How to download a shape file to a Programmable Shape Machine using PC Shapes Software

1. Open PC Shapes Program. Click File Open to find the shape you want to download to the machine.

BUG-O SHAPE TOOLS				
File MacroEdit BUGO St	uff Sketch It Pipe Functions	Options Help		
New				
Oren				
Savě				
Save As Clear				
Ciear				
Open Job				
Export CNC				
Di cup				
Print SHP				
Exit				
Shape No. 0	COM4 Uni	its: mm	XY Flat Shapes	

2. Click on the file you want to open from your computer and click Open. If you are working with a .cnc file, make sure to change the file type at the bottom of the screen to .cnc.

BUG-O SHAPE TOOLS					Ξ Σ	3
File MacroEdit BUGO Stuff	Sketch It Pipe Functions Options Hel	p				
🗇 Get File						x
Computer	r ▶ TI106268W0D (C:) ▶ SHAPE ▶		- - ↓	Search SHAPE		٩
Organize 🔻 New folde	ſ			!≡ ▼		2
🔆 Favorites	Name	Date modified	Туре	Size		
🧮 Desktop	\mu Arc	3/19/2012 2:03 PM	File folder			
🗼 Downloads	퉬 Shape Files	3/19/2012 2:03 PM	File folder			
Recent Places	simple.shp	2/17/2009 8:38 AM	Shape File	1 KB		
□ Libraries □ Documents □ Music □ Pictures □ Videos □ Computer ▲ TI106268W0D (C:) ♥ Network						
File na	me: simple.shp		•	Shape files (*.shp) Shape files (*.shp) CNC Files (*.cn All files (*.*)		
Shape No. 0	COM4 Units: mm	XY Flat Sh	apes			

3. Make sure Shapes program is set for proper units (in/mm) for the shape you are loading. You can change the units by clicking Options Units.



4. Next, check to make sure you are communicating with the machine. Plug the cable into the programmable shape machine and the computer you are working with. Click BUG-O Stuff Comm Terminal.



5. The Comm Terminal will open a new window. If you are using the correct Comm port, when you press the black reset button you should see Reset OK on the Comm Terminal. This tells you that you are talking to the machine properly.

🔿 BUG-O SHAPE TOOLS :: E-FOOT O	PENING (1).shp		- • ×
File MacroEdit BUGO Stuff Ske	BUGOTALK - Com Terminal: Escape to Exit		
SEGMENT LIST	Characters Beceived from BUG-0		*
Add Edit Insert Delete	Reset OK		
Seg TyQd Data			
$ \begin{array}{cccccccccccccccccccccccccccccccccccc$		50.0	
Shape No. 0			

6. If you DO NOT see Reset OK on the Comm Terminal when you press the black reset button on the machine, you will need to change the comm port. Click Options Comm Port to select the proper port. The software should tell you which ports are valid. Click on a valid comm port and click ok. Go back to BUG-O Stuff Comm Terminal and press reset on the machine. If you see Reset ok, move to step 7. If not, repeat step 6.



7. Now we are ready to download the file to the machine. Click BUG-O Stuff Download. You will get a window that tells you to press reset on machine then click ok. Press black reset button on machine then click ok. When the download is done, you should see a window that says Sent. Click OK and the machine should be ready to run.



How to convert a .dxf file through DeskCNC:

Installation of DeskCNC:

- 1. Insert CD labeled BUG-6241 into computer
- 2. Run DeskCNC.exe from the CD
- 3. Enter the user name and key code written on the CD when prompted the first time you run DeskCNC.
- 4. Copy the file "posts.cfg" from the CD to the following location on your computer:

My documents\DeskCNC\config

- Overwrite the file that is currently in that location with the file that came on the CD. This will allow the DeskCNC conversion file to be read by the BUG-O PC option

Requirements for .dxf file are as follows:

- 1. Drawing MUST be only 2D
- 2. Drawing must be cleaned to only show the actual cut lines to be cut. All reference marks and extra lines must be removed. This can be done in DeskCNC or in the original CAD drawing.
- 3. All blocks and Splines must be exploded completely before loading the .dxf file on DeskCNC

How to configure DeskCNC:

***This MUST be done every time you first open the program to set the configuration properly**

1 . Click the setup tab and click post processor setup

<u>ak</u>	DeskC	NC															×
File	Mode	Edi	t DXF	Setup	Sele	ct Vi	ew Ti	polpath	5 Hel	р							
				Ор	tions												_
				Pos	st Prec	essor :	Setup										
Machine Setup																	
				Too	ol Libra	ry											
				Wo	irk Coc	rdinati	e Offse	ets									
				Mo	dify Ac	cel Ra	mp										
									_								
								· · · · ·									' I
																	_
Command Timed Out G 0									то		0%	1	.0446,	6.8242	2		

2. On the general tab, click the dropdown name box and click BUGO. Click Edit to adjust the settings for the program, then click cancel to go back to the main program.

	Post Processor Setup	
	General Tokens Movement Script Header / Footer Before Move	-
	Hame Income	
	File Extension ISO G-Code - Non Modal	
RR_	Plasma	
	Comments Laser	
	Begin Comment 2D	
	End Comment Cl BUGO	
	BUGOm	
	Include Comments	
CC	Maximum Block Line Number	
	Number Start At U	
	Number Increment 1	

How to convert a .dxf file to a CNC file in DeskCNC:

1. Open .dxf file in DeskCNC

📥 DeskCNC												
File Mode Edit DXF	Setup	Select	View	Toolpaths	Help							
Open DXF Open STL Open HPGL Open Point Cloud	•			1 1		1						
Open Gerber Open Image	;+		_									
Open NC Digitize Foam Cutter												
Save Geometry DXF Save Toolpaths CNC Save Toolpath DXF												
Add Text			+									
View Log File	-11		+									
Close Exit												
Command Timed Out		GO			то			0%	-8	3.9333,	6.788	3

2. If using a clean drawing, click select menu and click select all.



3. Click the edit DXF menu and click scale, move, rotate selected.

📥 DeskCNC - DXF Mode [5-2-11 AutoCAD drawing	for Plasma Cutter	.dxf]	
File Mode Edit DXF Setup Select View Toolpaths Help)		
Delete Selected Regions Reverse Dir of Selected Regions	,		
Change First Entity			
Set Machining Order	{		
Mirror Selected X Mirror Selected Y			
Explode Selected Regions			
Join Selected Regions			
Change Depths			
Sca Move, Rotate Selected			
1 Boundaries 0 Islands Selected G 12	то	0%	70.6135, 36.8187

4. Click move to origin to move the piece to the center point of the grid. Then click the cancel button. At this point you will likely not see anything on your screen.

Selection Parameters Bottom Left X 70.1323 Bottem Left Y 24.4437	Enter New Paramete Bottom Left X	70.1323	
Center X 75.6323 Center Y 30.2875 Top Right X 81.1323 Top Right Y 36.1312	Scale Factor Rotation Angle Offset	0.00000 0.00 0.00	
Scale Rotate	Move Move	Origin	
Move Move By Offset	Cano	cel	



5. Click the view menu and click zoom all. This will bring the full drawing back into view.

6. Click on view menu and make sure that Show Directions and Information tags are both checked to determine number and direction of cuts.



7. To change the direction of the cut, make sure your drawing is still selected by clicking select and select all. Then click Edit DXF then Reverse Dir of Selected Regions.

<mark>2</mark> 61	DeskCN	IC - DXF	Mode [[5-2-11	Auto	CAD drav	ving	for l	Plasr	na C	utter.dx	ĵ]	(
File	Mode	Edit DXF	Setup	Select	View	Toolpaths	He	lp						
		Delete	Selecte	d Region	s									
		Revers	se Dir of	Selected	Regior	15 -								
		Chang	e 'Fyrst E	ntity					1					
		Set Ma	achining	Order					- {					
		Mirror	Selected	łх					- {					
		Mirror	Selected	ΥĽ										
		Exploc	le Select	ed Regio	ns									
		Join Se	elected F	Regions										
		Chang	e Depth:	s										
		Set W	all Profile	es										
		Scale,	Move, R	lotate Se	lected					Ì				
										+				
+		+			_					4				
					+					<u> </u>				
		+ $+$	_ 	│			\rightarrow							
			1	1			l í							
1 Bou	undaries	0 Islands :	5elect G	12			ΤO				0%	-1	.9084, 12	2.4415

8. Once you have the cuts set the way you want, you are ready to make toolpaths for the machine to read. Click Select menu and select all (If the shape is no longer highlighted), then click Toolpaths menu and Contour.



9. In the contour menu, there are a few things that need to be changed. Under machining parameters, the contour depth should be 0.0001. This must be done in order for DeskCNC to create a toolpath. The following parameters deal with kerf for plasma cutting as well as lead in and exit.

Assuming zero kerf and no lead in or exit, the following settings should be applied:

Under Roughing Parameters, the Tool Diameter should be 0.0001. Under Compensation, make sure None is checked. Under Entry, make sure None is checked. Under Exit, make sure None is checked.

After all the settings are correct, click the contour button to generate your toolpaths.

DeskCNC - DXF Mode [5-2-11 Auto File Mode Edit DXF Setup Select View	CAD drawing for Plasma Cutter Toolpaths Help	. dxf] 📃 🗖 🗙
Contour Machining Parameters Contour Depth 0.0001 Rapid Height 1 Coolant Off v	Finish Pass Use Finishing Pass Finishing Parameters Tool	Entry None Tangenl Perpendicular Arc Length 0
Spindle On Delay 1 Roughing Parameters Tool Tool Tool Oliameter 0.0001 Z Step Size 1	Tool Diameter 1 Z Step Size 1 Spindle RPM 10000 Feedrate 10	Exit None Tangent Perpendicular Arc Length 0
Stock To Leave 0 Spindle RPM 10000 Feedrate 10 Plunge Rate 10	Compensation None Cut Out Right C Left	Contour Load Cancel
1 Boundaries 0 Islands Selected G 1	12 T 0	0% 3.3231, 11.9405

10. Once you have created your Toolpaths through the contour menu, you are ready to save the file in a format to load in the PC option. Click the file menu and click Save Toolpaths CNC. Save the file somewhere that you can easily get to later.

**For the purpose of using this file in the PC option, the easiest place to save the file is in your Shape directory on your computer which by default is C:\Shape.



11. Now you are ready to open your CNC file in the PC option. Once you have it loaded, the PC option can be used normally to download. The only thing different you have to do is change the file type to be opened to CNC files and then open your file. The PC option will build your program automatically from there.

Cutting Pipes with DeskCNC and Shape using sample.dxf

Installation:

- 1. Run DeskCNC.exe
- 2. Copy posts.cfg to C:\Program Files\DeskCNC, overwriting the old file
- 3. Run the setup file for the Shape program

Prepare your DXF

- Remove lines that are not part of the shape (I left the top and bottom lines to show how to remove them in DeskCNC)
- Make sure dimensions are 1:1



Configure DeskCNC.

- 1. Machine Setup
 - a. Click Setup->Machine Setup
 - b. Click DeskCNC Setup
 - c. Set the Units to match your units
 - d. Click Save
- 2. Post Processor Setup. This must be done every time the program is started
 - a. Click Setup->Post Processor Setup

- b. Click General
- c. Click the Name dropdown and choose BUGO
- d. Click Edit to load the configuration
- e. Click Cancel

Load the DXF

- 1. Click File, Open DXF
- 2. Choose

sample.dxf

<mark>de</mark> De	skCNO	- DXF Mo	ode [sa	mple.d>	ɗ]				
File	Mode	Edit DXF	Setup	Select	View	Toolpaths	Help		
-									
1 Regio	ons Cre	ated 0 G	G 4			то		0%	750.1916, -142.5558

Edit the shape

You will have to make changes to the loaded DXF file to tell the machine how to cut it. At the minimum, you will have to move it to or below the origin and rotate it properly. The select menu is at the top of the screen, but can also be opened by right clicking on the drawing.

- 1. Remove any extra lines
 - a. Select->Select all
 - b. Edit DXF->Explode Selected Regions
 - c. Select->Select Single
 - d. Click each extra line that must be removed
 - e. Edit->Delete Selected Regions
 - f. Click OK
 - g. Select->Select All
 - h. Edit DXF->Join Selected Regions

ا 🛓	eskCN(C - DXF Mo	ode [sa	mple.d>	đ]				
File	Mode	Edit DXF	Setup	Select	View	Toolpaths	Help		
-									
Click	on Regio	on to select	. G 2			т о		0%	393.5824, 432.2526

- 2. Move and Rotate the Shape. The shape to cut must be vertical and at (below) the origin. If it is not, it must be rotated and moved.
 - a. Select->Select All
 - b. Edit DXF->Scale, Move, Rotate Selected
 - c. Enter 90 in Rotation Angle to rotate left, -90 to rotate right.
 - d. Click Rotate
 - e. Click Move To Origin. If you leave the shape like this, the track will have to start close to the machine, move away from it to the top of the shape, then begin cutting. To make this faster, you want to move the shape below the X axis.
 - i. Take the number displayed next to Top Right Y. Enter the negative of this number in Bottom Left Y.

ii. Click

Move							
Move, Scale	Move, Scale, and Rotate						
Selection Parameters			Enter New Parameters				
Botton	n Left X	0.0000	Bottom Left X	0.0000			
Botten	Bottem Left Y Center X		Bottom Left Y	-850			
Ce			Scale Factor	0.00000			
Ce	enter Y	425.0000	Rotation Angle	-90.00			
Top	Right X	279.2562	Offset	0.00			
Top	Top Right Y 850.000		-Rotation Axis	0.00			
			OX OY	⊙ z			
	Scale						
Rotate			Move To Origin				
Move			Center On Origin				
Move By Offset			Cancel				

- iii. Top Right Y should now display 0
- f. Click View->Zoom All



Setup Cutting Paths

- 1. Enable path info display
 - a. View->Show Directions
 - b. View->Information Tags





Picture the MM1 at the bottom left of the image. The left side of the image is the starting point for rotation. This shows that the tractor will move from its origin toward the machine until it reaches point 1. The arc will turn on at point 1, cut the pipe near the machine, turn off, travel to point 2 at the end of the pipe, turn on, and then cut while turning in the opposite direction. This won't work because the pipe will be cut and unable to rotate. You must change the direction and order of the cuts

- 1. Change the order of the cuts
 - a. Edit DXF->Change Machining Order
 - b. Click Ok
 - c. Click each cut in the correct order
- 2. Change the direction of the cuts
 - a. Select->Select Single
 - b. Choose each cut that needs reversed
 - c. Edit DXF->Reverse Dir of Selected Regions

Now the machine will start from its origin at point 1, make the top cut, then move to the bottom for cut 2. It is ready to be turned into a toolpath.



Creating Toolpaths

You must create the toolpath and save the resulting cnc file that will tell the machine how to move

- 1. Select->Select All
- 2. Toolpaths->Contour
- 3. Change Contour Depth to .0001
- 4. Change Tool Diameter to .0001
- 5. Change Compensation to Left

6. Click

Contour

Machining Parameters	Finish Pass	Entry					
Contour Depth 0.0001	Use Finishing Pass	💿 None 🔿 Tangent					
Rapid Height 1	Finishing Parameters	C Perpendicular C Arc					
Coolant Off 💌	Tool	Length 0					
Spindle On Delay 1	Tool Diameter 1						
Roughing Parameters	Z Step Size 1	Exit					
Tool (4)	Spindle BPM 10000	C Respondicular C Ara					
Tool Diameter 0.0001							
Z Step Size 1	Feedrate 10	Length 0					
Stock To Leave 0	Configuration						
Spindle RPM 10000	○ None ○ Cut Out None	Contour					
Feedrate 10	C Right C Left Load	Cancel					
Plunge Rate 10	Save	Delete					

- 7. Click File->Save Toolpaths CNC.
- 8. Make sure the Save as type is BUGO
- 9. Save the file. It can now be imported into the Shape program.

Importing a shape

You must import the shape into the Shape program before it can be cut.

- 1. Start the Shape program
- 2. File->Open
- 3. Change Files of type to CNC Files
- 4. Find and open the CNC file you created in DeskCNC



You can now work with the Shape program like normal

LIT-DESKCNC-TECH-1112