

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

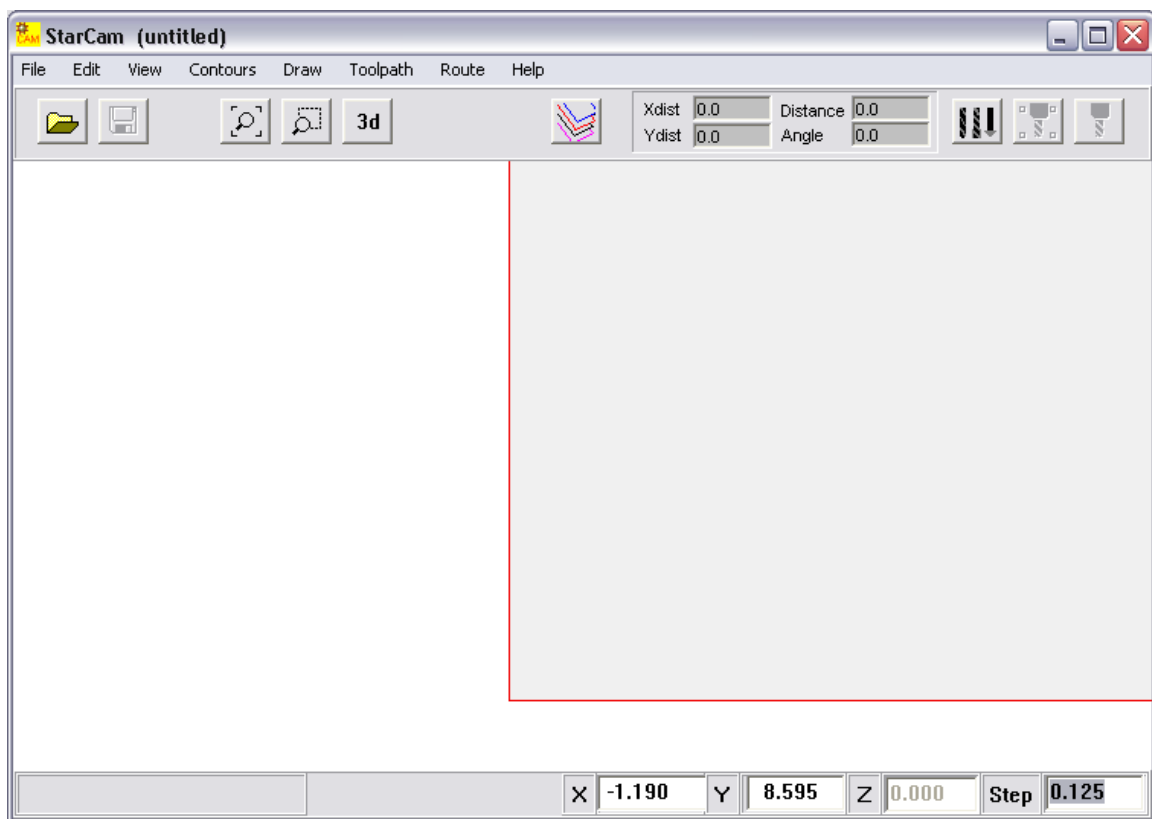
Loading a Design File in StarCam

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

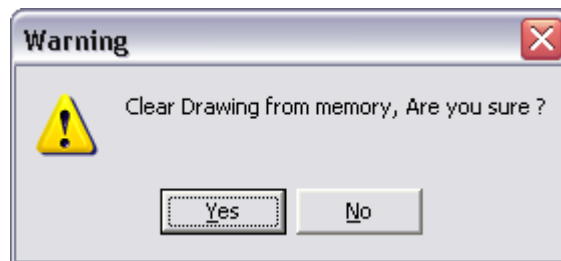
- Import ,layout and view the file using the **Starcam** layout software.
- Run the file using the Larken **StarCNC** spooler control software

Step 1

- Open the StarCam software



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

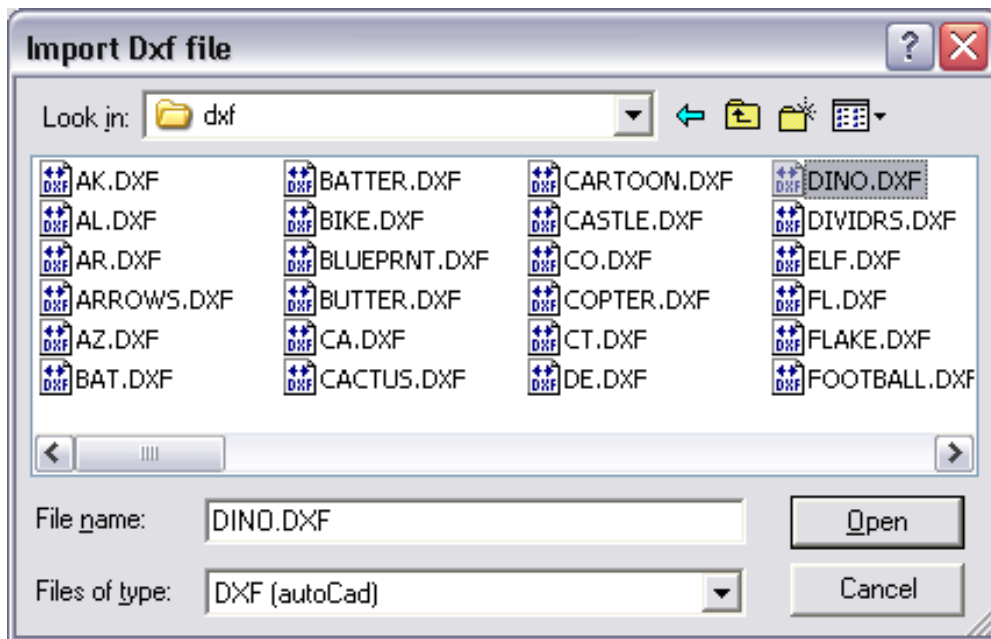


- Select File – Import DXF ,HPGL or Gcode depending on your file type.

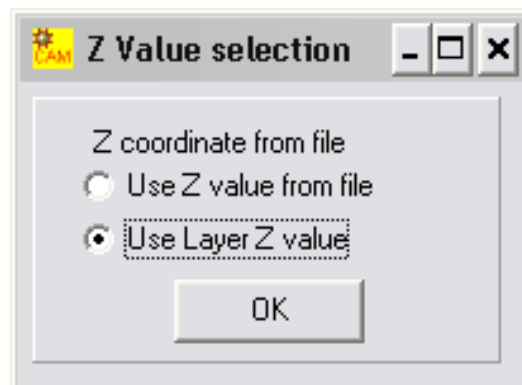
DXF is an Autocad **D**rawing **E**xchange **F**ormat this has to be R12 or less

HPGL usually has a file extension .PLT

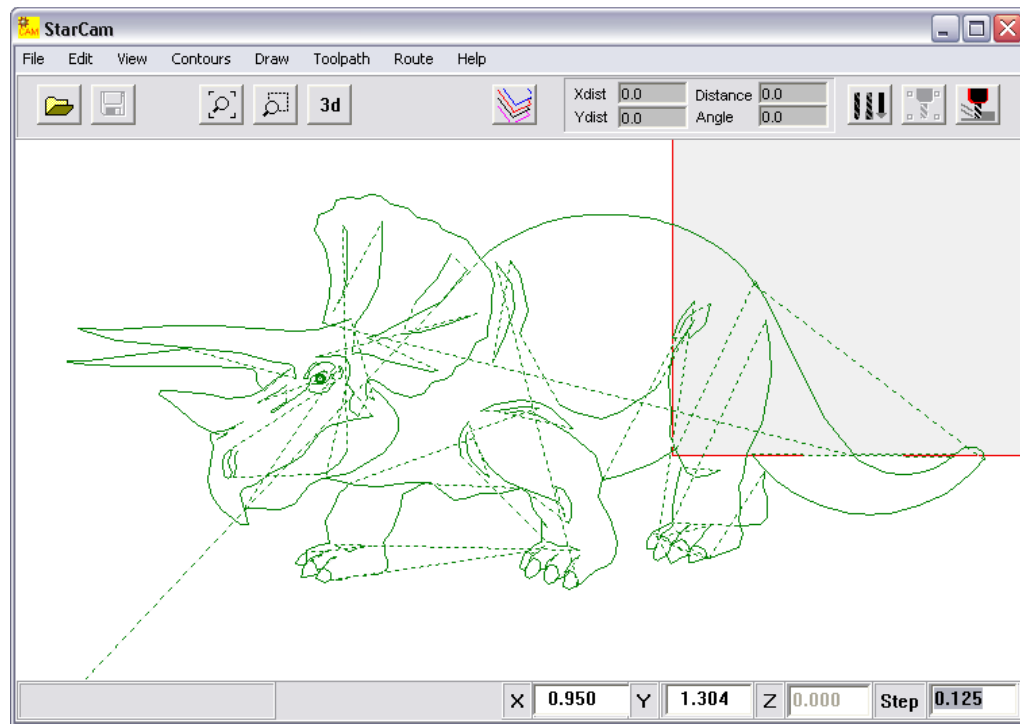
Gcode is a format for controlling CNC machines. It is output by all CAM programs. There are many variations of Gcode



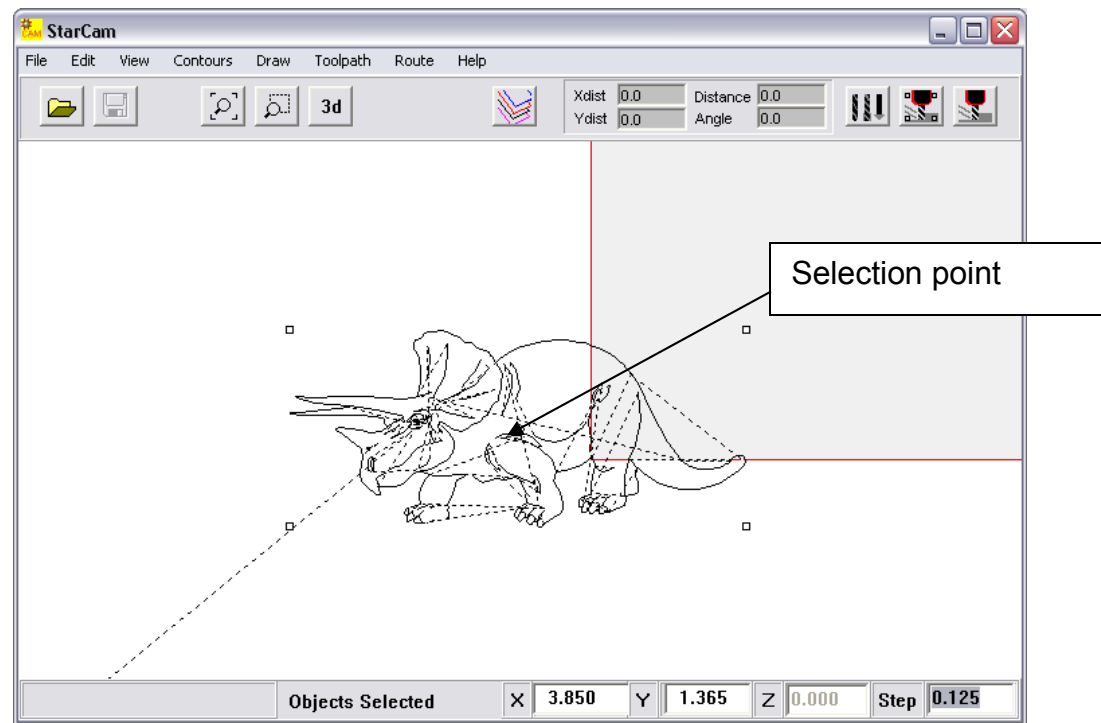
- Locate your file and select open



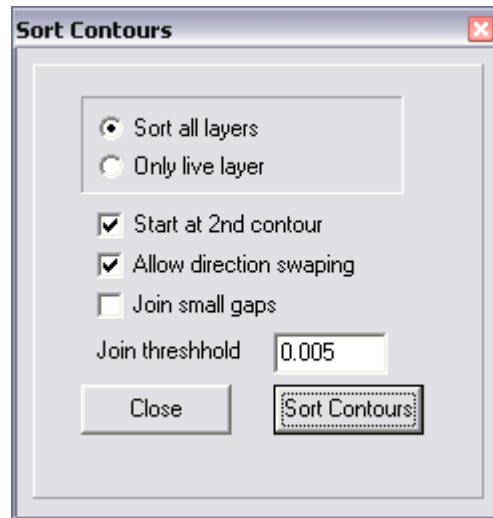
- For DXF a selection of Z value coordinate options will appear. Select **Use Layer** and click OK



- 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.



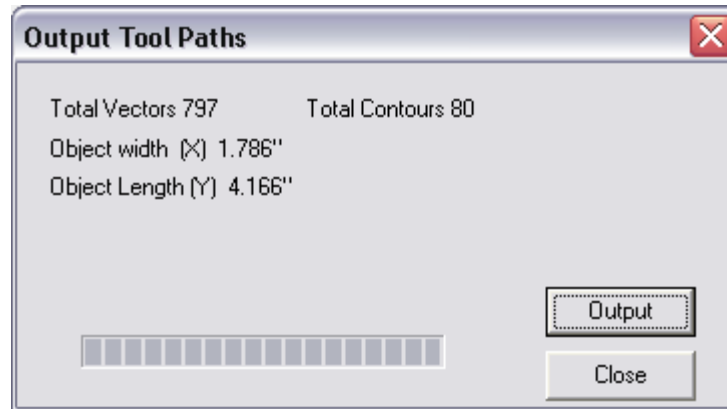
- Click and drag the selection point to move your drawing into the gray machine work area. Place it where you would like your finished part to be referenced to the lower left corner of the work area (Program Reference Zero). Right click and select zoom all.
- Select Contour > Sort Contours . This joins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



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



- 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



- 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'**

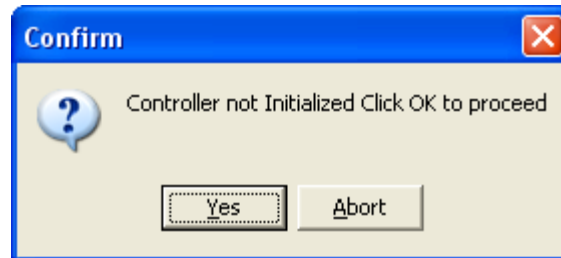
Step 2 Cutting your design on the CNC Router

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)
- The Controller Not Initialized box should appear



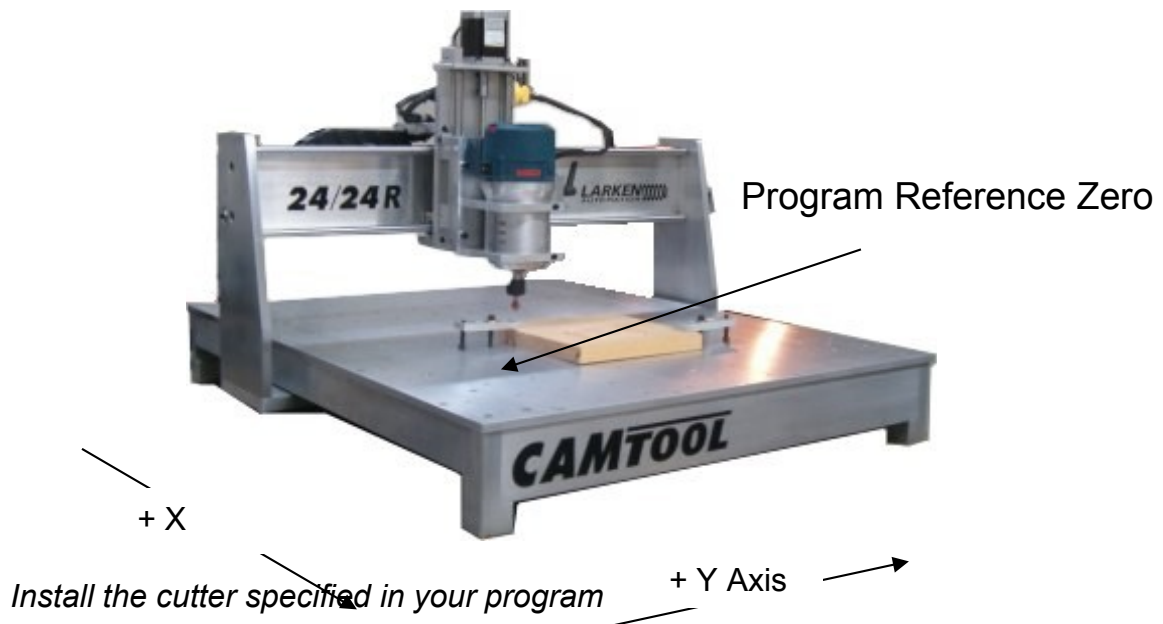
- Select yes



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

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)*

➤ Click on the Set new Z Position Button



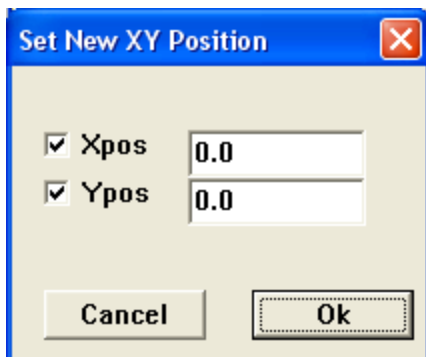
A dialog box titled "Set New Z Position" with a close button (X) in the top right corner. It contains a label "Z pos" followed by a text input field containing the value "0.0". At the bottom, there are two buttons: "Cancel" and "Ok".

➤ Ok



The main interface of the StarCNC Controller software. It features a menu bar (File, Functions, Setup, Help) and a toolbar with buttons for Home XY, Home Z, Spindle Start, Aux 1, and Aux 2. The status area shows "Ready" and various position readouts: GCL1 (-0.857"), GCL2 (-1.037"), GCL3 (-1.610"), GCL4, and GCL5. There are buttons for "Z to Top", "Z to Zero", "Set New Z Position" (circled in red), "Set New XY Position", "Move to XY position", "Run around Boarder", "Run", "Pause", and a large red "EStop" button. On the right, there are controls for "Rapid Velocity" (25%, 50%, 75%, 100%), "Feed 24", "FRO 0", and "Rpm 1200". A status bar at the bottom shows "Local" and "Actual Feed 5".

➤ Click on Set New XY Position Button.



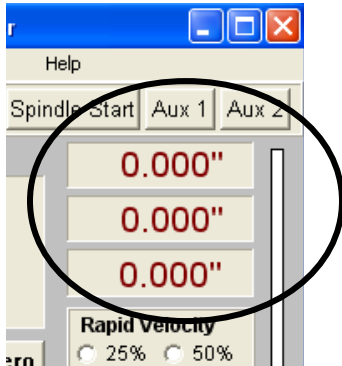
A dialog box titled "Set New XY Position" with a close button (X) in the top right corner. It contains two checked checkboxes: "Xpos" and "Ypos", each followed by a text input field containing the value "0.0". At the bottom, there are two buttons: "Cancel" and "Ok".

➤ Ok



The main interface of the StarCNC Controller software, similar to the previous one. The status area shows "Ready" and position readouts: GCL1 (-0.857"), GCL2 (-1.037"), GCL3 (0.000"), GCL4, and GCL5. The "Set New Z Position" button is circled in red. The "Set New XY Position" button is also circled in red. The "Move to XY position" button is circled in red. The "Run" button is circled in red. The "EStop" button is circled in red. The status bar at the bottom shows "Local" and "Actual Feed 5".

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 is 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.

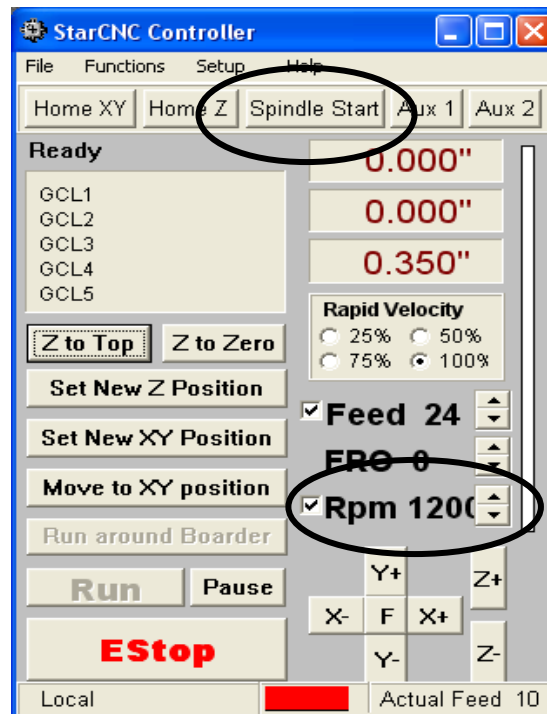
4) – START ROUTER AND ADJUST THE SPINDLE SPEED

- 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.

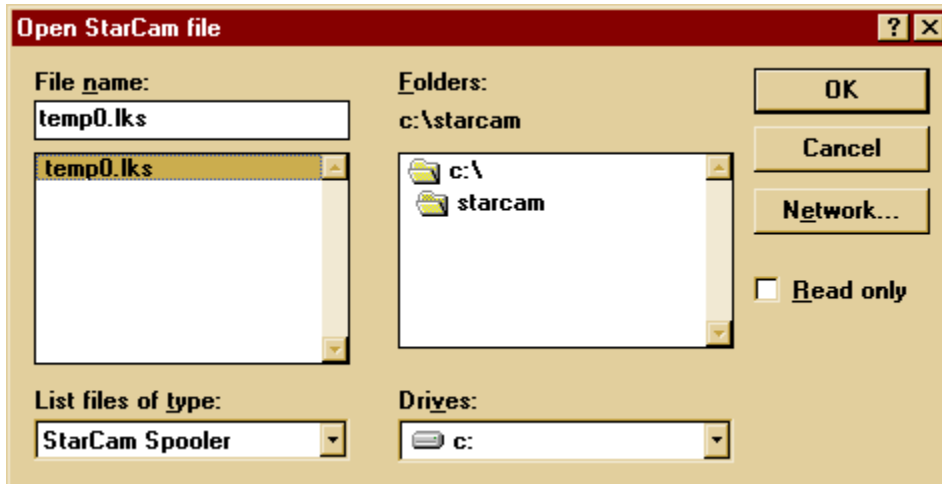
If the spindle does not start, check the on/off switch and the power cord.



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.



- 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.*

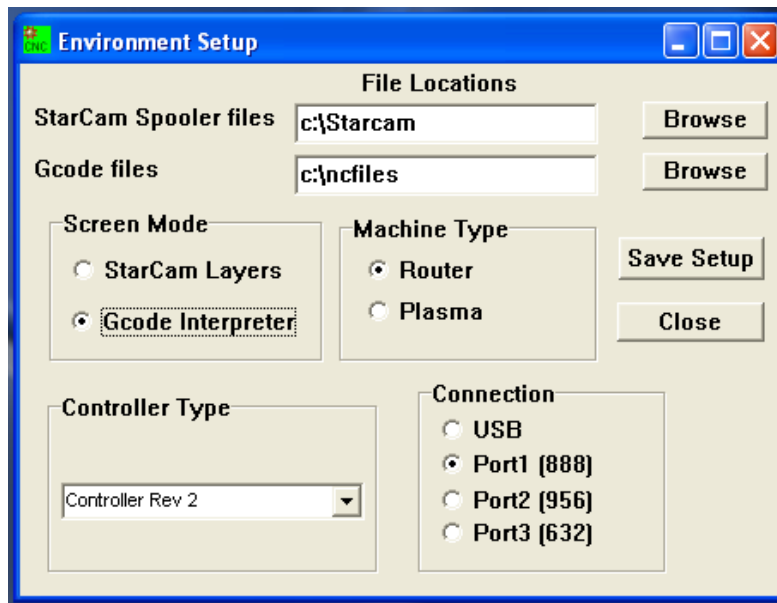
*Hitting the Z to Zero button causes the cutting head to immediately move to the zero position. If the cutter is positioned 1 inch above the surface of the stock yet the position indicator reads 2 inches (this can happen after a tool change) the cutter will attempt to move to z zero (1 inch **below**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

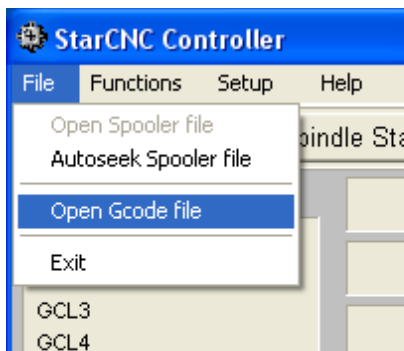
If you have a Cam program that outputs Gcode like Mastercam, Visual mill etc, you can bypass using Starcam and directly run your Gcode (.NC) files with the StarCNC spooler.

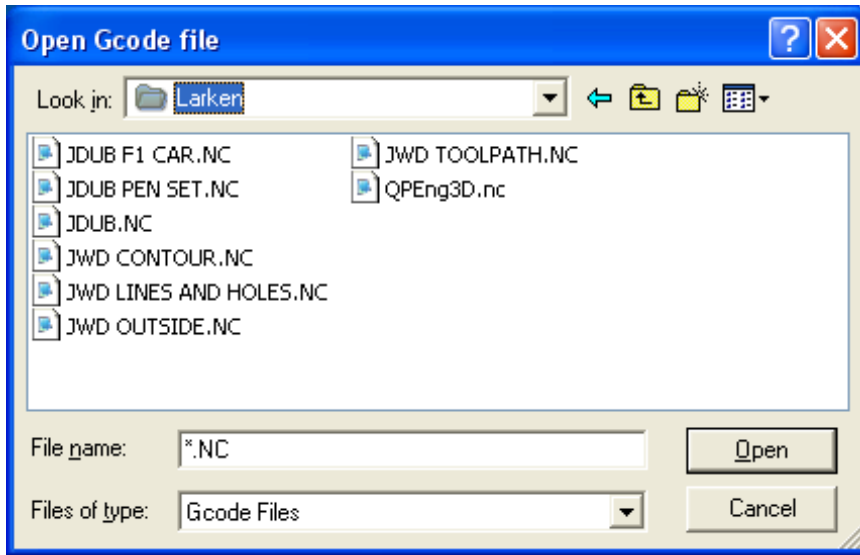
To set the Spooler to run Gcode files directly you need to set to Gcode_Mode in the Environment Setup. Then click Save_Setup and close.



SELECTING THE .NC FILE TO RUN

➤ Select File > Open Gcode file





- Select the .NC file and click on Open.

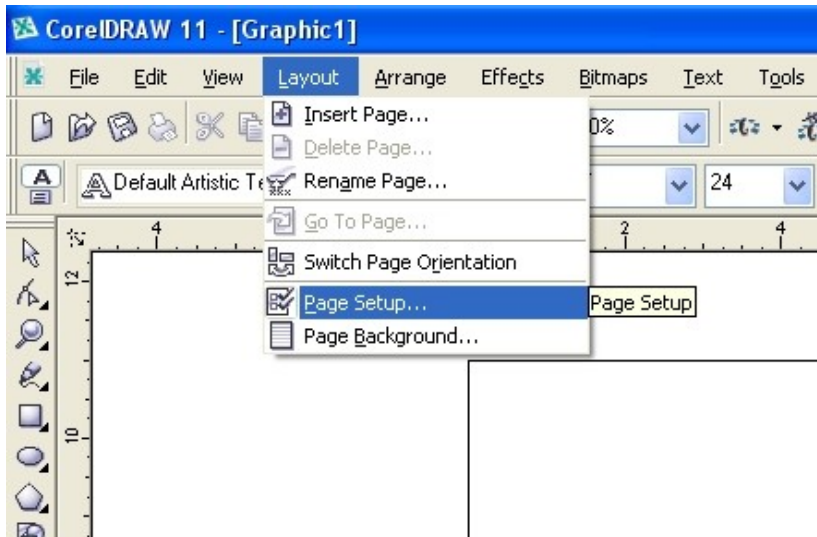
Note: Some CAM programs will use the extension .TAP .CNC or .TXT. You will need to change this Cam systems Post to .NC

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

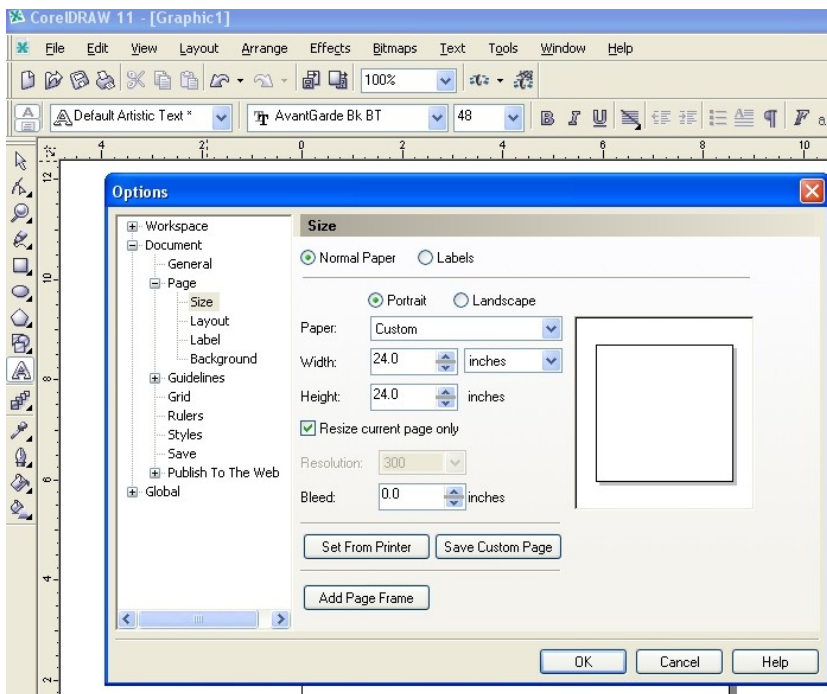
Using Corel Draw to create Signs and Tool Paths

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



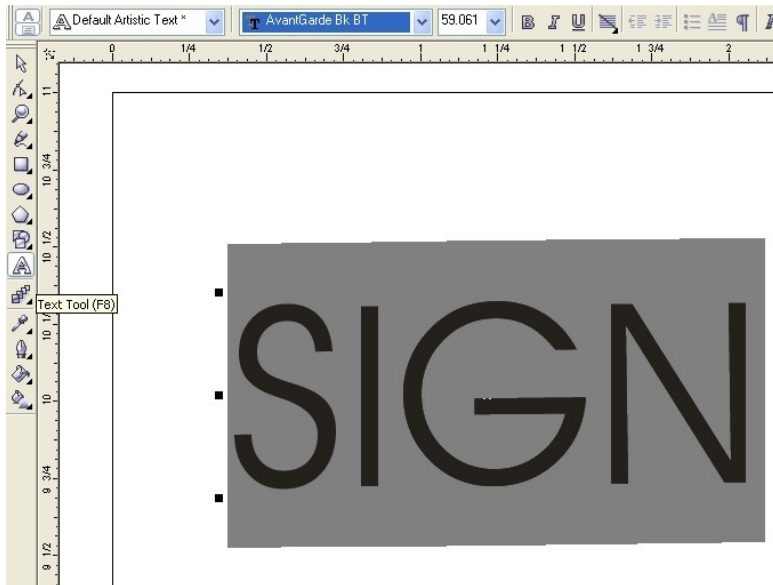
Change page type to Custom, and enter the Height and width you want.



Creating a Simple Sign

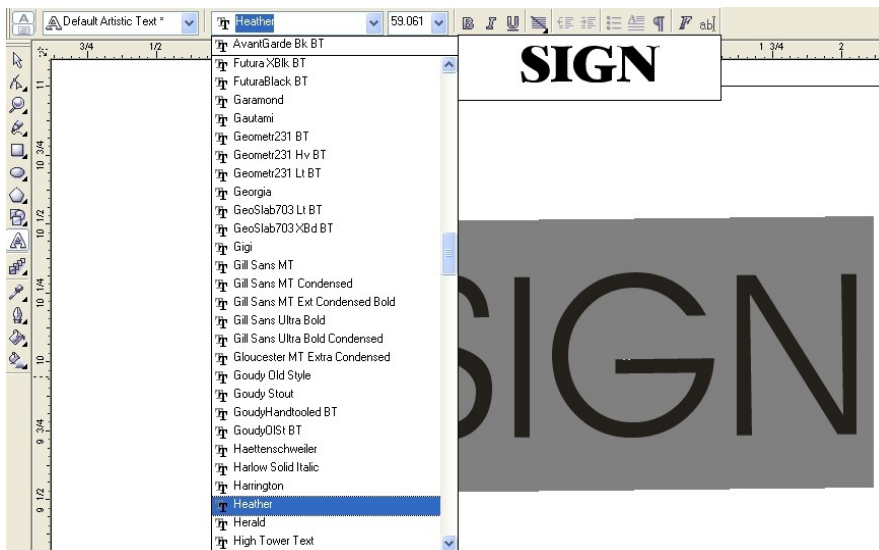
Here we will create a simple sign with a border.

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 where 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 with 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.

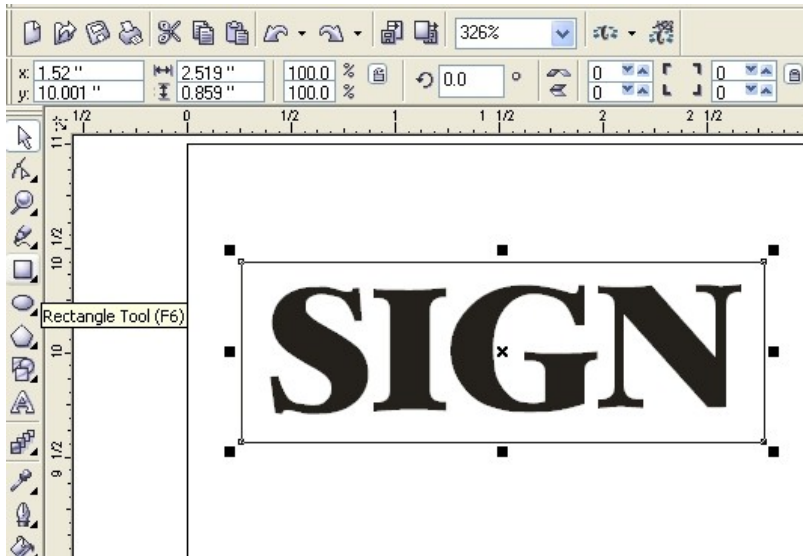
Click on the **Font Dropdown** on the upper tool bar, and use the up and down cursor keys to scroll through the fonts. Click to select the font.



Creating a Simple Box boarder

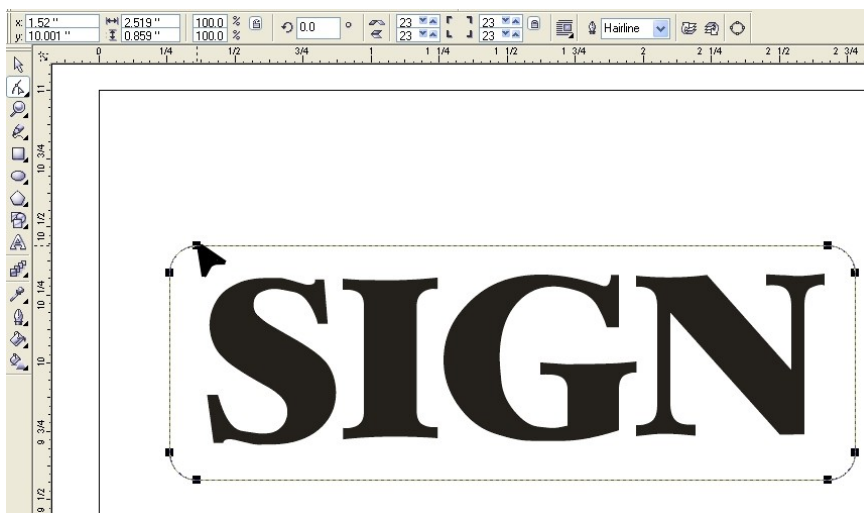
Click on the shape tool . The shapetool Icon has a Flyout menu to allow you to select a number of different shapes. To get the **Fly-out menu** menu hold the mouse clicked down on the small black corner in the lower right

Select the Box. With the cursor, draw a box by holding down the mouse and dragging. The release the mouse when done.



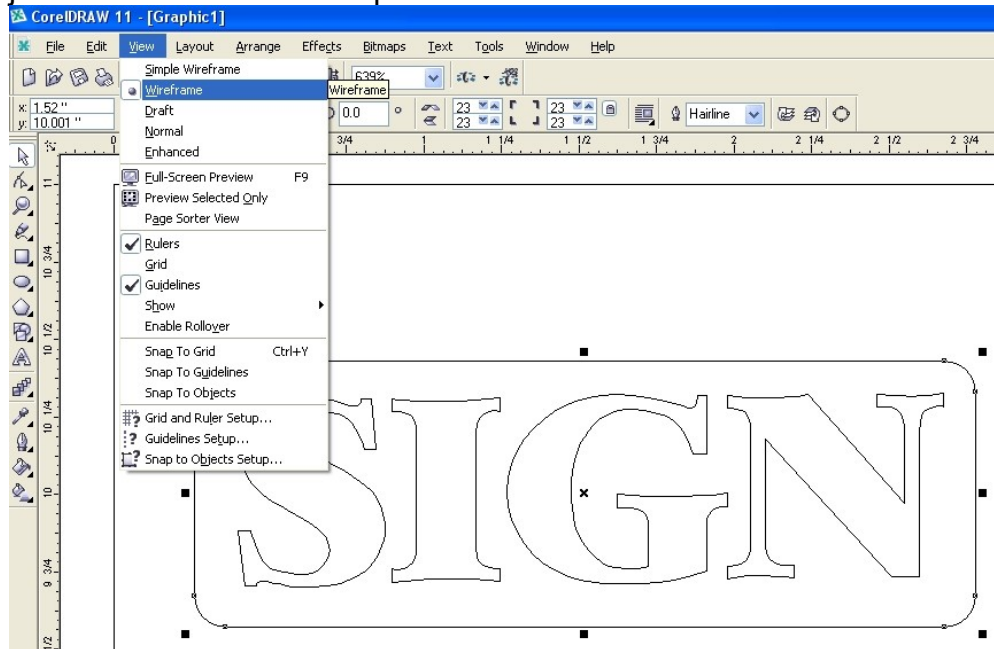
To edit the Box and round the corners , use the Node edit tool. It's the Icon just above the Zoom Magnifier in the Tool.

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)



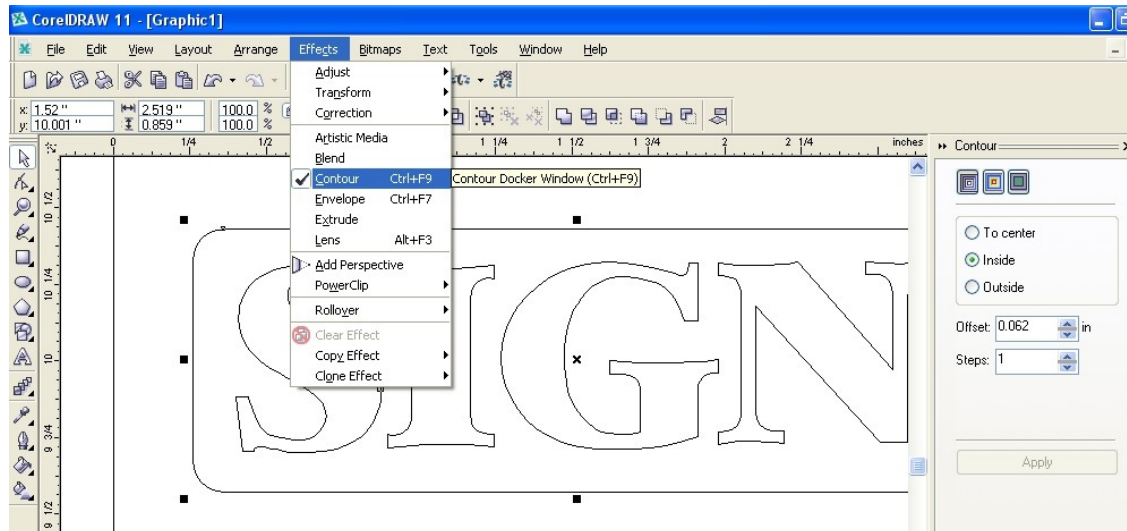
Changing View to Wire frame mode to see ToolPaths

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"

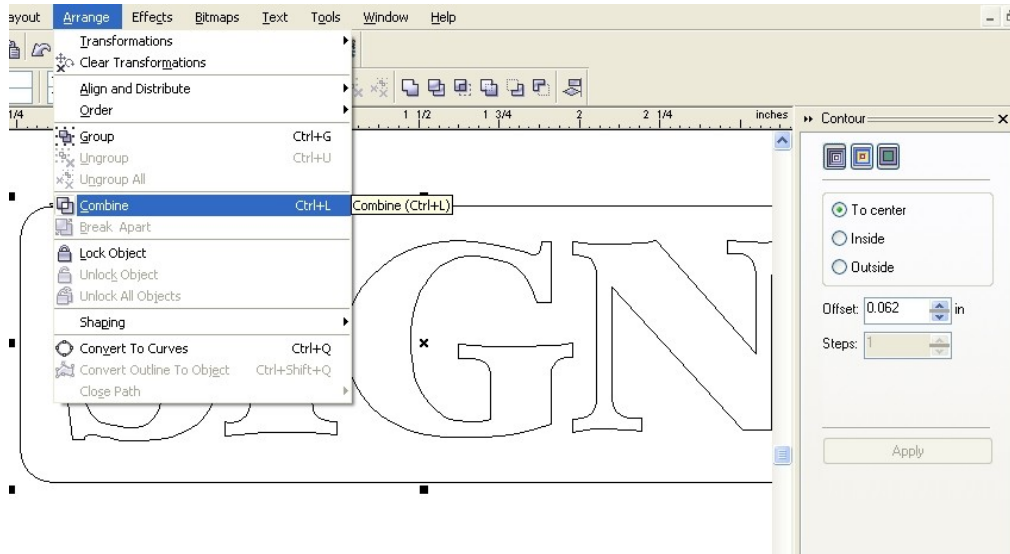


You can select whether you want the tool path on the outside of the object or the inside by checking the box. Also set the offset to the cutter radius. The number of steps sets the number of offset paths it creates. If you check 'To Center' it will create a fill path completely between objects.

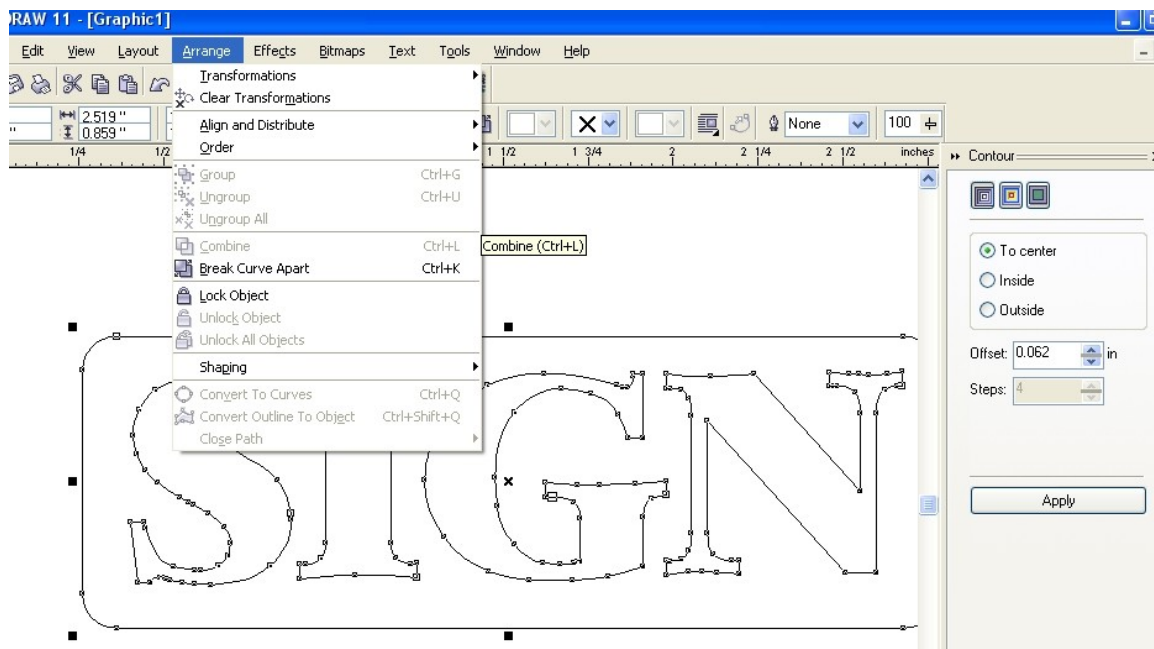
The Combine Command

Note that in the example shown the Apply button is not enabled, indicating that something is not ready. Use the Combine command to prepare the objects for the Contour command.

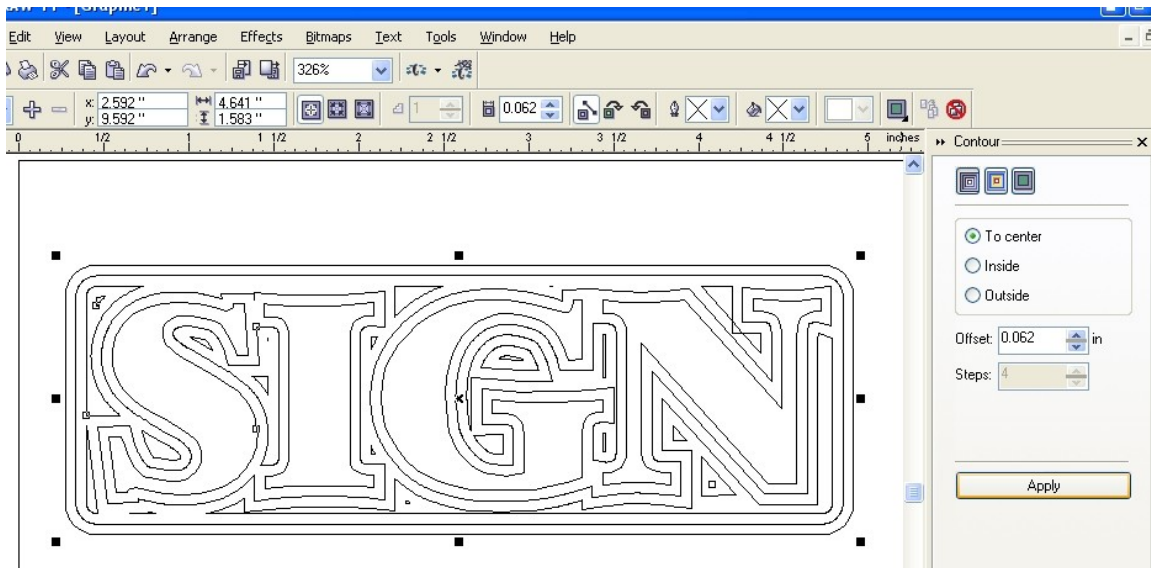
This command presets contours path direction (CCW=Inside and CW=outside) and order.



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.

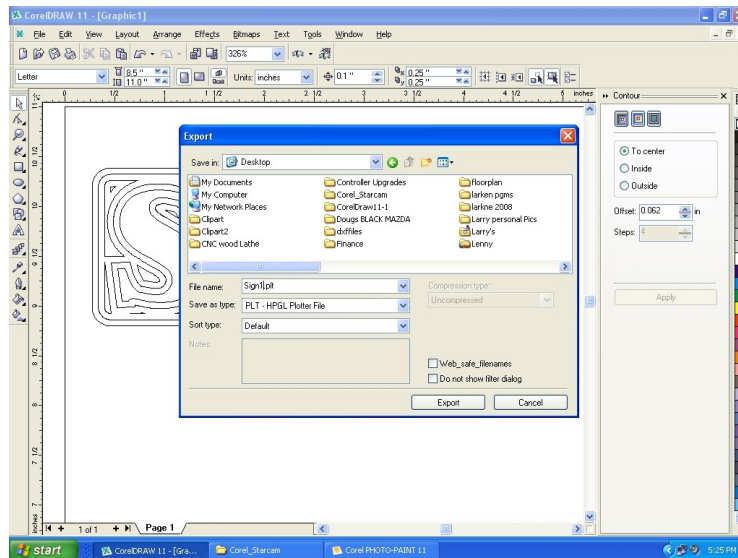
After they are separated you can set different colors for them so that they are separated then they are exported into up to 8 router Layers when they come into Starcam using the HPGL file format.

Saving output to a File

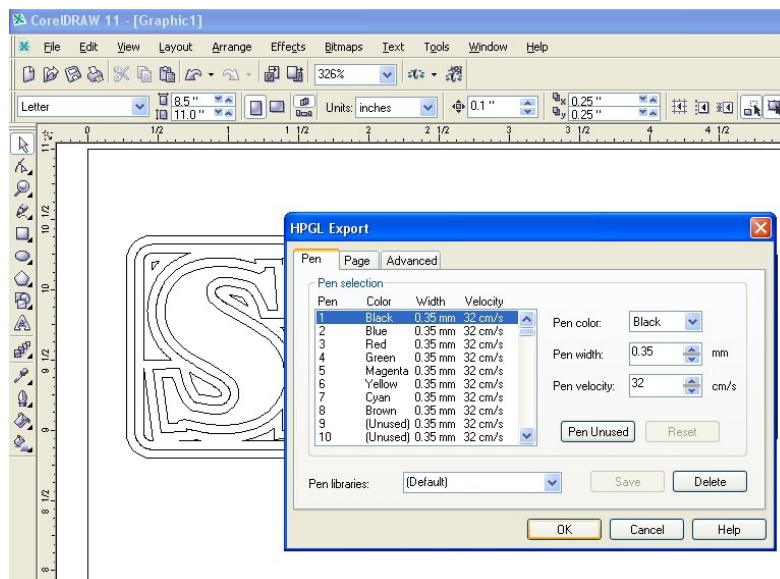
After you have created your Toolpaths, you need to Export the paths in HPGL (or DXF) format to a file.

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.



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.

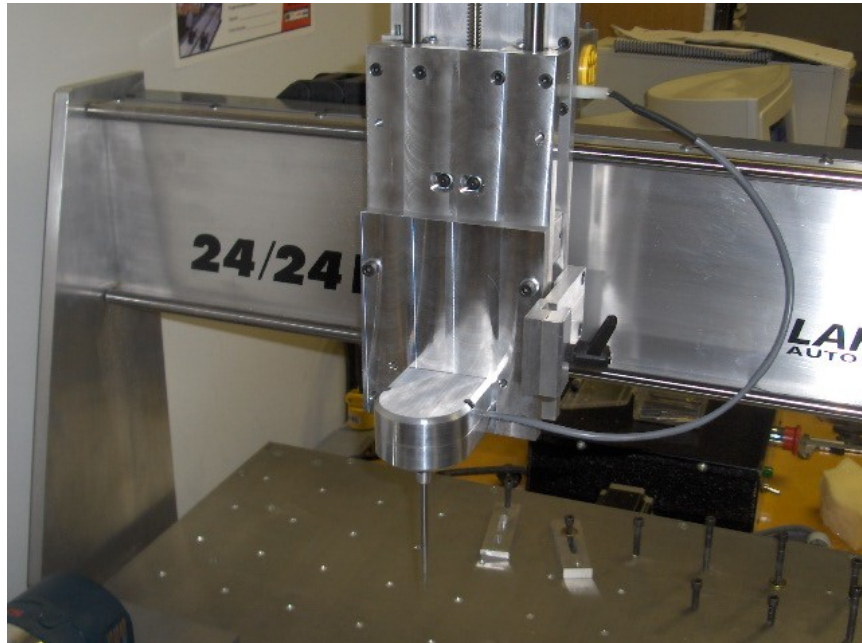


Use the HPGL import command in StarCam to load the file. Be sure to Sort it after importing.

Using the 3D Digitizing option With the Larken Camtool

Step 1 – Attach the digitizing probe

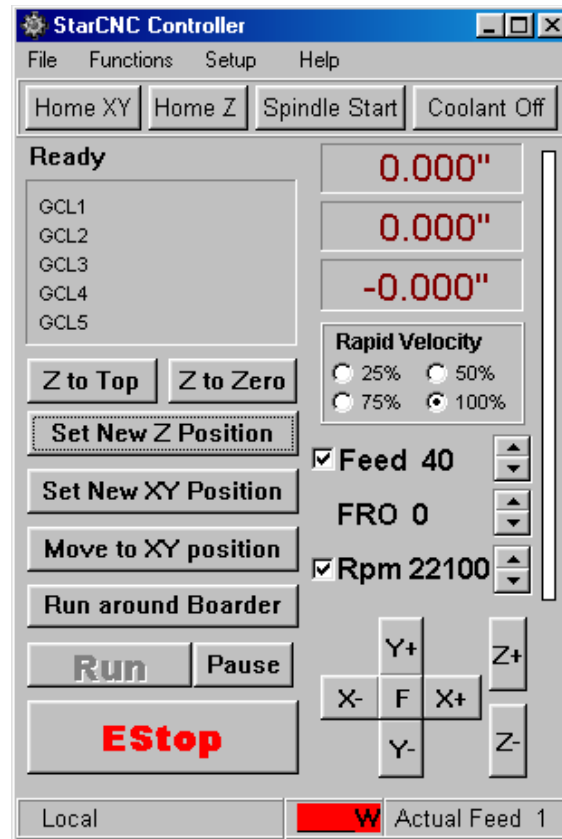
- Remove the 4 cap screws securing the spindle plate
- Replace with the probe (2 screws are usually sufficient). Secure it in a position that will allow the tip of the probe to reach the bottom level of your model.



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

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

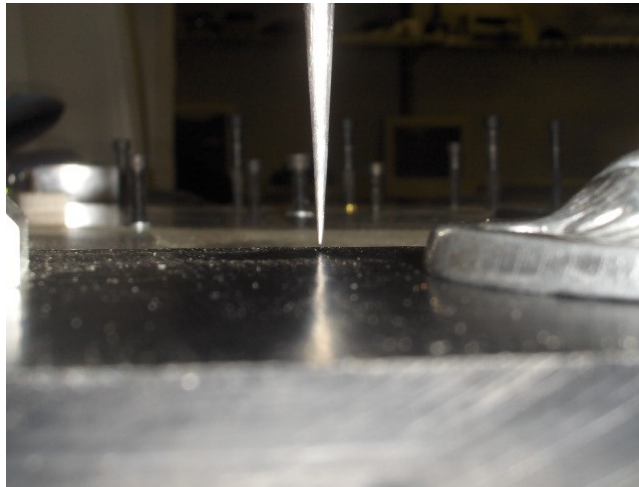
- 2-sided tape works well for models with flat bases.
- Power up the controller and open the Larken Router Control software. Notice that there is a W in the red display at the bottom. This indicates that the probe is connected.



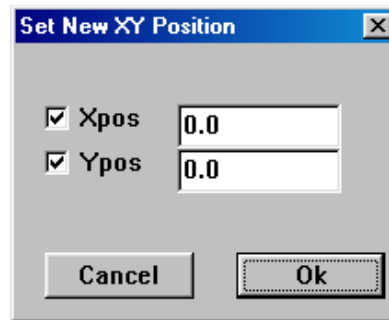
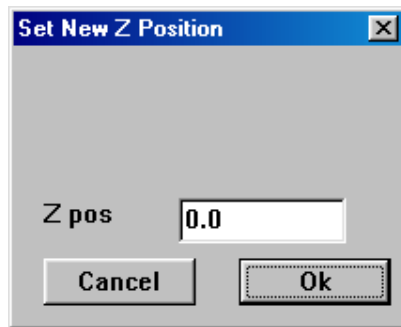
- Jog the tip of the probe to your start position (PRZ). It should be positioned so that the positive quadrant (x and y axis) will enclose your entire model. Set the z axis to the lowest level on your model that you wish to scan.

→
Positive Y Axis



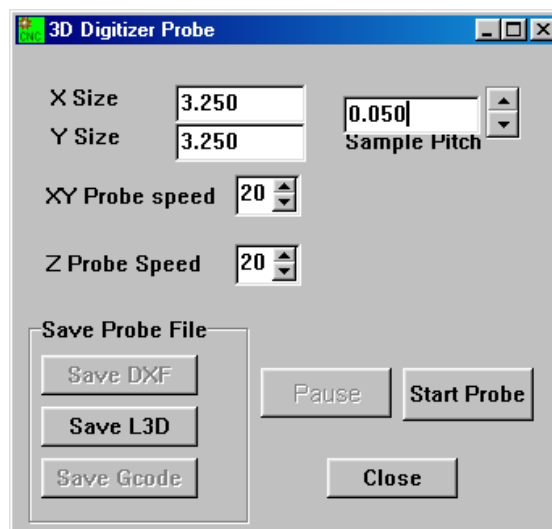


- Set the x, y, and z axis positions to zero.



Step 3 – Scan the model and save the DXF file

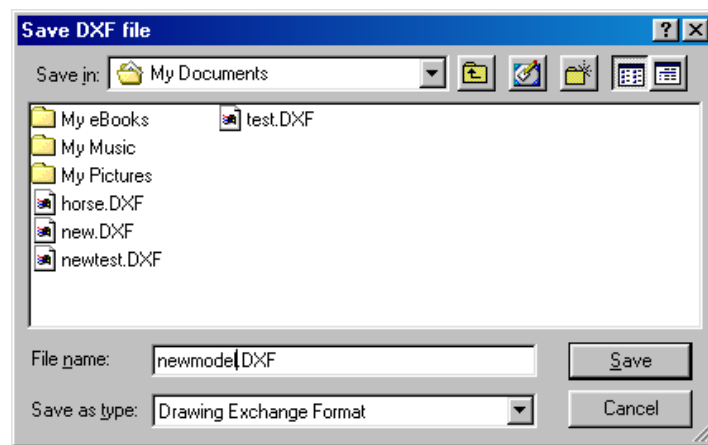
- Select the 3D Probe menu from the Functions drop down menu



- 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.

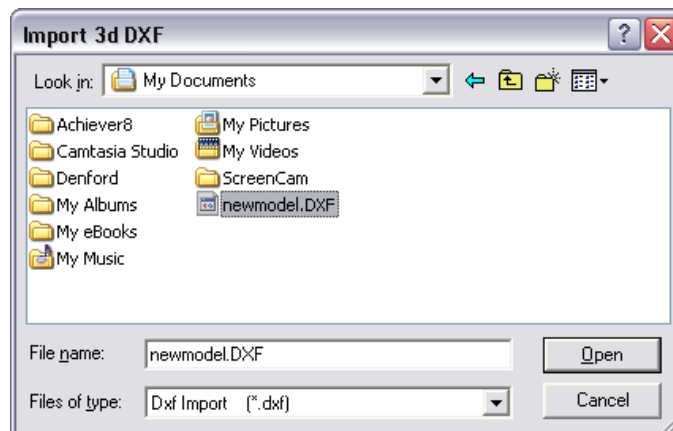
- Set the sample pitch. A lower number will create a more accurate model but will require more time to scan.
- Set the XY and Z probe speed to 20. This speed will usually result in a good error free scan.
- Select start probe
- When the scan is complete select Save DXF.



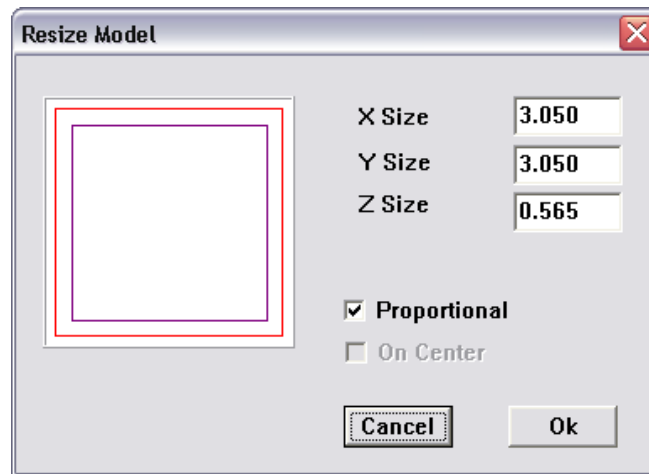
Take note of where the file is being saved

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

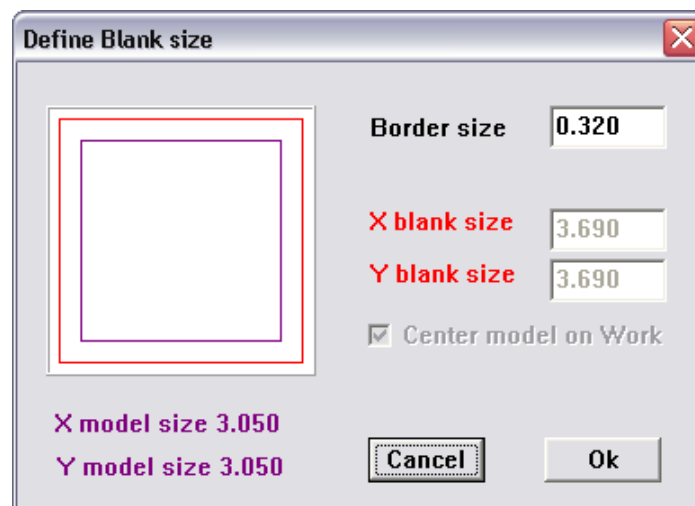
- Open Surfer 3D
- Select Files – Import 3D surfaces (DXF or STL)



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



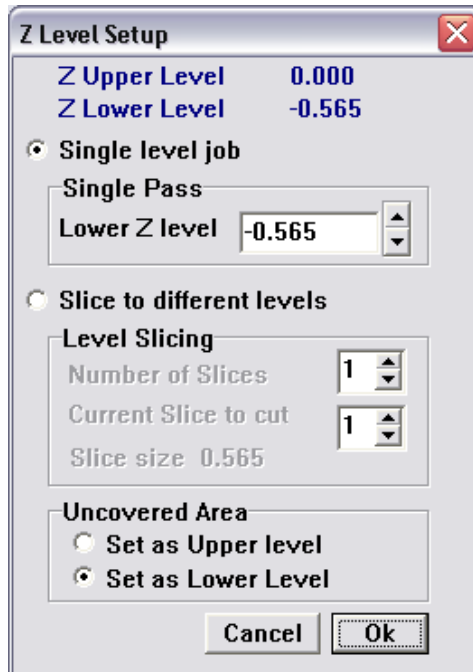
- Select Auto adjust origin from the edit menu or use the toolbar button. This will adjust the model so that it is below z level zero (top of the stock)
- Select Set border from the edit menu or use the toolbar button.



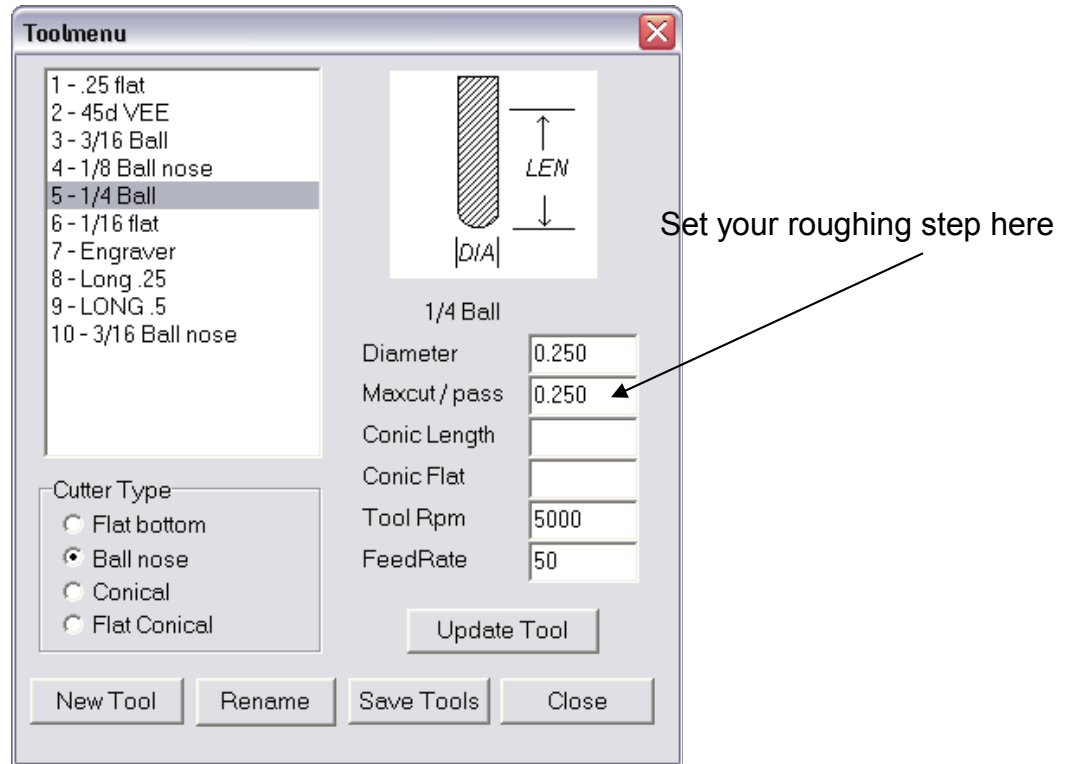
Warning – Make sure that your border is wide enough. It must be a minimum of the diameter of the cutter you will use to rough plus the stock you plan to leave for finishing. If it's too narrow the finish pass may cut a full depth cut because there was no roughing done to the border.

- 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

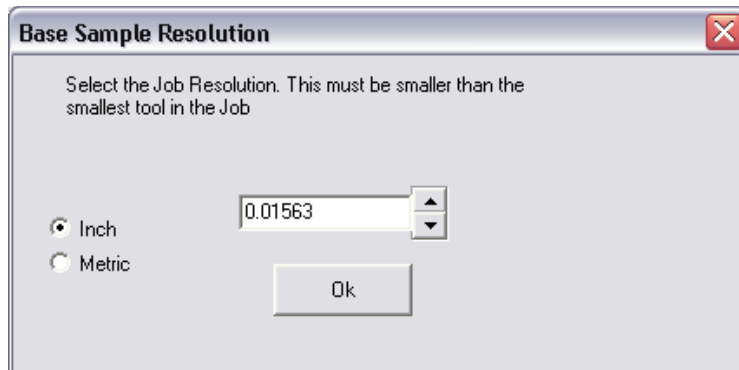
of separate pieces. Once they have all been cut they can be glued together to create a finished piece.



- 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.

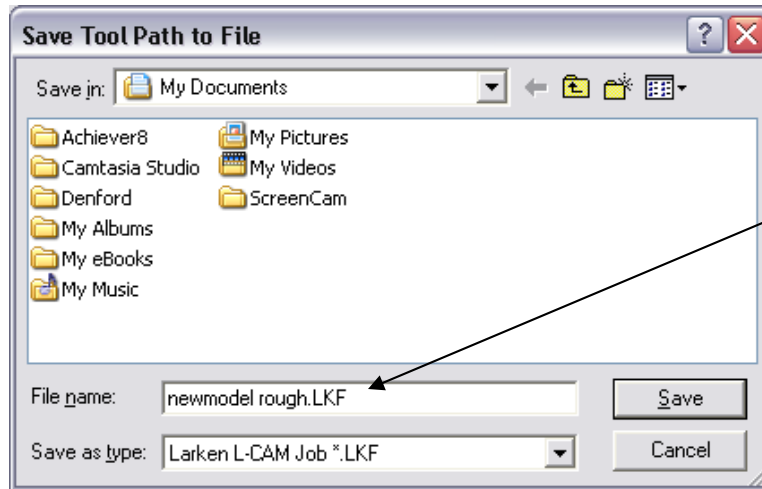


- Select the toolpath button
- You must set the base resolution first. This setting will limit the stepover of the tool during the finish toolpath. Using a smaller setting will create a smoother finish with less sanding but will take more time to cut.



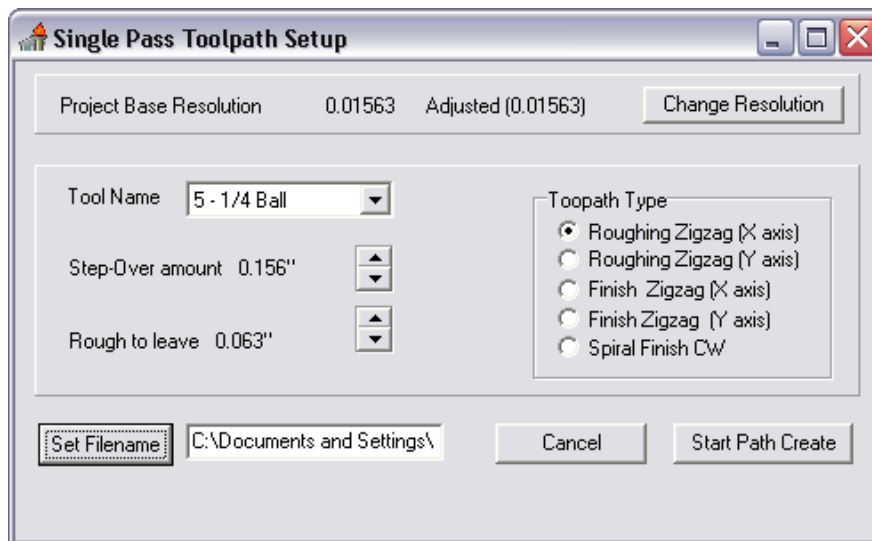
- Select the Set filename button. Give it a name that will indicate that it is a roughing toolpath.

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.

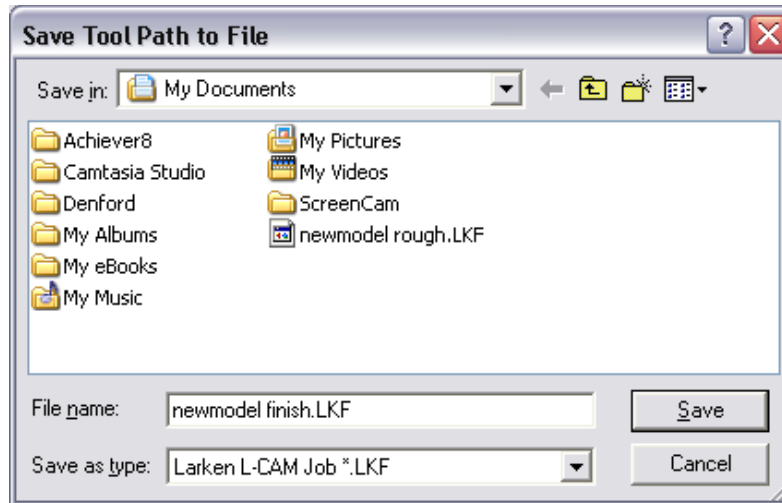


Make sure the .LKF is in your filename

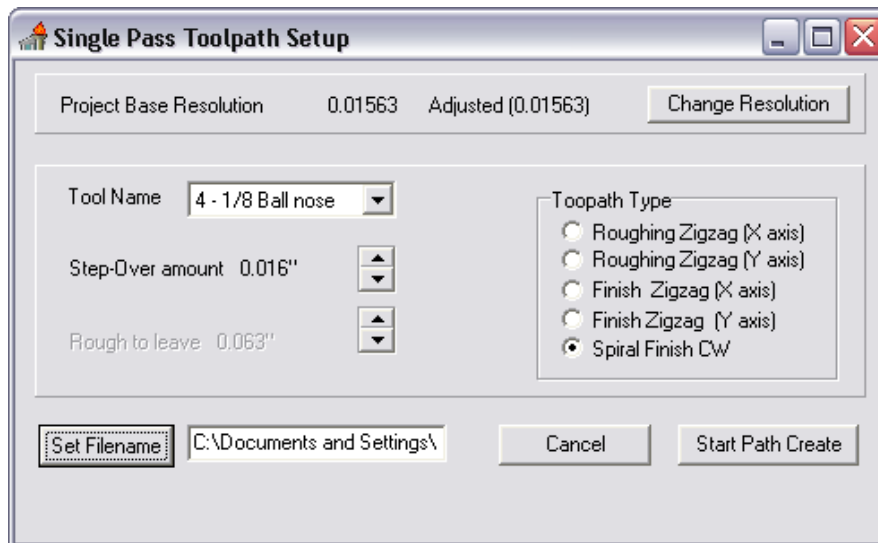
- Select save
- Select the tool to use, the stepover, and the amount of stock to leave. For the roughing cycle the stepover can be set to about 75% of the tool diameter. This will cut down on the machining time.
- Select the Roughing Zigzag (X axis)



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



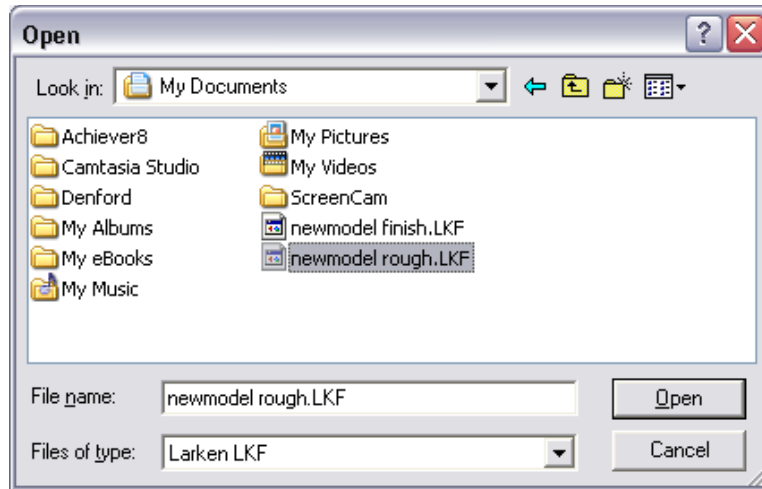
- Select Save
- Select the Spiral Finish CW
- Select the tool to use and the stepover. A smaller tool will create the most detail but will take longer to machine.



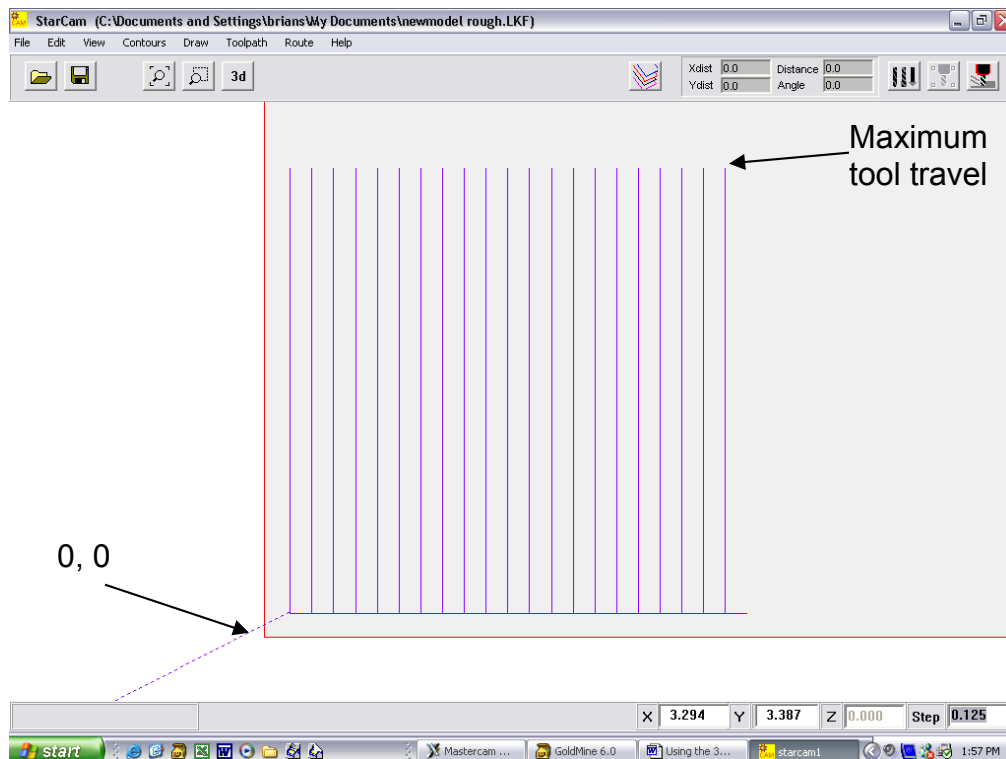
- Select Start Path Create. This will create the finish toolpath file.

Step 5 – Cut the Roughing Toolpath

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

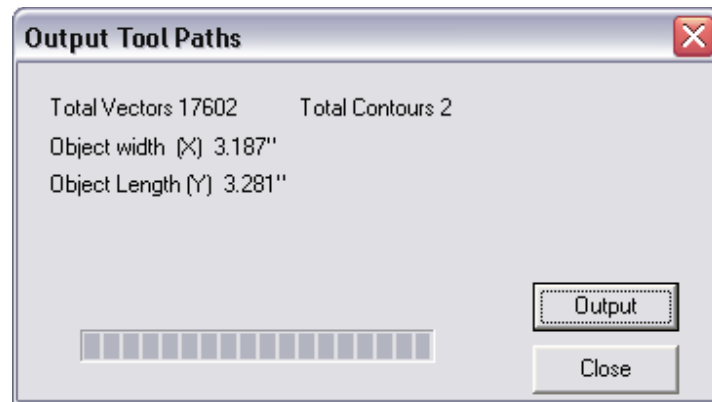


- Open
- Right click and select zoom half



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

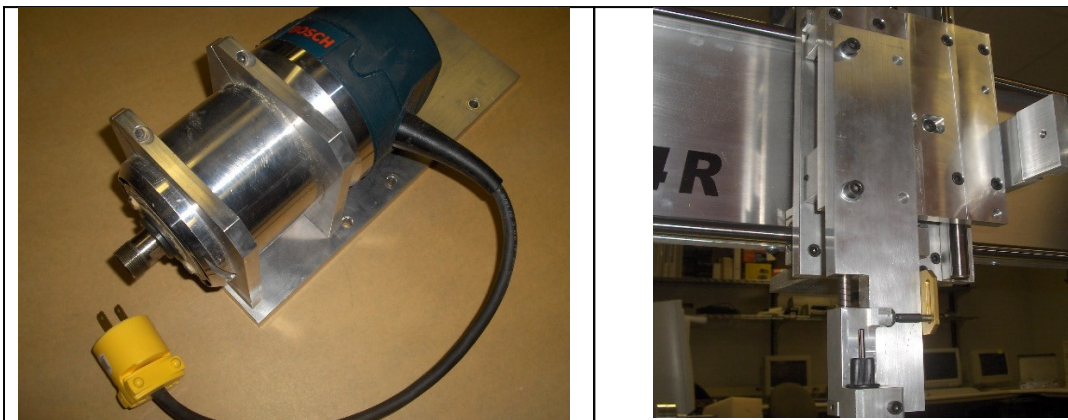
Step 6 – Cut the Finish Toolpath

- Repeat step 5 using the finishing toolpath file.

Using the Vinyl Sign Attachment with the Larken Router

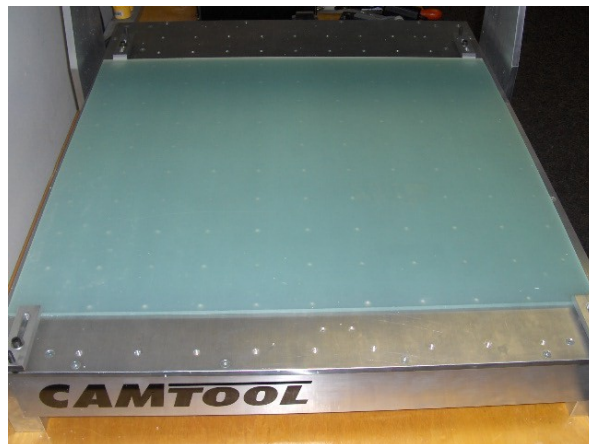
Step 1 – Remove the Spindle head complete with bracket and attach the vinyl sign attachment

- 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



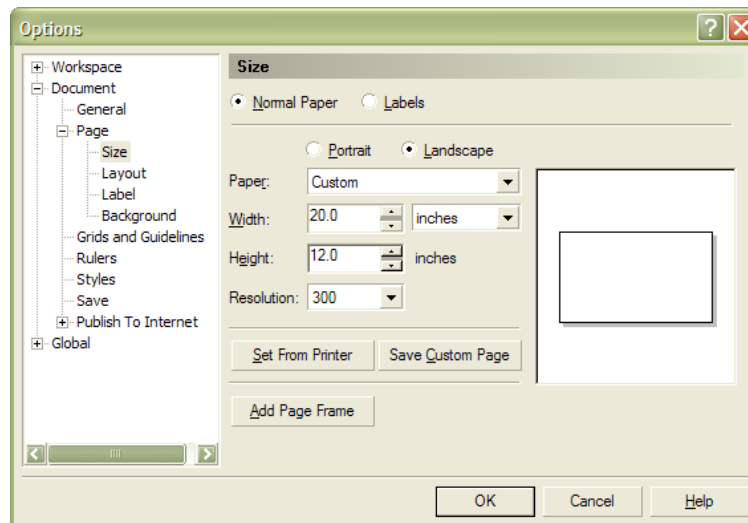
Step 2 – Mount a suitable platen to the worksurface of your machine

- 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

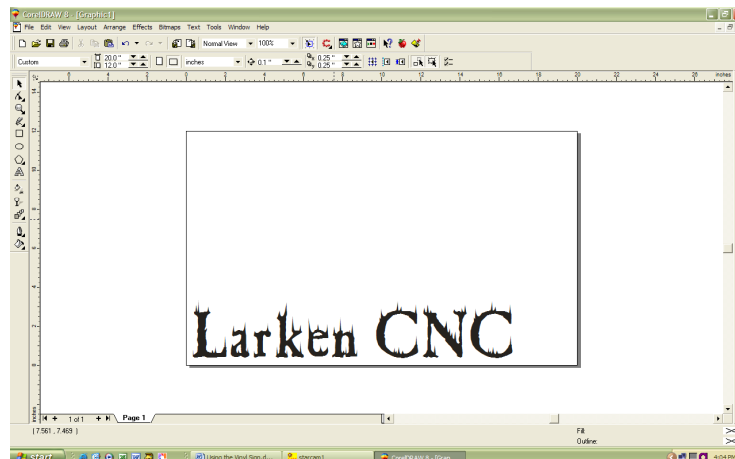


Step 3 – Create an HPGL (plotter file) to run on the Larken

- 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



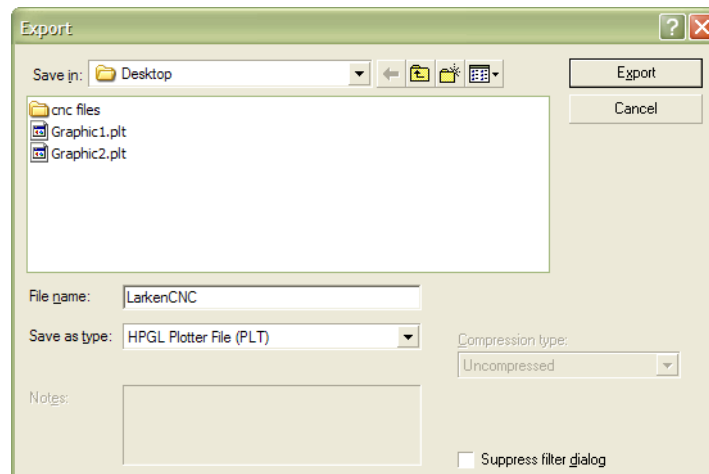
- The maximum page width is determined by the size of your machine
- Select the text tool and click in the graphics area
- Enter your text
- Select the selection tool and adjust the size of the letters and choose a font
- Position your letters on the page. The letter position determines where they are 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)



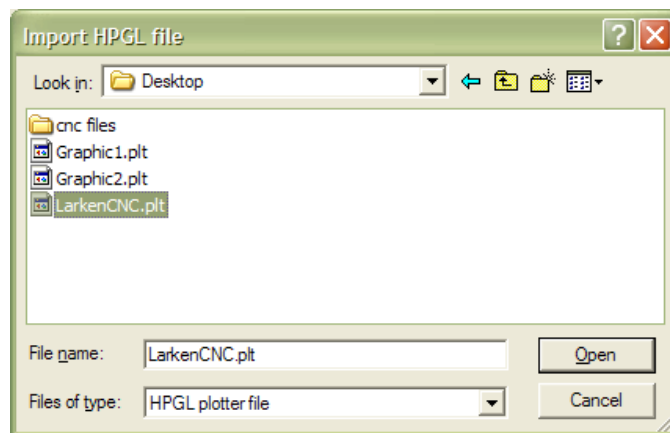
Step 4 – Cut and mount a piece of vinyl that matches the size of the page setup in your drawing program.

- 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

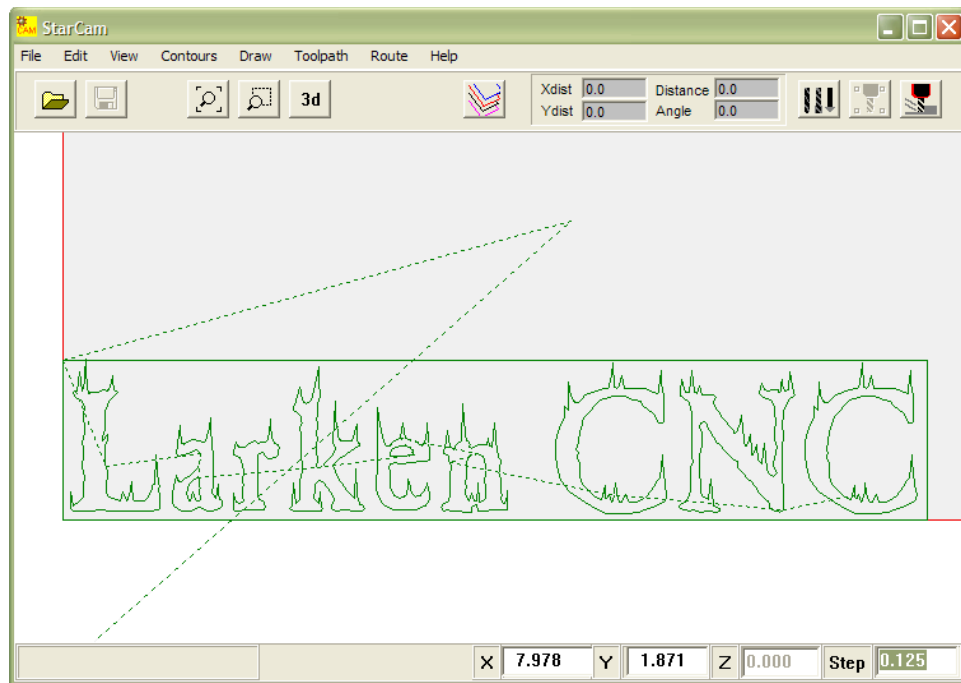


Step 5 – Import file into Starcam and create spooler file

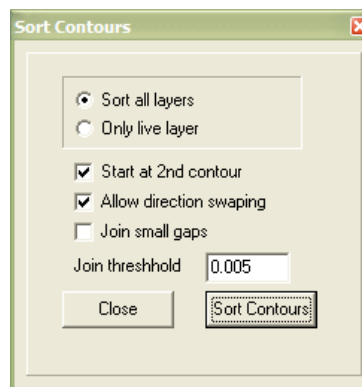
- Open the Starcam software
- Select file – import hpgl



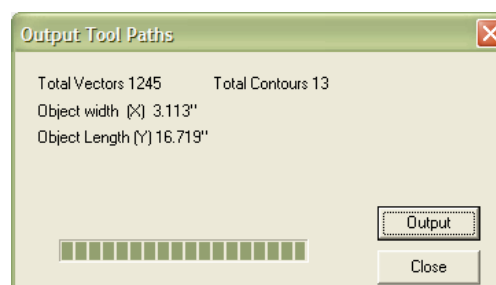
- Open
- Right click in the drawing area and select zoom all. Your design should be shown in the lower left corner of the machine work area (grey section)



- Select contours – Sort contours



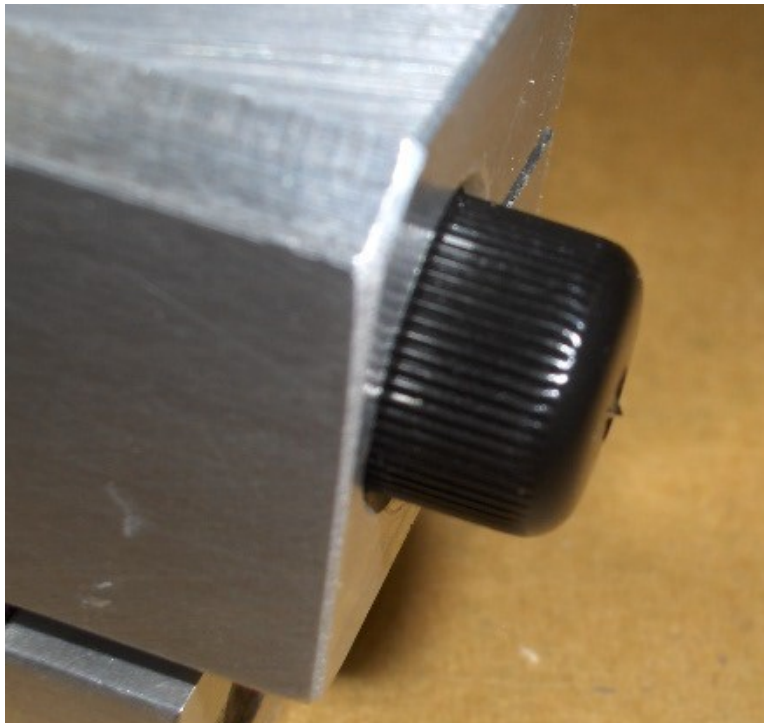
- Sort Contours
- Close
- Select Route – Route all



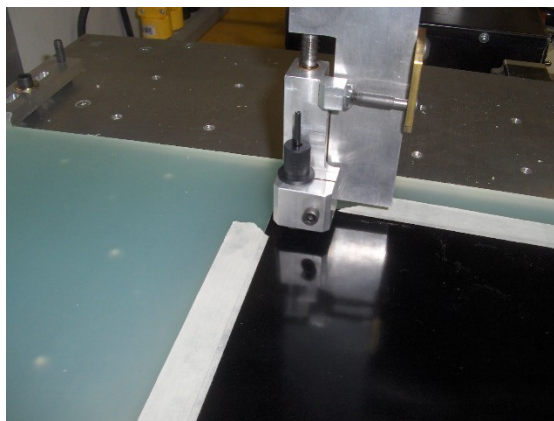
- Output

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

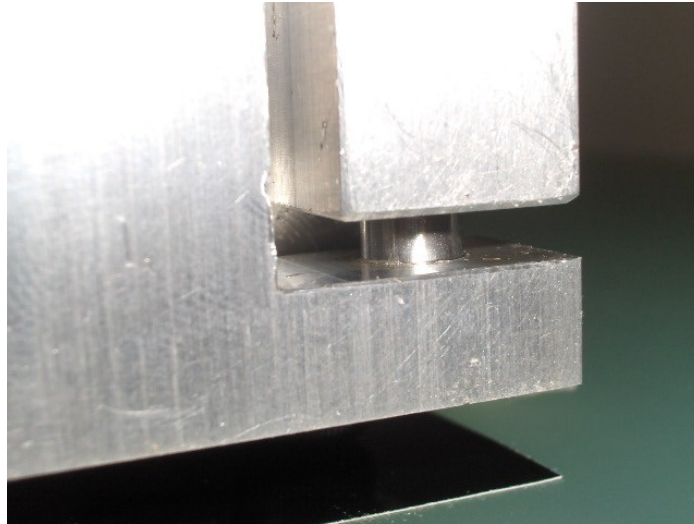
- Adjust the cutter head so the cutter tip is just protruding from the end of the holder. Adjust it by turning the end cap on the holder in or out. *The adjustment is correct when the vinyl is cut completely through but not through the backing underneath. A few test cuts may be required to get it correct.*



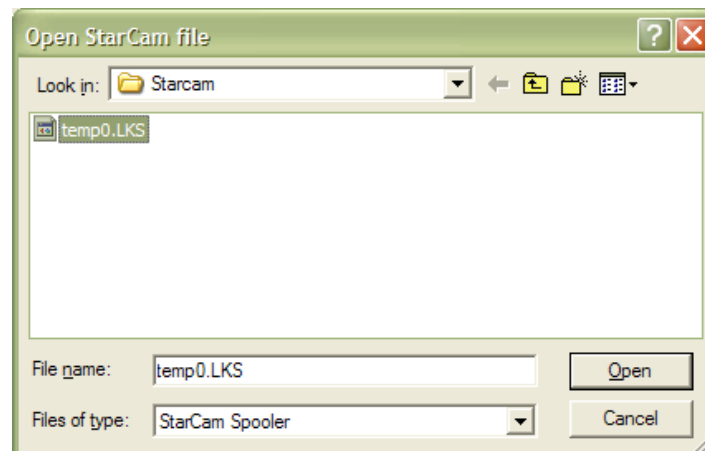
- Open the Larken router control software
- Jog the tip of the cutter to the lower left corner of the vinyl. It should be far enough in from the corner to avoid hitting the hold down tape.



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



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



- 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.*



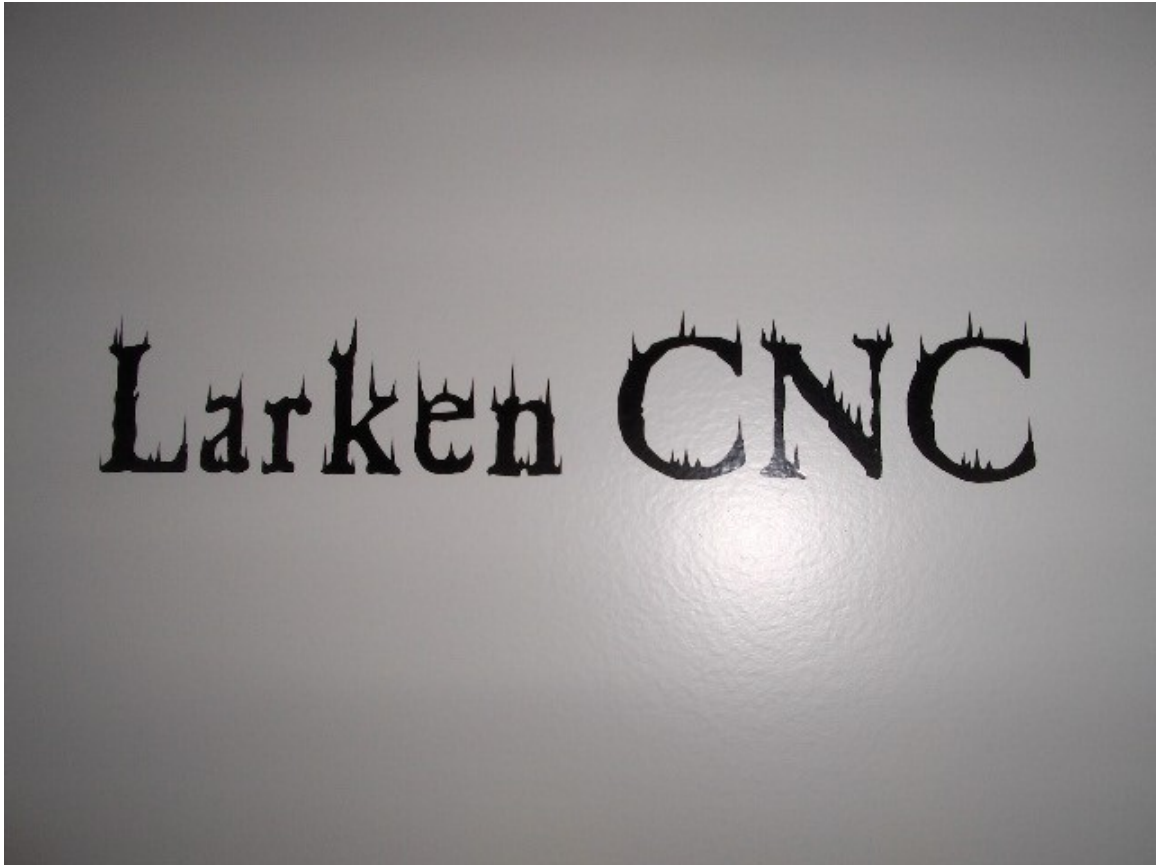
- Lift the image from the backing



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

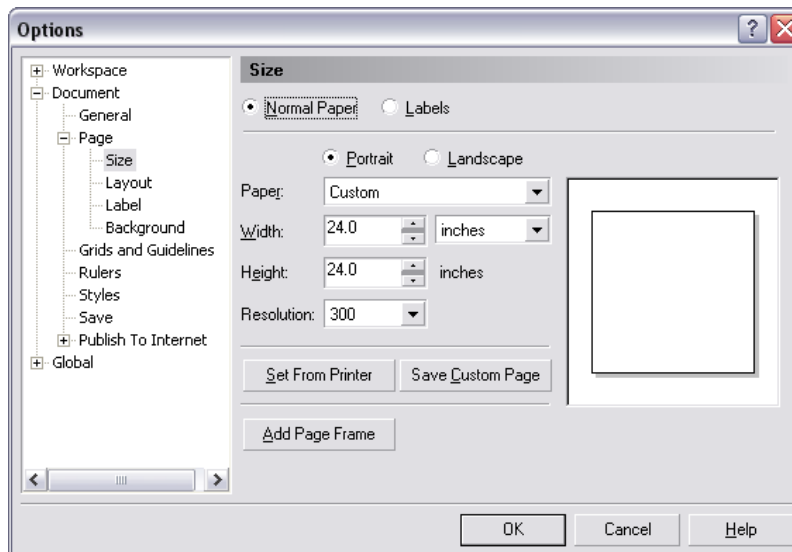


Step 2 – Mount the sign stock to the worksurface of your machine

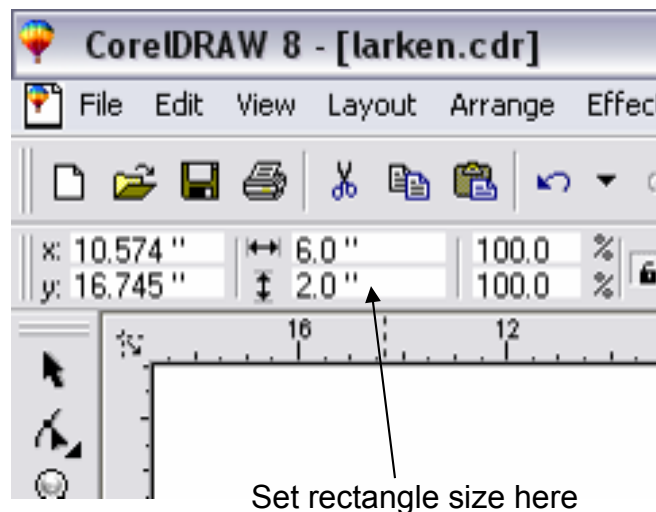
- The stock can be held with 2 sided tape to a suitable platen to prevent damage to the machine table
- Consider designing a hold down fixture if you are cutting multiple signs out of the same sized stock

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

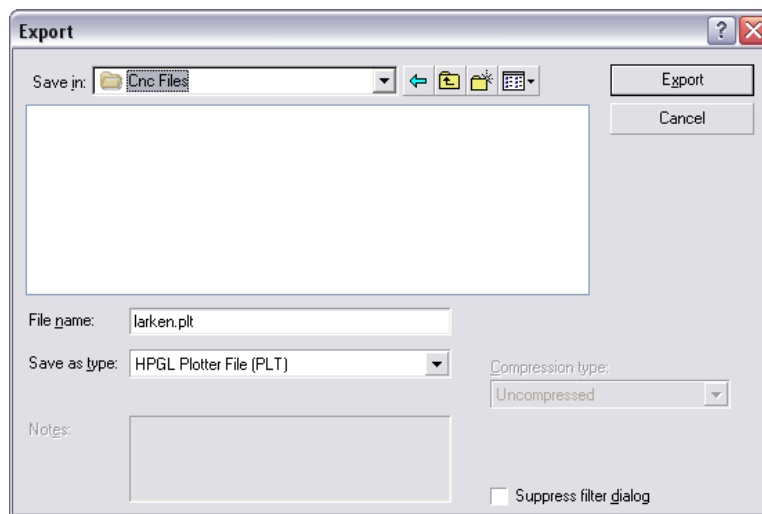


- Use the rectangle tool to create a rectangle of any size
- Edit the height and width of the rectangle to match the size of the sign material that you are using
- Select the text tool and click in the newly created rectangle
- Enter your text
- Select the selection tool and adjust the size of the letters 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.



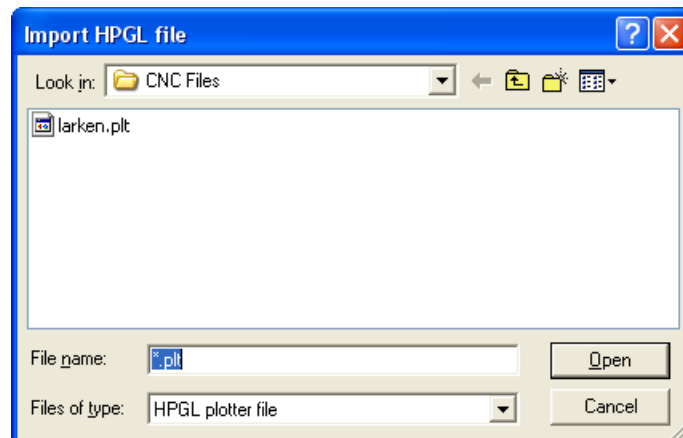
Larken

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

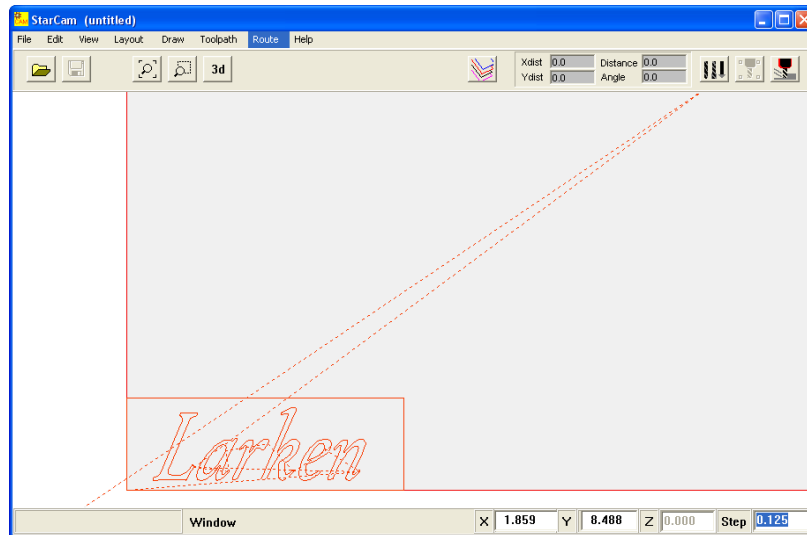


Step 4 – Import file into Starcam and create spooler file

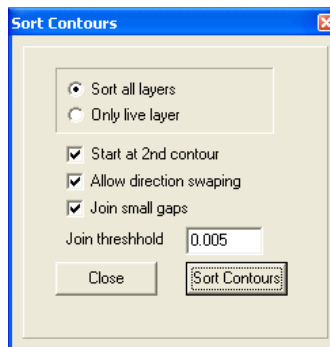
- Open the Starcam software
- Select file – import hpgl



- 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)



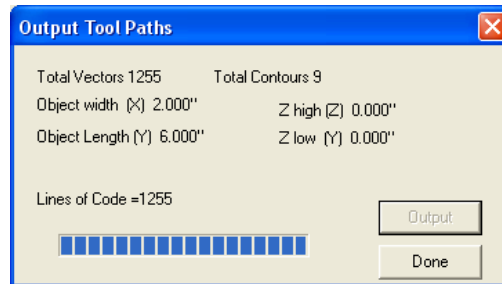
- Select Layout – Sort contours



- 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

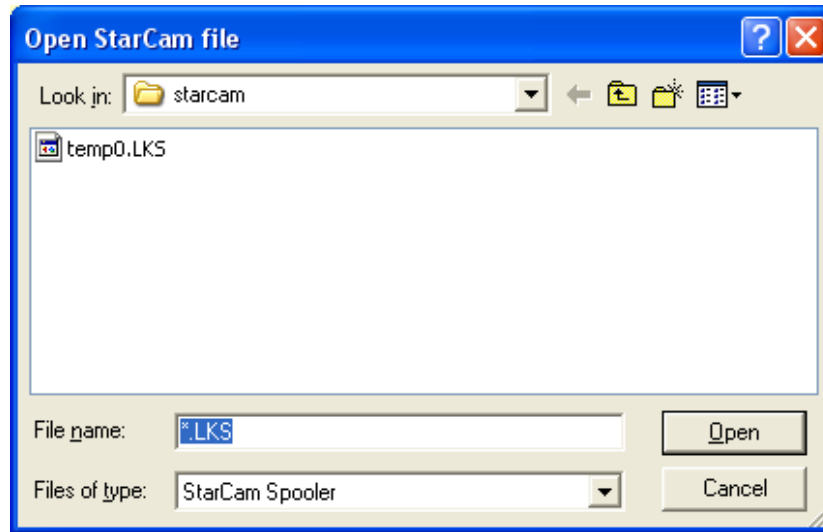
- Set the cutter to a zero depth. *Turn the adjusting head until the tip of the cutter is flush with the end of the holder. The indicator should read zero. If it doesn't you may have to adjust the cutter in the collet. (assembly pics at the end of tutorial)*



- 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 $\frac{1}{4}$ inch.

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



- 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



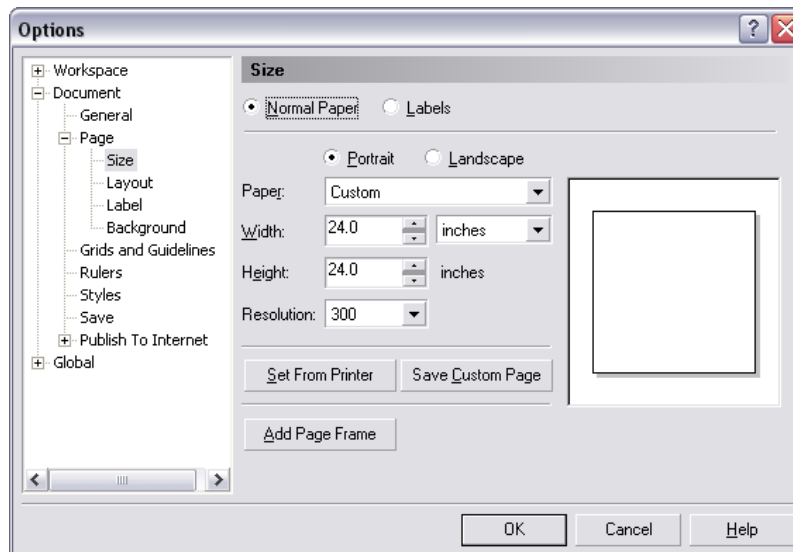
Step 2 – Mount the material to be engraved to the worksurface of your machine

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

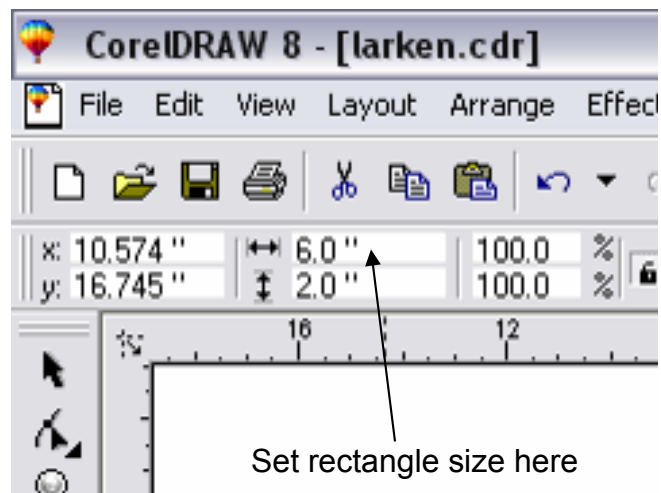
Step 3 – Create an HPGL (plotter file) to run on the Larken

- 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

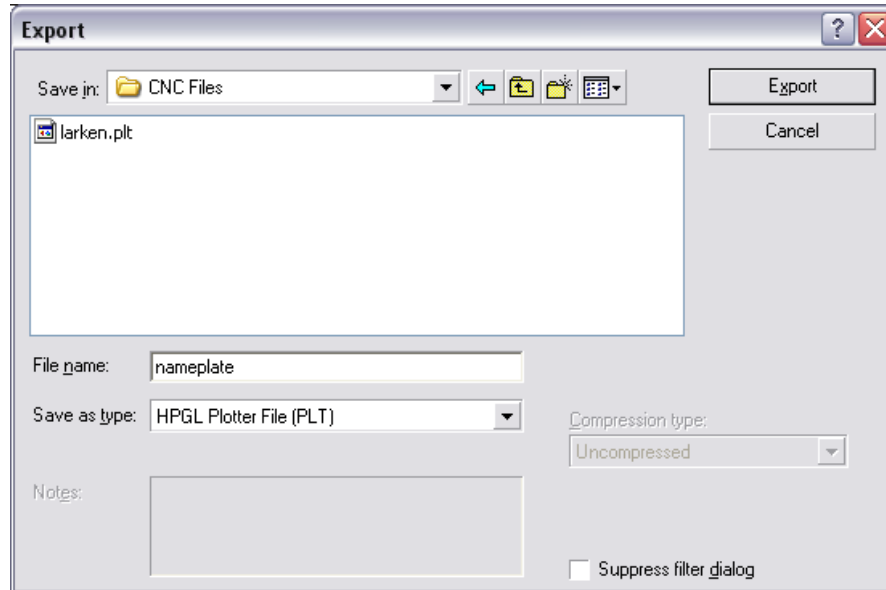


- Use the rectangle tool to create a rectangle of any size
- Edit the height and width of the rectangle to match the size of the material that you are using
- Select the text tool and click in the newly created rectangle
- Enter your text
- Select the selection tool and adjust the size of the letters 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.



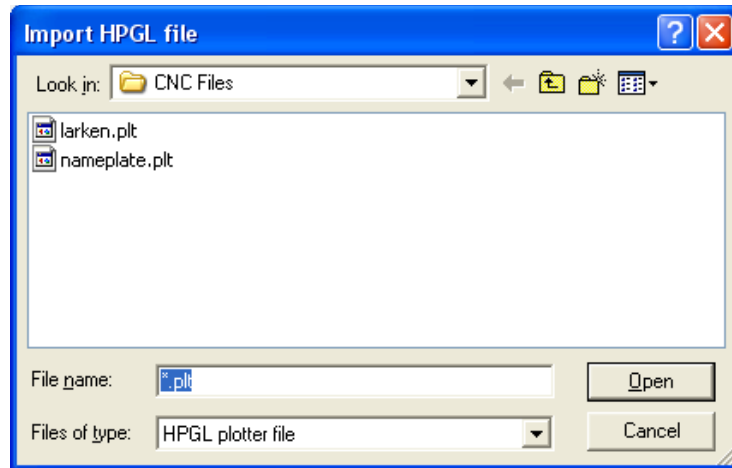


- Select File – Export
- Export a plotter file (*.plt)

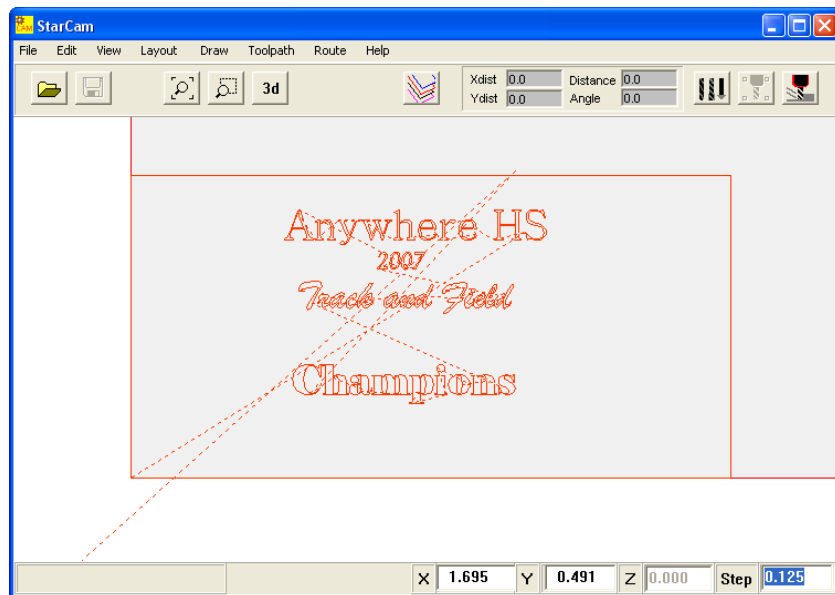


Step 4 – Import file into Starcam and create spooler file

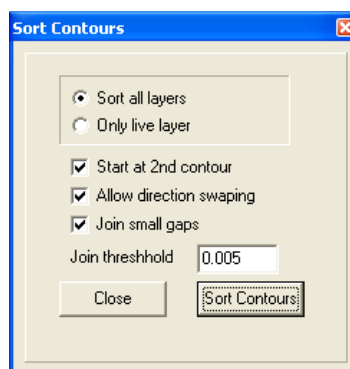
- Open the Starcam software
- Select file – import hpgl



- 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)



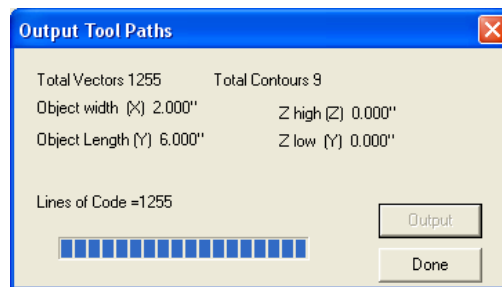
- Select Layout – Sort contours



- 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

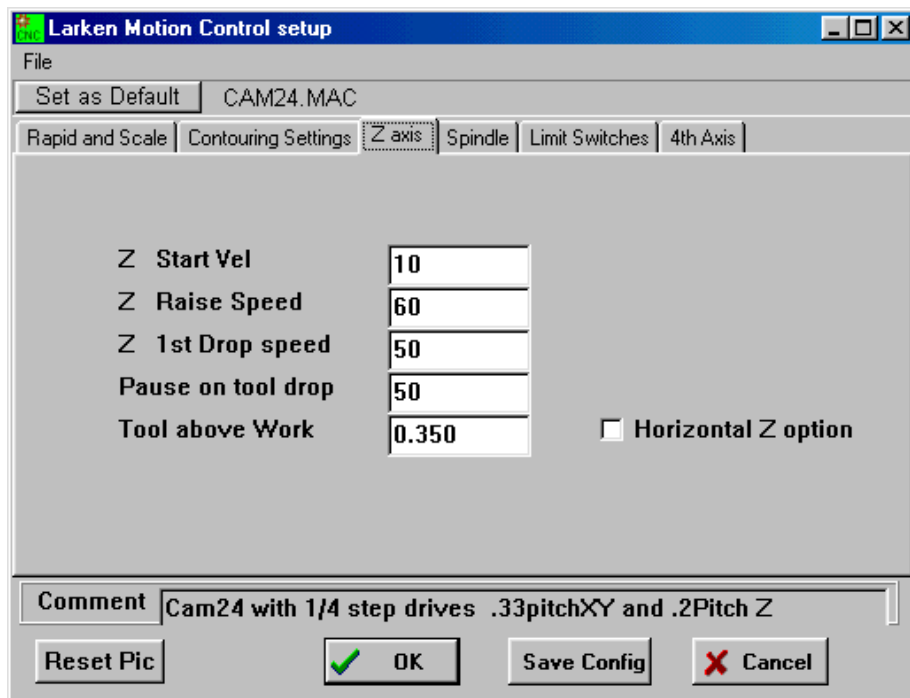


Step 5 – Set the PRZ (program reference zero), adjust the drag pressure and run the program

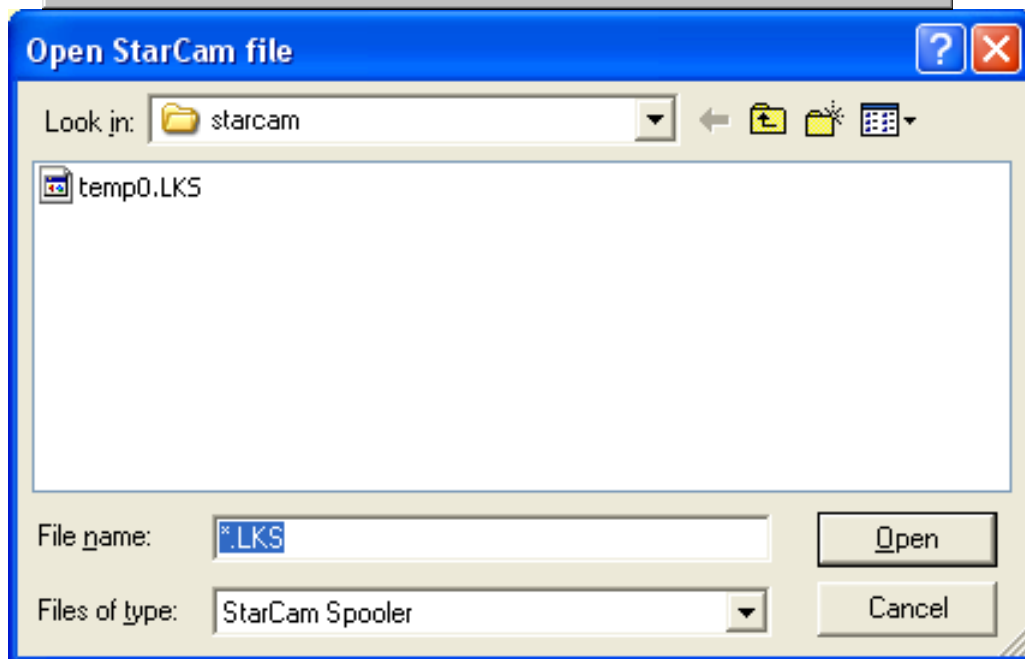
- Open the Larken Control Software
- Jog the tip of the diamond drag to the lower left corner of the material to be engraved

- 
- 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



➤ Select File – open spooler file



➤ 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:
<http://www.larkencnc.com/downloads/index.shtml>
- 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 lspooler.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

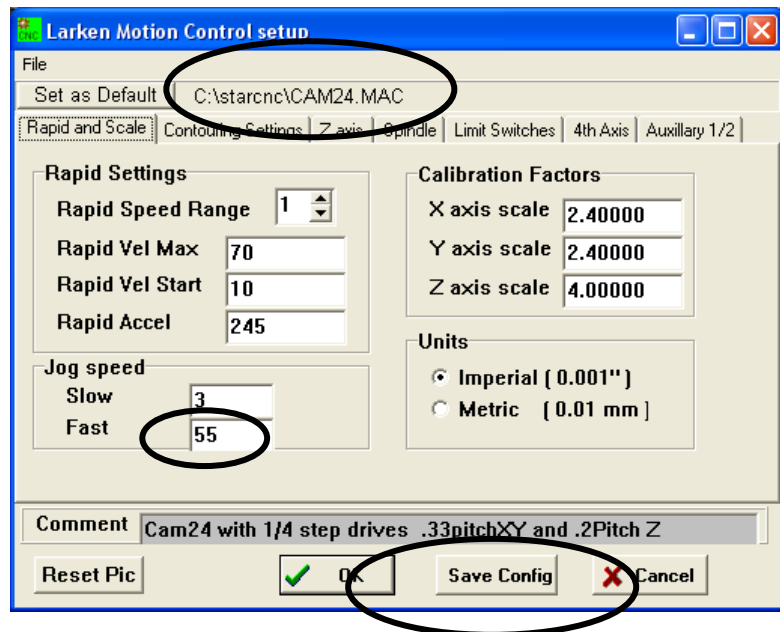
The Larken Router must be connected to the parallel port and be turned on to perform the next step

- 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)

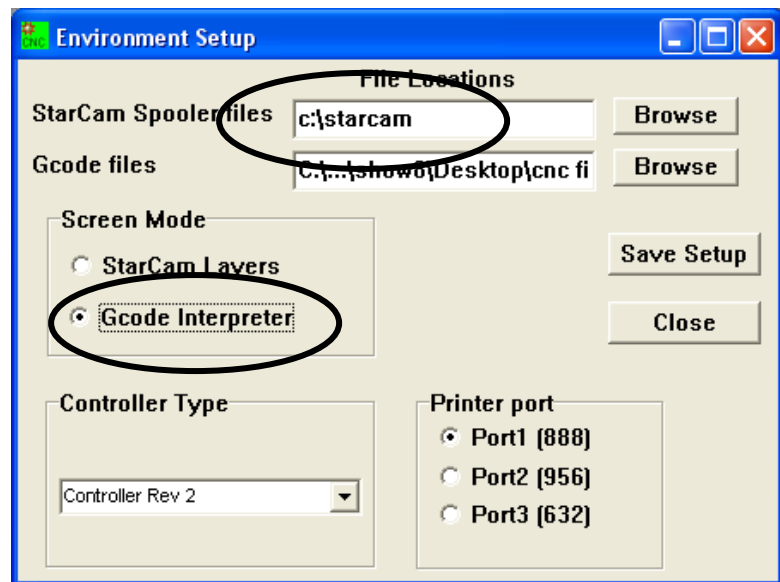
- Change the **Fast Jog speed** to 55 under the Rapid and Scale tab

- Select the **Save Config** button



- 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



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

