB         Port1 (888)       Port2 (956)         Port3 (632)	ave Setup Close

# SELECTING THE .NC FILE TO RUN

Select File > Open Gcode file



Open Gcode	file				? 🔀
Look jn: 间	Larken		•	* 🖻	
JDUB F1 C/ JDUB PEN S JDUB.NC JWD CONT JWD LINES JWD OUTS	AR.NC SET.NC OUR.NC AND HOLES.NC IDE.NC	JWD TOOLPATH	H.NC		
File <u>n</u> ame:	*.NC				<u>O</u> pen
Files of <u>t</u> ype:	Gcode Files			•	Cancel

➤ Select the .NC file and click on Open.

Note: Some CAM programs will use the extension .TAP .CNC or .TXT. You will need to change this Cam systems Post to .NC

On the LarkenCNC.com website (under downloads) there are a number of Posts that are compatible with Larken routers.

# Using Corel Draw to create Signs and Tool Paths

#### Setting Page size

You can configure the page size to the size of the Workpiece or Bed of your router. In the main menu go to Layout > Page Setup



Change page type to Custom, and enter the Height and width you want.



#### Creating a Simple Sign

Here we will create a simple sign with a border.

Click on the Text Tool in the **Toolbar** on the left side of the screen. The mouse will change to a blinking text cursor. Click on the screen wher you want the text. You can pre select the **Font** and **Size** at this time or change it later. Type the letters at the cursor.



You can edit the Font later by selecting the text object wit the **Select arrow tool** (top of Tool bar) and then clicking on the Text tool and clicking into the text. You should have a cursor now in your text that you can use to select the letters. Use the cursor keys or a mouse wipe, to select the letters.

Click on the **Font Dropdown** on the upper tool bar, and use the up and down cursor keys to scroll through the fonts Click to select the font.



### Creating a Simple Box boarder

Click on the shape tool. The shapetool lcon has a Flyout menu to allow you to select a number of different shapes. To get the **Fly-out menu** menu hold the mouse clicked down on the small black corner in the lower right Select the Box. With the cursor, draw a box by holding down the mouse and dragging. The release the mouse when done.



To edit the Box and round the corners, use the Node edit tool. It's the Icon just above the Zoom Magnifier in the Tool.

Click in the corner node and drag slightly along the line,, and you should see the corners become rounded. (The object needs to be selected first)



#### Changing View to Wire frame mode to see ToolPaths

When working with tool paths for the cnc router, its better to turnoff the Fill, and just see the outline of the paths.



#### Creating a Tool path using the Contour command

The Contour command in Corel is very powerful. It can create a offset-line to compensate for the radius of the router bit. EG: If you are using a 0.250" diameter router bit, you need to keep the cutter away from the letter by 0.125"



You can select whether you want the tool path on the outside of the object or the inside by checking the box. Also set the offset to the cutter radius. The number of steps sets he number of offset paths it creates. If you check 'To Center' it will create a fill path completely between objects.

## The Combine Command

Note that in he example shown the Apply button is no enabled, indicating that something is not ready. Use the Combine command to prepare the objects for the Contour command.

This command presets contours path direction (CCW=Inside and CW=outside) and order.



After the Combine is used, the Apply should be active



Here is the result of '**Offset to Center**' This has created a Fill tool-path for removing material between letters.



When you select the object after the Contour command it selects all as one object. You can use the Break\_Apart command to separate the paths, and also ungroup them as well.

After they are separated you can set different colors for them so that they are separated then they are exported into up to 8 router Layers when they come into Starcam using the HPGL file format.

#### Saving output to a File

After you have created your Toolpaths, you need to Export the paths in HPGL (or DXF) format to a file.

Use the File > Export command and set the save as type to HPGL format. This is a Pen Plotter format used by Hewlet Packard .

The HPGL filename should have a .PLT on the end.

3. Cont BRAW 11 - [Cranbh1]  3. Cont BRAW 11 - [Cranbh1]  4. Ele Exit yew Layout Annape Effects Ethnops [set Topis Window Help  5. Cont Braw 10, 10, 10, 10, 10, 10, 10, 10, 10, 10,	- ×
Image: Section of the section of th	
Start Start Start CorebRAW 11 - [Gra Core] Starcam	

HPGL allows up to 8 different colors to be exported separated by a pen # command in the file. This lets Starcam set them as layers which you can define different depths for. Its best to use major solid colors in your drawing to define these colors. Experiment with colors and exporting to see the results when they are displayed in Starcam.



Use the HPGL import command in StarCam to load the file. Be sure to Sort it after importing.

# Using the 3D Digitizing option With the Larken Camtool

# Step 1 – Attach the digitizing probe

- > Remove the 4 cap screws securing the spindle plate
- Replace with the probe (2 screws are usually sufficient). Secure it in a position that will allow the tip of the probe to reach the bottom level of your model.



> Plug the probe into the connector beside the router ac receptacle

# Step 2 – secure the item to be scanned and setup the PRZ

- > 2-sided tape works well for models with flat bases.
- Power up the controller and open the Larken Router Control software. Notice that there is a W in the red display at the bottom. This indicates that the probe is connected.

StarCNC Controller		
File Functions Setup	Help	
Home XY Home Z Sp	oindle Start Coolant Off	
Ready	0.000"	
GCL1		
GCL2	0.000"	
GCL3	0.000"	
GCL4	-0.000	
GCL5	Rapid Velocity	
Z to Top Z to Zero	C 25% C 50%	
Set New Z Position		
Set New XY Position	Feed 40	
	- FRO 0 🖵	
Move to XY position	Rpm 22100 ≑	
Run around Boarder		
Run Pause	Y+ Z+	
15.5411	X- F X+	
EStop	Y- Z-	
Local	Actual Feed 1	

Jog the tip of the probe to your start position (PRZ). It should be positioned so that the positive quadrant (x and y axis) will enclose your entire model. Set the z axis to the lowest level on your model that you wish to scan.





> Set the x, y, and z axis positions to zero.

Set New Z Position	Set New XY Position
Z pos 0.0	IV Xpos 0.0 IV Ypos 0.0
Cancel Ok	Cancel

# Step 3 – Scan the model and save the DXF file

> Select the 3D Probe menu from the Functions drop down menu

💏 3D Digitizer Probe	
X Size 3.250 Y Size 3.250	0.050
XY Probe speed	20 💐
Z Probe Speed	20 🗶
Save Probe File	
Save DXF	Pause Start Probe
Save Goode	Close

> Set the X and Y Size to enclose your entire model.

Tip – Once the X and Y zero position has been set the machine can be jogged to a position that will enclose the model. Make note of the values for x and y and use these for the X and Y Size settings.

- Set the sample pitch. A lower number will create a more accurate model but will require more time to scan.
- Set the XY and Z probe speed to 20. This speed will usually result in a good error free scan.
- Select start probe
- > When the scan is complete select Save DXF.

Save DXF fil	e			?×
Save in: 合	} My Documents	- 🗈	<u></u>	* 🔳
My eBook My Music My Picture horse.DXF new.DXF	s 🛋 test.DXF :s :XF			
File <u>n</u> ame:	newmodeLDXF			<u>S</u> ave
Save as <u>type</u> :	Drawing Exchange Forma	t	•	Cancel

Take note of where the file is being saved

# **Step 4 – Use Surfer 3D to create the toolpath files**

- > Open Surfer 3D
- Select Files Import 3D surfaces (DXF or STL)

Import 3d DX	(F		? 🗙
Look jn: [ 🔒	My Documents	💌 🗢 🖻 📑	
Achiever8	My Pictures udio 🥮 My Videos 🍋 ScreenCam 🗃 newmodel.DXF		
File <u>n</u> ame:	newmodel.DXF		<u>O</u> pen
Files of <u>type</u> :	Dxf Import (*.dxf)	•	Cancel

- > Open your file
- > If you want to adjust the finished size of your model select edit resize all

Resize Model		
	X Size Y Size Z Size	3.050 3.050 0.565
	✓ Proportion ✓ On Center	al
	Cancel	Ok

- Select Auto adjust origin from the edit menu or use the toolbar button. This will adjust the model so that it is below z level zero (top of the stock)
- Select Set border from the edit menu or use the toolbar button.

Define Blank size	×
	Border size 0.320
	X blank size 3.690
	Y blank size 3.690
	Center model on Work
X model size 3.050 Y model size 3.050	Cancel Ok

Warning – Make sure that your border is wide enough. It must be a minimum of the diameter of the cutter you will use to rough plus the stock you plan to leave for finishing. If it's too narrow the finish pass may cut a full depth cut because there was no roughing done to the border.

Define the Z cut Level. This is selected from the toolbar buttons. The Single level job will work for most models. If your model lower z level is larger than the longest tool you have the model can be sliced and cut out of separate pieces. Once they have all been cut they can be glued together to create a finished piece.

Z Level Setup	X
Z Upper Level 0.000	
Z Lower Level -0.565	
Single level job	
Single Pass	1
Lower Z level -0.565	
<ul> <li>Slice to different levels</li> <li>Level Slicing</li> <li>Number of Slices</li> <li>Current Slice to cut</li> </ul>	
Slice size 0.565	
Uncovered Area C Set as Upper level Set as Lower Level	
Cancel	

Select the tool list button. Select the tool you wish to use to rough the model out. Set the maxcut / pass setting to the roughing cut that you wish to use.

Toolmenu		
125 flat 2 - 45d VEE 3 - 3/16 Ball 4 - 1/8 Ball nose 5 - 1/4 Ball 6 - 1/16 flat 7 - Engraver 8 - Long .25 9 - LONG .5 10 - 3/16 Ball nose	I/4 Ball Diameter 0.250 Maxcut / pass 0.250 Conic Length	Set your roughing step here
Cutter Type C Flat bottom Ball nose C Conical C Flat Conical New Tool Rename	Conic Flat Tool Rpm 5000 FeedRate 50 Update Tool Save Tools Close	

- Select the toolpath button
- You must set the base resolution first. This setting will limit the stepover of the tool during the finish toolpath. Using a smaller setting will create a smoother finish with less sanding but will take more time to cut.

Base Sample Resolution 🛛 🔀		
Select the Job smallest tool in	Resolution. This must be smaller than the the Job	
<ul> <li>Inch</li> <li>Metric</li> </ul>	0.01563	

Select the Set filename button. Give it a name that will indicate that it is a roughing toolpath.

Note: Make sure the filename is followed by a .LKF. There is a small programming error in surfer 3D that prevents the extension from being added automatically. This should be corrected in later versions.

Save Tool Path to File	
Save jn: 🕒 My Documents 💽 🖛 🗈 💣 🎫	
Cambridge Achiever8 A My Pictures	vour filename
Denford	
My Albums	
My eBooks	
File <u>n</u> ame: newmodel rough.LKF <u>S</u> ave	
Save as type: Larken L-CAM Job *.LKF	

- Select save
- Select the tool to use, the stepover, and the amount of stock to leave. For the roughing cycle the stepover can be set to about 75% of the tool diameter. This will cut down on the machining time.
- Select the Roughing Zigzag (X axis)

Single Pass Toolpath Se	tup		
Project Base Resolution	0.01563	Adjusted (0.01563)	Change Resolution
Tool Name 5 - 1/4 Ball Step-Over amount 0.156" Rough to leave 0.063"	▼ ▲ ▼	⊂Toopath	n Type ughing Zigzag (Xaxis) ughing Zigzag (Yaxis) ish Zigzag (Xaxis) ish Zigzag (Yaxis) ral Finish CW
Set Filename	s and Setting	gs\ Cancel	Start Path Create

- > Select Start Path Create. This will create the roughing toolpath file
- Select the toopath button
- Select the Set Filename button. Give it a name that will indicate that it is a finish toolpath.

Save Tool Path to File 🛛 🔹 💽 🔀				
Save in: 📋	My Documents 💽 🔶 📸 📰 -			
Achiever8 Camtasia St Denford My Albums My eBooks My Music	Iudio Importante My Videos Importante ScreenCam Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importante Importa			
File <u>n</u> ame:	newmodel finish.LKF Save			
Save as <u>t</u> ype:	Larken L-CAM Job *.LKF			

- Select Save
- Select the Spiral Finish CW
- Select the tool to use and the stepover. A smaller tool will create the most detail but will take longer to machine.

👍 Single Pass Toolpath Set	up		
Project Base Resolution	0.01563	Adjusted (0.01563)	Change Resolution
Tool Name 4 - 1/8 Ball nos Step-Over amount 0.016'' Rough to leave 0.063''		Toopath 1 C Roug C Roug C Finist C Finist € Spira	Type Jhing Zigzag (X axis) Jhing Zigzag (Y axis) n Zigzag (X axis) n Zigzag (Y axis) I Finish CW
Set Filename C:\Documents	and Setting	gs\ Cancel	Start Path Create

> Select Start Path Create. This will create the finish toolpath file.

# Step 5 – Cut the Roughing Toolpath

- Open Starcam
- Select File New
- Select File Open job
- Select your roughing toolpath



- > Open
- Right click and select zoom half



Tip - moving your mouse to the maximum extents of the toolpath will display the x and y coordinates at the bottom of the screen. Use these to do a quick check of the stock size needed and to help in clamp placement.

Select Route – Route All (or select the far right toolbar button)



- Select Output
- > Open the Larken Router Control Software
- > Use the normal procedure to run the file

# Step 6 – Cut the Finish Toolpath

> Repeat step 5 using the finishing toolpath file.

# Using the Vinyl Sign Attachment with the Larken Router

**Step 1** – Remove the Spindle head complete with bracket and attach the vinyl sign attachment

- Unplug the spindle and remove the 4 cap screws securing the spindle bracket
- Attach the vinyl attachment to the 2 holes on the left using 2 of the cap screws from the spindle bracket



**Step 2** – Mount a suitable platen to the worksurface of your machine

- A sheet of ¼ inch acrylic cut to the size of the cutting area on your machine works well.
- > Any consistently flat material will make a great platen. MDF, Arborite, etc



## **Step 3** – Create an HPGL (plotter file) to run on the Larken

- This tutorial shows the procedures using Corel Draw. Any vector based program that exports a plotter file will work.
- Setup your page size to be the size of the material you will cut. Vinyl is available in 12 and 24 inch rolls
- Select Layout Page setup

Options	Ĩ	? 🗙
Workspace     Jocument     General     J-Page     Size     Layout     Label	Size       • Normal Paper     Labels       • Portrait     • Landscape       Paper:     Custom	
Background Grids and Guidelines Rulers Styles Save 	Width:       20.0       inches         Height:       12.0       inches         Resolution:       300       Image: Compare the second secon	]
	Add Page Frame OK Cancel Hel	p

- > The maximum page width is determined by the size of your machine
- > Select the text tool and and click in the graphics area
- Enter your text
- Select the selection tool and adjust the size of the letters and choose a font
- Position your letters on the page. The letter position determines where they are cut relative to the 0,0 position (the start position when you setup the machine). Allow about a ¼ inch of space at the sides to tape the vinyl down.



To make the weeding process easier create a rectangle around your design using the rectangle tool



- Select File Export
- Export a plotter file (\*.plt)

Export			? 🔀
Save in: ն	Desktop	▼ ← 🗈 📸 <b>.</b>	E <u>x</u> port
Cnc files			Cancel
Graphic1.p	lt It		
File <u>n</u> ame:	LarkenCNC		
Save as type:	HPGL Plotter File (PLT)	Compression typ	pe:
		Uncompressed	<b>–</b>
Not <u>e</u> s:			
		Suppress fil	ter dialog

**Step 4** – Cut and mount a piece of vinyl that matches the size of the page setup in your drawing program.

- > Position the vinyl so the width setting in the page setup is along the x-axis
- > Secure the vinyl with masking tape on all four sides



Step 5 – Import file into Starcam and create spooler file

- Open the Starcam software
- Select file import hpgl



- > Open
- Right click in the drawing area and select zoom all. Your design should be shown in the lower left corner of the machine work area (grey section)



Select contours – Sort contours

Sort Contours	×
<ul> <li>Sort all layers</li> <li>Only live layer</li> </ul>	
<ul> <li>Start at 2nd contour</li> <li>Allow direction swaping</li> <li>Join small gaps</li> </ul>	
Join threshhold 0.005	
Close Sort Contour:	

- Sort Contours
- > Close
- Select Route Route all



> Output

**Step 6** – Adjust the cutter depth set the PRZ (program reference zero), and run the program

Adjust the cutter head so the cutter tip is just protruding from the end of the holder. Adjust it by turning the end cap on the holder in or out. The adjustment is correct when the vinyl is cut completely through but not through the backing underneath. A few test cuts may be required to get it correct.



- > Open the Larken router control software
- Jog the tip of the cutter to the lower left corner of the vinyl. It should be far enough in from the corner to avoid hitting the hold down tape.



- Set the xy position to zero
- Jog the cutter head down until the spring loaded holder has compressed about ¼ inch.



- Set the z positon to zero
- Select File open spooler file

Open StarCa	m file			? 🗙
Look in: 🗀	Starcam	•	- 🗈	-111 *
temp0.LKS				
File <u>n</u> ame:	jtemp0.LKS			<u>O</u> pen
Files of type:	StarCam Spooler		•	Cancel

- Select the temp0.lks file and select open
- > Before selecting the run button check:
  - 1. The correct file has been loaded and converted (a quick check is to look at the file that is loaded in starcam. If you leave the output window open, that confirms the conversion has been done)
  - 2. Enough vinyl has been taped down for your design
  - 3. The program reference zero (start point ) has been correctly set
- > Select Run

**Step 7** – Weed the image and transfer the finished sign

- > Remove the vinyl and cut the sign from the material
- > Weed out the unwanted parts of the design.



Apply the transfer tape and smooth over the design. Autobody spreaders make great squeegees.



> Lift the image from the backing



Place the image in the desired location and squeegee it down



> Remove the transfer tape and admire the finished product



# Using the Floating Engraving Spindle Attachment with the Larken Router

**Step 1** – Remove the Spindle head complete with bracket and attach the floating engraving spindle

- Unplug the spindle and remove the 4 cap screws securing the spindle bracket
- Attach the floating spindle to the 4 holes using the 4 cap screws from the spindle bracket



Step 2 – Mount the sign stock to the worksurface of your machine

- The stock can be held with 2 sided tape to a suitable platen to prevent damage to the machine table
- Consider designing a hold down fixture if you are cutting multiple signs out of the same sized stock

## **Step 3** – Create an HPGL (plotter file) to run on the Larken

- This tutorial shows the procedures using Corel Draw. Any vector based program that exports a plotter file will work.
- Setup your page size to be the size of your Larken machine. Select Layout

   Page setup
- This will allow you to create as many signs as will fit in the work area of your machine

Options		? 🗙
Workspace     Document     General     Page     Size     Layout     Label     Background     Grids and Guidelines     Rulers     Styles     Save     Publish To Internet     Global	Size         • Normal Paper       Labels         • Portrait       Landscape         Paper:       Custom         Width:       24.0         inches       •         Height:       24.0         inches       •         Set From Printer       Save Custom Page         Add Page Frame       •	
	OK Cancel H	elp

- > Use the rectangle tool to create a rectangle of any size
- Edit the height and width of the rectangle to match the size of the sign material that you are using
- Select the text tool and and click in the newly created rectangle
- Enter your text
- Select the selection tool and adjust the size of the letters and choose a font
- Position your letters in the rectangle. Since the rectangle matches the size of your finished sign, create them as you want your sign to look.



Larken

- Select File Export
- Export a plotter file (\*.plt)

Export				? 🔀
Savejn: 📄	Cnc Files			E <u>x</u> port
				Cancel
File <u>n</u> ame:	larken.plt			
Save as type:	HPGL Plotter File (PLT)	•	Compression tur	1P'
			Uncompressed	
Not <u>e</u> s:				
			Suppress fil	ter <u>d</u> ialog

**Step 4** – Import file into Starcam and create spooler file

- Open the Starcam software
- Select file import hpgl

Import HPGL file	? 🛛
Look in: 🔁 CNC Files	- <b></b>
■ larken.plt	
File <u>n</u> ame: <mark>*.plt</mark>	<u>O</u> pen
Files of type: HPGL plotter file	Cancel

- > Open
- Right click in the drawing area and select zoom all. Your design should be shown in the lower left corner of the machine work area (grey section)



Select Layout – Sort contours

<ul> <li>Sort all layers</li> <li>Only live layer</li> <li>✓ Start at 2nd contour</li> <li>✓ Allow direction swaping</li> <li>✓ Join small gaps</li> </ul>	
<ul> <li>✓ Start at 2nd contour</li> <li>✓ Allow direction swaping</li> <li>✓ Join small gaps</li> </ul>	
Join threshhold 0.005	
Close Sort Contours	

- Sort Contours
- > Close
- Window the text only to select it. This will prevent the machine from cutting the reference rectangle



- Select Route Route selected
- > Output



**Step 6** – Adjust the cutter depth, set the PRZ (program reference zero), and run the program

Set the cutter to a zero depth. Turn the adjusting head until the tip of the cutter is flush with the end of the holder. The indicator should read zero. If it doesn't you may have to adjust the cutter in the collett. (assembly pics at the end of tutorial)

Adjust the cutter to the desired depth. Each rotation of the dial is .025". Experiment with some scrap pieces to find the best depth for the material being used. A good start point is about .015"

- > Open the Larken Router Control Software
- > Jog the cutter head to the lower left corner of the sign stock (normal PRZ position)
- > Jog the cutter head down until the spring loaded holder has compressed about 1/4 inch.

- Set the z positon to zero
- Set the XY position to zero
  Select File open spooler file

Open StarCa	m file	? 🛛
Look in: 🔎	starcam	- 🖬 🍋 🖃
temp0.LKS		
File <u>n</u> ame:	*.LKS	<u>O</u> pen
Files of <u>t</u> ype:	StarCam Spooler	✓ Cancel

- > Select the temp0.lks file and select open
- > Before selecting the run button check:
  - 4. The correct file has been loaded and converted (a quick check is to look at the file that is loaded in starcam. If you leave the output window open, that confirms the conversion has been done)
  - 5. The cutter depth has been set
  - 6. The correct size stock has been loaded
  - 7. The program reference zero (start point ) has been correctly set
  - 8. The spindle has been tested (turn it on and off with the spindle start button)
- Select Run

Assembly pictures

# Using the Diamond Drag Attachment with the Larken Router

The diamond drag attachment can be used to create professional looking trophy plaques. It's also great for engraving on painted metal. A piece of sheet metal painted flat black has been used for this tutorial

**Step 1** – Remove the Spindle head complete with bracket and attach the Diamond Drag

- Unplug the spindle and remove the 4 cap screws securing the spindle bracket
- Attach the floating spindle to the 2 holes on the left of the spindle mount using 2 cap screws from the spindle bracket





**Step 2** – Mount the material to be engraved to the worksurface of your machine

The stock can be held with 2 sided tape to a suitable platen to prevent damage to the machine table

Step 3 – Create an HPGL (plotter file) to run on the Larken

- This tutorial shows the procedures using Corel Draw. Any vector based program that exports a plotter file will work.
- Setup your page size to be the size of your Larken machine. Select Layout

   Page setup

This will allow you to create as many signs as will fit in the work area of your machine

Options		? 🗙
Workspace     Document     General     Page     Size     Layout     Label     Background     Grids and Guidelines     Rulers     Styles     Save     Publish To Internet     Global	Size         ● Normal Pape       Labels         ● Portrait       Landscape         Pape:       Custom         Width:       24.0         + gight:       24.0         + gight:       24.0         * inches       •         Resolution:       300         Set From Printer       Save <u>C</u> ustom Page	
	OK Cancel	<u>H</u> elp

- > Use the rectangle tool to create a rectangle of any size
- Edit the height and width of the rectangle to match the size of the material that you are using
- Select the text tool and and click in the newly created rectangle
- Enter your text
- Select the selection tool and adjust the size of the letters and choose a font
- Position your letters in the rectangle. Since the rectangle matches the size of your finished sign, create them as you want your sign to look.





- Select File Export
- Export a plotter file (\*.plt)

Export				? 🔀
Savejn: 🗀	CNC Files	- 🗢 🔁 (	*	E <u>x</u> port
🔟 larken.plt				Cancel
File <u>n</u> ame:	nameplate			
Save as <u>t</u> ype:	HPGL Plotter File (PLT)	•	Compression typ	pe:
			Uncompressed	<b>_</b>
Not <u>e</u> s:				
			Suppress fill	ter <u>d</u> ialog

Step 4 – Import file into Starcam and create spooler file

- > Open the Starcam software
- Select file import hpgl

Look in: CNC Files 💽 🗭 🖻 📸 📰 -	Import HPGI	. file	?
I arken.plt I nameplate.plt	Look in: 🗀	CNC Files	
	iarken.plt	plt	
File name: Den	File <u>n</u> ame:	<mark>j".pt</mark>	<u>O</u> pen

- > Open
- Right click in the drawing area and select zoom all. Your design should be shown in the lower left corner of the machine work area (grey section)



Select Layout – Sort contours



- Sort Contours
- > Close
- Window the text only to select it. This will prevent the machine from cutting the reference rectangle



> Output

Output Tool Paths		×
Total Vectors 1255 1 Object width (X) 2.000'' Object Length (Y) 6.000''	Total Contours 9 Z high (Z) 0.000'' Z low (Y) 0.000''	
Lines of Code =1255	Output Done	

**Step 5** – Set the PRZ (program reference zero), adjust the drag pressure and run the program

- > Open the Larken Control Software
- Jog the tip of the diamond drag to the lower left corner of the material to be engraved

- Set the XY position to 0,0
- > Jog the z-axis down until the spring compresses about .25" (different materials may require more or less pressure)

- Set the z position to 0
  Select Z to top. The tip should lift above the material. If it does not readjust the z zero position for less pressure or the ' tool above work ' can be adjusted (setup, configure machine). Set it to a higher value. Do not select 'save config'. Select OK

🏪 Larken Moti	ion Control setup					
File	<u></u>					> Solaat
Babid and Scale	IIT CAM24.MAC	Z axis   Spindle	Limit Switches LA	Ho Avia ]		Select File –
Z S Z F Z 1 Pau Tool	Start Vel Raise Speed Ist Drop speed se on tool drop I above Work	10 60 50 50 0.350	Limit Switches 4	izontal Z o	ption	open spooler file
Comment Reset Pic	Cam24 with 1/4 st	ep drives .3: OK	3pitchXY and . Save Config	2Pitch Z X Canc	el	
Open StarCa	am file				?	
Look jn: 🔀	starcam		•	- 🗈 🖻	* 🎟 🕇	
temp0.LKS	5					
File <u>n</u> ame:	*.LKS				<u>O</u> pen	
Files of type:	StarCam Spoole	1		•	Cancel	

> Select the temp0.lks file and select open

- > Before selecting the run button check:
  - 9. The correct file has been loaded and converted (a quick check is to look at the file that is loaded in starcam. If you leave the output window open, that confirms the conversion has been done)
  - 10. The drag pressure has been set
  - 11. The correct size stock has been loaded
  - 12. The program reference zero (start point ) has been correctly set
- Select Run

# Appendix Installing and configuring Starcam and the Larken Router Control Software

## Step 1 – Download and extract the latest Larken Software

- Download the Starcam and StarCNC files from here: <u>http://www.larkencnc.com/dloads/index.shtml</u>
- Create 3 new folders on the Local Disk (C:) called Starcam, Starcnc, and Starcncgcode
- Extract the files from starcam???.zip to the Starcam folder and the files from st-cnc???.zip to the Starcnc and the Starcncgcode folder.

## Step 2 – Create shortcuts

- Create shortcuts to the starcam1.exe file in the Starcam folder and the Ispooler.exe file in the Starcnc and Starcncgcode folder. Place these icons where the instructors and students will have access.
- Rename the Starcam1.exe shortcut to Starcam and the other 2 to Larken Router Control and Larken Router Control for Gcode

The Router Control software needs to be configured in 2 different ways depending on the procedure being used. Rather than have the students adjusting the configuration it is sometimes easier to have 2 separate installs (this is an optional step)

*If there are problems running the software more file rights may be required for these 3 folders.* 

## Step 3 – Configure the Router Control Software

The Larken Router must be connected to the parallel port and be turned on to perform the next step

- > Open the Larken Router control software
- Select Setup Configure machine

- Check that the correct MAC file is selected for your machine (if not select file – open config and select the correct one, then Set as Default)
- Change the Fast Jog speed to 55 under the Rapid and Scale tab
- Select the Save
   Config button

File Set as Default C:\starcnc\CAM24.M	MAC
Rapid and Scale Contouring Settings Z avis	Spindle   Limit Switches   4th Axis   Auxillary 1/2
Rapid Settings     Rapid Speed Range     1     Colspan="2">T     Rapid Vel Max     70     Rapid Vel Start     10     Rapid Accel     245	Calibration Factors X axis scale 2.40000 Y axis scale 2.40000 Z axis scale 4.00000
Jog speed Slow 3 Fast 55	<ul> <li>Imperial (0.001")</li> <li>Metric (0.01 mm)</li> </ul>
Comment Cam24 with 1/4 step dri Reset Pic	save Config

- Repeat for the Larken Router Control for Gcode installation
- Select Setup Environment Setup
- Set the Screen Mode to Gcode Interpreter
- If the Starcam software was installed in a different location Change the location of the Spooler Files
- Select Save Setup

	🐍 Environment Setup		
p ode	StarCam Spoolerfiles Gcode files	File Losations c:\starcam C:\\show0\Desktop\cnc fi	Browse Browse
eter	Screen Mode StarCam Lavers Gcode Interpreter		Save Setup Close
n on es	Controller Type	Printer port Port1 (888) Port2 (956) Port3 (632)	

Check the Environment setup in the Larken Router Control installation. It should be set to Starcam Layers.

*These are just suggested installation configurations. Please call me if you have problems or questions. Brian Steers 800-792-6933 ext 312 brian@electrolab.ca*