# Using the CNC Plasma Cutter (SMARTCUT)

Digital Media Lab Tutorial

# **Create a Profile to Cut**

The Plasma Cutter reads DXF profiles. Create or open your profiles in Autocad and scale them to the size you need to cut. DO NOT include the material blank dimensions in your drawing.

The max bed size for the plasma cutter is 94" long by 47" wide.

Note: profiles and openings inside the profiles all need to be closed as well, and holes cannot overlap each other.



Save your file to **E:\temp** and create a folder with your initials.

For "Save as Type," select: AutoCAD 2000/LT 2000 DXF

NOTE: When you draw shapes in programs like AutoCad, using arcs, line segments, etc., the resulting cut will be the same bunch of arcs and lines. This will cause the torch to go on and off after each segment, travel in unpredictable directions, etc.

The shape must be converted into a continuous outline, without any breaks.







# Open the SMARTCUT CAD Software

Custom Shop Palette Wizard

SMARTCUT CAD generates the tool paths for the plasma cutter. Select **Start > Programs > Fabrication > Torchmate CAD** to launch the program.

				<u>a a a a</u>	MODELA Roland MODELA Player 4 Roland Virtual MODELA CROMA 1.53	* * * *
<b>i</b>	Programs	, 🖻	3D Applications Accessories		, CodeBreaker_STD_V5.exe PowerMILL (Metric) 8.0.09	
Ì	Documents	•	Documentation	+	PowerSHAPE 7.2.40	
	Settings	. 🕅	Drafting Engineering	+ 🖻	Help Files Torchmate CAD	>
	Search	•	Fabrication	•	) Torchmate 3	•
	Help and Support		Graphics Internet	1		
2	Run		Startup Utility	;		
	Log Off eberhart		Communication	2		
	Shut Down		Web Design Video	;		



X

If Custom Shop Palette Wizard window appears, click **Cancel** to exit the wizard, which is not used by YSoA.

cts : 0	.,	0 0	• x: 0.	10 시	₩ 0.00	관 1	00.00	-1% -	0.0	0.00	÷10 I															
,1.230	))	• •	<sub>е</sub> <sup>е</sup> у: О.	00	I 0.00	1	00.00				-	P														
<b>8</b> 6.	<b>6</b>	) līši k	i (il 🗐																							
୍କା		5																								
sø ∰ 	ି କ୍ଳା ଭିକ୍ ଚ	4 <	-2	-1	0	1	3	4	5					1	0 1	1	12 1	13	14	15 16	17	18	19	20	21 .	22
					_										_											
												M	ach	ine	Be	d X										
					-							AI	way	/s c	orie	nt v	our	dra	iwe	าต						-
					_							-\/I	= D1		A I I	V ł	o fil	t th	b	.g nd						_
												VI		10/	4LL	. T L	0 III	L LI II		eu.						
					_																			_		+
					_																					_
					_																			_		-
					_																			_		-
					_																					_
	Ma	chir	ie B	ed Y																						
					-																		_	_	_	-
<																								in la		2
Move	Around D	rawing											_				_		_	_	_				1 0/0	_

Import the DXF file:	🖉 Torchmate CA	D - Untitl				
choose File > Import	File Edit Layout A	Arrange Ma				
-	New	Ctrl+N				
	Open	Ctrl+0				
	Zoom Open					
	Save Caus As	Ctrl+5	Import File			? 🗙
	Jave As					
	Import		Look in:   🥯	Scratch (E:)		
	Export	-	🚞 temp		👜 image5.rtf	Ď
	Export Image		📄 Torchmate	e	🔟 profiles1.BAK	ō
	Publish to PDF		🛛 👜 image 1.rtf	:	🚟 profiles1.cdl	õ
			image2.rtf	:	profiles1.dxf	
			image3.rtf	F	國 profiles1.fgc	ō
			image4.rtf	Ŧ	profiles to cut.	d×f [
			<			>
			File name:	profiles to cut.dxf		Import
			Files of type:	All Files (*.*)		▼ Cancel
In the <b>Import File</b> window DXF file to import	w, choose the		🔽 Thick Line	e 🔽 Ma	erge	Preview
Byte mo to import.			📃 🔲 Retain file	e name		
Make sure <b>Thick Line</b> , <b>N</b> <b>Preview</b> options are all c click <b>Import</b> .	<i>llerge</i> , and checked, then		E:\profiles to File size: 492 Created: Tue Last written: Accessed: Ti	i cut.dxf 234 esday, October 21, 2008 Tuesday, October 21, 20 uesday, October 21, 200	08	
			Filter		4	/

In the **DXF Import Options** window:

Under "Import objects as," choose **Polyarcs** for files consisting mainly of curved lines, or **Polygons** for files with mainly straight lines.

Under "Scale options," select the scale of your file.

Other options can be left at defaults.

When finished, click  $\ensuremath{\text{OK}}$  to import the file.

NOTE: Keep in mind that polyarc profiles require more moves, making them more difficult to cut, and will therefore cause more problems than polygon profiles.

If you have an intricate design with a lot of polyarc cuts you will have better luck on the FLOW JET

DXF Import Options	
Import objects as     Polyarcs     Polygons	Circle options C 2 arcs of 180 C 4 arcs of 90
DXF layer color options C Use layer color only Use dxf layer name to creat C Sheet layers	<ul> <li>Use layer color with override</li> <li>Named colors</li> </ul>
Scale options inches mm cm	POINT entity radius 0.1
Shapes Close shapes Remove duplicates	Gap Tolerance 1st pass 0.001
Notes	Do not show this dialog in the future 🗖 🛛 OK

Your profiles should be loaded. Verify that there were no translation errors and everything looks correct.

# Closing gaps and correcting line direction in the CAD Module.

Highlight the entire shape with your mouse by clicking on the arrow icon and dragging a box around the shape.

#### Next select Arrange > Make Path



Zoom in on one of the breaks between the lines. Next, double click on one of the lines to be joined. Small blue nodes will appear on the ends of the two lines as shown below. Disregard the small window that appears.



Move your cursor over one of the two blue nodes until your cursor becomes a small cross. Left click on it and it will become red. Now press the shift key, and left click on the other blue node. They will now both be red.



Move your cursor away a short distance and hold the right mouse button down. The small icon group shown below will appear. Move the cursor over the top center icon while the right button is down. Release the right mouse button, and the two line segments will be joined.



Perform the same operation with each gap in your shape. When all are closed, you will have one continuous shape.

The direction the lines were originally drawn in AutoCad will no longer matter.

Apply separately

0K

Cancel



# Verify you are not Grouping the profiles with the tool path.



Verify that "Group tool paths with originals" is **UNCHECKED.** If this is set incorrectly the plasma cutter will cut your profiles twice.

NOTE: If your profile is positioned horizontally it will not path correctly on the machine bed, which is oriented vertically.

Select your profiles by drawing a selection box around the objects and choose: **Arrange > Transform > Rotate** 

# Creating the Tool path:

Select your profiles by drawing a selection box around the objects.

#### Click on Machine > Create Tool Path > Online, Male, or Female

If you are cutting a piece with interior holes, choose **Female**. If cutting a single solid piece with no interior holes, choose **Male**. If you are cutting a piece drawn without considering the male/female problem and with no lead in/out then use **Online**.

NOTE: Simple **Female** tool paths will generally compensate for interior holes within a larger exterior cut and place the lead in/out correctly. When cutting more elaborate pieces it is advisable to create two seperate drawings for male and female paths. Each drawing should have a registration dot or line in the lower left corner to accurately line up the dual paths on the machine.

Machine	Transform	Image	View	Opt	ions	Help
Outpul Machir Tool O Tiling	t iing Defaults ptions		Alt+P		100.0 100.0	0
Tool Li Materi	brary al Library					
Use Ea Cut Te	sy Templates mplate Wizar	; d			16	<u></u>
Create	e Tool Path			Þ	On	line
Apply 1	Tool Path			F	Ma	le
Delete Edit To Conve Edit St Job Ar	Tool Path ool Path rt to Online T art Point palysis	ool Path			Fer	

# Setting the Tool path Options:

You will be using only the **Basic Cut** and **Lead in/Out** tabs to start with. The **Template** tab is used for advanced features that YSoA does not use.

Male: Default Templa	te				
Template 🗸 Basic C	ut Lead In/Ou	t]			
Material					
Material Library	Default Material Default Material Default Material Default Material	1/4in 1/2in 1/4in 3in			
Template					
🔽 Filter by Material					
Color					
Delete					
🔲 Auto Make Path					
Lock Template	Add	Change	OK.	Cancel	Help

#### In the Basic Cut tab:

A dialogue box with three tabs now appears. Under the tab **Basic Cut**, select plasma as the tool.

The default kerf width for plasma is .059".

Note that if plasma is selected as the tool, the kerf allowance will be automatic. There is no need to specify an offset in this dialogue box.

Select **Climbing** as your direction for outside cuts, and for outside cuts with interior holes. The torch will automatically travel clockwise for the outside cuts, and counter-clockwise for interior cuts. Select **Conventional** when you are only making interior cuts. Ignore the four small window settings at the bottom, as they are not used in our application.

Male: Default Template		
Template V Basic Cut	Lead In/Out   Tool  Plasma	<b>.</b>
- Direction		
Conventi	onal C	Climbing 💿 💽 🔶
– Tool Path Cornering Si	harp 🖲 🔳	Roll 🔿 🕞
– Move Control Feed rate: Assist	0.00 × in/s	Power 0 × % Dwell time: 0.000 × s
Lock Template	Add Change	OK Cancel Help

#### In the Lead In/Out tab:

# Under Control, check Create Lead In and Create Lead Out.

You can specify whether you want these to be lines or arcs. You can set the radius for arcs, and the length and angle for lines. Arcs can provide a smoother transition from to and from the shape. Ignore the overlap setting for now.

Use any length that accounts for the default kerf width of .059".

Now click the **OK** button to generate your tool path.

Male: Default Template	$\overline{\mathbf{X}}$
Template 🗸 Basic Cut 🗸 Lead In/Out	
Control	🔽 Create Lead Out
Lead in C Arc C Line Length 0.25 in A Angle: 80 A	Lead Out C Arc C Line Length 0.25 in x Angle: 80 x Overlap: 0.00 in x Adjust start point
Lock Template Add Chang	e OK Cancel Help

# **Editing Tool Paths in SMARTCUT CAD**

Your tool path will be generated; it should look just like your part, with the addition of the lead-ins and lead-outs that the plasma cutter will use in cutting.

NOTE: that the interior cut tool path is to the inside, and the lead-in and lead-out are to the inside. The outside cut tool path is to the outside, as are the lead-in and lead-out. You must have a check in the **View** pull down menu next to **Show Tool Paths** before the tool path will be visible. You will now delete the original shape by removing the check, deleting the shape, and replacing the check, leaving only the tool path.

Check View Tool Diameter if you want to see the kerf width.



## Nesting of parts:

Once a completed tool path has been created for your shape, you can duplicate it into any number of copies.



To automatically nest the parts, highlight all of them with your mouse. From the pull-down menu bar, select **Arrange/Nesting**. The nesting window, shown below, will open. There are two kinds of nesting, box nesting, and full nesting. In box nesting, an imaginary box encloses each shape, and these boxes are nested. Each part does not enter the space of another. In full nesting, the shapes are interleaved with each other. The Torchmate CAD Module does both kinds. After making your settings in the various boxes, click **OK**. It may take several minutes for the parts to be nested, depending on your computer.





Completed nesting pattern

## Sequencing of cuts:

Your last step before exporting your completed work as a DXF file, is to specify the order in which your shapes are to be cut. The program is smart, and already knows that the inside of each shape is to be cut first. You have only to set the order for all the shapes to be cut.

Move your cursor over the first shape you want cut until the cursor becomes a set of cross-hairs. Then left click your mouse to highlight it. From the pull-down menu, select Layout/Start Sequence. A blue line will then be Attached to that shape. In the order you want the shapes cut, move the cursor over each suc-cessive shape, left clicking your mouse on the shape when the cursor becomes a set of cross-hairs. You will see the blue line attach itself to each shape as you go, as shown at below.





When you are finished, left click your mouse on the Arrow Select Tool, at the top of the tool bar on the left of the screen. The blue lines will disappear, and the shapes will be sequenced. Now export your shape(s) as a DXF file, and import them into the Torchmate driver software for cutting.

The Torchmate CAD Module has many additional features, including full drawing capability. It will take some time to master them all. However, this chart should get you started.

When you're done, select **File > Export**.

# **Exporting your Tool paths:**

In the **Export File** window that opens, type a name for your exported tool path. For "Save as type," select **DXF (Polyline)**. Then click **Save**.

🐓 Torchmate CAD - Un	titled1.CDL
File Edit Layout Arrange	Machine Transform Image View Options Help
Export File	?≈≈ ↔ 0.00 € ∞
Save in: 🥪 Scratch (E:)	
Torchmate Torchmate or profiles 1.dxf or profiles to cut.dxf or test2.dxf or test3.dxf	Image: State
File name: cutting file Save as type: DXF(Polyline	Save Save Cancel Selected Cancel Cancel Cancel Cancel
EN	

# Cutting the Job: Starting the Machine, Loading a File, and Running the Machine

## Place and Ground your Material

Place the material you wish to cut on the SMARTCUT bed.

Make sure the grounding clamp is attached to a slat on the cutter bed in contact with your material. If your material is not grounded, it won't cut correctly.

## Power on the Torchmate

Turn on the switches on the front of the Torchmate power supply and signal generator in the computer cabinet. The system will do a check on the Torchmate computer system; lights 1-5 will light green, and if everything is working, the orange USB light on the signal generator will stay lit.



Signal Generator & Power Supply



Turn on the plasma cutter which is at the foot of the Torchmate bed.



Grounding Clamp



**Plasma Cutter** 

Turn on the Torch Height Control Module using the power switch on the rear left of the box. The screen on the control module will flash a message that cutting is disabled, and the should have the following:

Current Voltage: 0 Set Voltage: Will be a number set by the previous operator Auto Press Ok to Change

**Torch Height Control Module** 

# Launch Torchmate 3

Torchmate 3 is the software used to control the plasma cutter. To launch it, click the Torchmate 3 shortcut on the workstation desktop or select **Start > Programs > Fabrication > Torchmate 3 > Torchmate 3** 

If the plasma cutter was powered on correctly, you'll see the Welcome screen asking to establish a connection to the Signal Generator. Click **Yes, Start Online** to continue.



You will see a **Safety First** screen warning you that **the plasma cutter is now online, and all moves made by the Torchmate software will be executed by the tool**. Keep this in mind, and use caution when adjusting your material with the cutter online.

Safety First!								
Safety First!								
The controller is now online. ALL MOVES WILL BE EXECUTED BY THE MACHINE TOOL.								
A computer controlled machine tool is potentially dangerous. You are responsible for the safety of yourself and others during the operation of your CNC machine. For safe use of this software and your CNC machine, you must follow these safety guidelines:								
1. Never let the machine tool run unattended.	1							
2. Require any person in the same room as a running machine tool to wear safety goggles, and to stay a safe distance from the machine.								
3. Allow only trained operators to run the machine tool. Any operator must have:								
Knowledge of machine tool operation.								
If you do not understand and agree with all of the above safety guidelines, please choose Go Offline below.								
<u>G</u> o Offline <u>A</u> gree								

## Load your Tool Path

First, you need to load your tool path. From the File menu in the Torchmate window, choose Import DXF...

🖉 Torchmate - No G-Code File Open - ya	letorch.STP - Torchmate.tl	B	2
Eile Edit Configuration Controller View Coor	dinates Favorites <u>H</u> elp		
New G-Code Ctrl+N Open G-Code Ctrl+O Close G-Code		Program <u>Set</u>	Machine <u></u> 0 X 0.0000
New Setup Open Setup Save Setup		Y 0.000	Y 0.0000
Save Setup As		Relative Set	Dist To Go
New Tooling Open Tooling Save Tooling Save Tooling As		X 0.0000 Y 0.0000	x 0.0000 y 0.0000
Editor Ctrl+E			
Import DXF Ctrl+I		Tool: None	Length
1 E:\test-new-CAD14-01.fgc 2 E:\test-new02.fgc 3 E:\test-new01.fgc		Offset: X 0.0000 Comp: Positive (G43	Y 0.0000 Z 0.0000 ) ▼ Diam: 0.0000
4 C:\Torchmate Data\G-Code\Tutor.fgc	60 80	x	Inline
Exit	G-Code	Jog Point Home A	ux MDI Probe
	<u>v</u>	Y + X - X + Y -	C Fast       © Contin         C Med       © 0.1         © Slow       © 0.01         Hold       © Step

In the Open window, browse to the folder you created for your project on the Scratch Disk E: and select the dxf file you exported from the Torchmate CAD software (NOT your original dxf file that you exported from AutoCAD, unless you overwrote that file with the one from Torchmate CAD) and click **Open** 

You'll then be prompted to save your machine file. Choose the folder you created for your project, give the file a name, and click **Save**.

	Configuration		
In the <b>Configuration</b> window that opens, enter the following settings:	Import	Import General Scale: 1.0000 C Optimize Toolpath	Feedrates XY Feedrate: 150.00 (in/min)
Scale: 1.0		Decimals: 4 C Line Numbers Join Tol.: 0.00100 (inches)	Plunge Feedrate: 5.00 (in/min)
XY Feedrate: 70.00 - 150.00		Z Axis Coordinates	Circles
NOTE: Feedrate can be changed later, after determining optimal rates		Image: Promittie       Fool Up:       0.2500       (inches)         Incremental Depth of Cut:       0.1000       (inches)         Final Tool Down (Milling):       0.1000       (inches)         Final Tool Down (Holes):       0.1000       (inches)	Ellipses and Splines Chord Error: 0.01000 (in)
through line-speed tests.		- Program Zero Location	Additional G-Code
Other settings can be left at defaults.		C X, Y of Import File:         X:         11.0000         (inches)           Y:         -8.5000         (inches)	Edit File End: Feature Start (Before Rapid): Feature Start (After Rapid):
<b>OK</b> to continue.	OK Cancel	C Lower Left of Toolpath	Feature End:

# Zero the Program

Your tool path should not appear in the Torchmate window. You now need to move the plasma cutter head to the home position of your tool path, which is the lower left corner of your part.



Move the cutter head to where the home position of your tool path should be on your material by clicking on the **Jog** button (located below the bottom right corner of the tool path window) and using the X and Y buttons. You can adjust the speed of the tool head movement by clicking **Fast**, **Med**, or **Slow**. For most purposes Fast is most useful.

Once the tool head is at the home position of your tool path, click the **Set** button next to **Program** and choose **Zero All**. The coordinates in the Program and Relative windows should now all read 0.0000, which means the cutter head is at the origin of your tool path. You're now ready to cut.

NOTE: If you would like to have the ability to change program zero anytime in the middle of production, without losing the original zero, then set Relative Zero in the beginning as well.

## Sending the Cutter to "Home"

With the Torchate the concept "going home" is achieved by sending the cutter to Program Zero/Relative Zero. The machine understands this as sending the cutter to a point on the bed. Click on the **Point** button, which is to the left of the **Jog** button, and select "go to program zero."

This will send the cutter to the original Program Zero position, or "Home."

# Activate the Plasma Cutter

To activate the cutter, click and hold the Cut button on the Torch Height Control Module (the black box with the blue LCD screen). The "Cut Disabled" message will disappear and the red **Cut On** light will activate.\*

## \*Put on the welding goggles.

Flash from plasma cutter could damage your vision, even if viewed peripherally.

# Adjust Voltage

Voltage controls the distance between the torch and cutting material. This distance is different for each type of material blank. Once set, the voltage will allow the Torch Height Controller to automatically maintain the distance during operation. This setting is determined by running a line-speed test file. In Torchmate 3, under file, open g-code and navigate to the Scratch Disk **E: > Test Files**.

## There are three different test files:

*Line\_Speed\_190-130.fgc* : For aluminum sheet and thinner gauge steels *Line\_Speed\_130-10.fgc* : For most gauges of steel and thicker gauge aluminum *Line\_Speed\_70-10.fgc* : For thicker gauge steels

Once the test begins cutting, the "Current Voltage" reading will fluctuate widely and settle on a median Voltage for each line. By tracking each median voltage during a line-speed test the operator can gauge an ideal Voltage for each line-speed.

## Press Ok to Change:

This will allow you to use the "up" and "down" buttons to adjust to the voltage settings determined from the line-speed test. Once correct voltage is selected, press Ok again to set.

NOTE: If you press up/down before first pressing Ok, you will navigate to other settings menus. If this happens, just keep going until you return to the voltage setting screen.

## **Amperage Current Control**

# Adjust Amperage

Amperage controls the power to the plasma cutter. Lower amperage for thinner gauge materials, higher amperage for thicker gauge materials. The amperage control on the Plasma Cutter can be adjusted between cuts.

## Two simple ways to judge if your amperage settings are correct:

Too High = If the greater density of sparks hits the floor or beyond. Too Low = If the material is not being cut all the way through.

# **Set Function**

The Plasma Cutter has three function settings that we use:



**SET:** Used to purge air through the torch to in order to adjust gas pressure.



RUN: Used for most cutting operations



RAPID AUTO RESTART: Used for perforation type cuts

**Function Control** 





866-571-1066

ORCHMATE

**Torch Height Control Module** 

# **Start Your Job**

When you're ready, click the **Start** button in the Torchmate window to begin cutting.

The Torchmate window will show the progress of your job as it cuts. Blue lines show tool moves, including cuts, that have already been made, and red lines show the moves that have yet to be made.



#### **Check the Cutting Amperage**

As the Torchmate cuts your job, observe to see if your amperage is correct. If the amperage is too low, the cutter will not be cutting fully through your material. If your amperage is too high, you'll see a high density of sparks cascading onto the floor below the material (some sparks are normal). If you're unsure about the amperage, see one of the Fabrication Lab Faculty to help you.

#### **Pausing Your Job**

To pause your job, press any key or click **Feed Hold** to stop the machine. Do the same to resume cutting.

G-Code Jog	Point Home	Aux MDI Probe
Jump To Line	Faster	Program Feedrate: 150.0
Cance <u>l</u> <u>G</u> o	Slower	100 👻 % Override: 150.0
Run Time: 00:01:46 To Go: 00:01:02	<u>R</u> eset	C <u>S</u> tep © Conti <u>n</u> uous
	St <u>a</u> rt	Feed Hold (Any Key)

## In Case of Misfires

Misfires can occasionally occur if the electrical connection between the cutter head and the material is broken. If a misfire occurs, you can press any key to pause the cut, then click the **Jump To Line** button in the Torchmate window to allow you to select a line of your cutting program from before the misfire occurred.

Select the line you want to go to and click **Go**. In the "Jump to Line Completed" window that opens, click **Move Now** to move the cutter head to the previous position. Then resume cutting. You will see the red and blue lines in the cut window update to reflect the current cut status.



## To Change the Cutting Speed

If you need to adjust the cutter speed for better cutting, hit any key to pause cutting. Under Program Feed rate, choose a % override rate to either speed up or slow down your cutting job.

REMEMBER TO USE YOUR COMMEN SENSE:

Wear welding goggles when operating the Torchmate Consider fire hazards around the cutting bed Never leave the Torchmate unattended during a job Clean up the slag left over by your job Follow the Safety First guidelines you have agreed to

Failure to follow these guidelines will result in a loss of Fabrication Labs access