MadCam 5.0: 2D Profile Toolpath

Digital Media Tutorial Written By John Eberhart

MadCAM can create toolpaths to mill two dimensional profiles in a range of material thicknesses. This tutorial goes through the entire process of setting up your file and creating a toolpath for the large CNC Mill.

Step 1: Open or Create a 2D file in Rhino. MadCAM runs inside Rhino, and is automatically opened when Rhino is opened. (If the MadCAM toolbar does not open, see the next page for instructions on loading the toolbar).

Step 2: Prepare Model. Correctly place your part in the modeling window. The large CNC mill uses the model origin location as the start point for the mill. You need to move your part so that it is completely in the positive X and Y axis. **NOTE: Model in Rhino MUST be scaled to match the actual part to be milled.**





Tools Analyze Render Panels He



The first time you launch Rhino, the Madcam Plugin May have to be loaded. To load the MadCAM plugin, choose Tools>Options. The following window will appear.



If the MadCAM toolbar is also not always visible, it can be loaded by either typing in a MadCam command like **MC_Options or MC_SelectModel**:



Or by selecting Tools>Toolbar Layout.

Find MadCAM5 by following the path: C:\Program Files\madcam 5.0 (64-Bit)\UI\ madcam.rui



Organize 🔻 New fold	er			8== 👻	
🔶 Favorites	Name	Date modified	Туре	Size	
🧮 Desktop	퉬 System	12/20/2012 3:02 PM	File folder		
🚺 Downloads	🎉 Temp	12/20/2012 3:02 PM	File folder		
Cropbox	📄 default.rui	12/20/2012 3:02 PM	RUI File	8,884 KB	
Music					
Computer					
Videos					

Once you choose the MadCAM toolbar file, it will take you to this window. Click **Choose All** and then click **OK**. You have now loaded the Madcam toolbars.

Step 3: Load the geometry into MadCAM. Click on the **Surfaces** icon, and select the 2D surface to load. Follow the prompt box when loading the object.



The loaded object will have a box around it.



Step 4:Create and load a cutting tool. Click the Create Cutter button to create a new cutter and or load an existing cutter.



Step 5: Cutter parameter window: Load a predefined cutter based on the material you want to cut by selecting it from the right side. This will give you a cutter with predefined Feed and Spindle Speed rates.

Name: Give a unique name.

Type of Bit:

Choose the bit that best represents your bit shape.

Diameter: Should be set

Length: Measure while in a collet and input length

Cutting Length: Measure and input length

Tool Number: Set to any number.

Tolerance: Use 0.1 - 0.01, depending on your material. The lower the tolerance, the longer your toolpath will take to process. Setting too low a tolerance can cause the program to run out of memory.

Coolant: Leave unchecked.

If no changes are made, click – **OK** to load the tool.

list of cutters.	
Input the exact length and flute length - measure the bit when it is in a collet.	
Click Save to set the tool lengths once set.	×
Cutter Name: .25 - Flat - Acrylic - Set Vengths	
Cutters Library C:\ProgramData\5XCI/C\madCAM\5.0\Cutters\Cutters_inch Cutter tool Ball End Flat End Comer End Taper Tool holder C 1.5 Diameter 25 Length L Flute Length F	120 - Ball - Acrylic-Set Lengths 121 - Ball - Foam - Set Lengths 122 - Ball - Wood - Set Lengths 125 - Flat - Acrylic - Set Lengths 125 - Flat - Foam - Set Lengths 125 - Flat - Norvic - Set Lengths 25 - Ball - Foam - Set Lengths 25 - Ball - Norvic - Set Lengths 25 - Flat - Youric - Set Lengths 25 - Flat - Foam - Set Lengths 25 - Flat - Norvic - Set Lengths 25 - Flat - Norvic - Set Lengths 25 - Flat - Norvic - Set Lengths 5 - Ball - Norvic - Set Lengths 5 - Ball - Norvic - Set Lengths 5 - Flat - Norvic - Set Lengths 5 - Flat - Foam - Set Lengths 5 - Flat - Novic - Set Lengths 5 - Flat - Novic - Set Lengths 5 - Flat - Wood - Set Lengths
Speed / Settings Colors / Style Other Speed Settings Tool number Feed Z 100 Tolerance .005 Spindle 14000 Coolant OK Cancel	

You can adjust the settings of a tool to fit your particular needs. Do this only if you are comfortable changing the settings.

Setting the Feeds and Spindle Speeds - Rules of thumb:

Feed X,Y - This is how fast the bit is pushed through the material and is based on a number of factors. Fior you tools, it should not be set to **more than 280** inches/min. This speed can be adjusted at the machine

Feed Z - Set this to a max of 100 inches/min as this can be hard on the bit

Spindle Speed - Set to Max of 12,000 rpm (except for metal) this can be adjusted at the machine.

If you want to make your own tool, make changes to the tool settings and change the name of the tool under cutter name:	Click the "Save" icon to save the tool to the libr	
	Cutters	×
	Cutter Name: 25 - Flat - Acrylic - 1.5 long with .75 flute	
	Cutters Library C:\ProgramData\5XCNC\madCAM\5.0\Cutters\Cutters_inch	
	Cutter tool Ball End Rat End Comer End Taper	.125 - Ball - Acrylic-Set Lengths .125 - Ball - Foam - Set Lengths .125 - Ball - Wood - Set Lengths .125 - Flat - Acrylic - Set Lengths
	Tool holder C 1.5 Diameter D .25	.125 - Flat - Foam - Set Lengths .125 - Flat - Wood - Set Lengths .25 - Ball - Acrylic - Set Lengths .25 - Ball - Foam - Set Lengths
	Length L 1.5	.25 - Ball - Wood - Set Lengths .25 - Flat - Acrylic - 1.5 long with .75 flute
Your cutomized tool will be saved to library for use down the road.	Flute Length F .75	25 - Flat - Acrylic - Set Lengths .25 - Flat - Yoan - Set Lengths .25 - Flat - Wood - Set Lengths .5 - Ball - Acrylic - Set Lengths .5 - Ball - Foam - Set Lengths .5 - Flat - Acrylic - Set Lengths .5 - Flat - Acrylic - Set Lengths .5 - Flat - Wood - Set Lengths .5 - Flat - Wood - Set Lengths
	Speed / Settings Colors / Style Other	
	Speed Settings	
	Feed XY 200 Tool number 1 Feed Z 100 Tolerance .005 Spindle 14000 Coolant	
	OK Cancel	
Click OK to make th	at tool active /	

Step 6: Pocketing. In order to pocket cut, your curves must be completely closed. Go to 3P toolbar and select **Pocketing.**

Pocketing is best for removing a depth of material inside a boundary curve. If you want to cut all the way through, you should used profile cutting.

Note: You should perform all pocket cuts **BEFORE** cutting any profile cuts.

Select the closed curves for the pocket cut and hit Enter.



5Xtra

3**1**5 🔰 🦉 🚳 🖂 🗱 🖓

F 🗐 🗟 🔸

5X

Top (1)

3X

1

2

Step 7: Pocket Toolpath Setup:

Material Top: Set to 0 by default.

Material Bottom: The thickness of the material. In this case set this to -.75 (3/4" plywood)

StepDown: How thick of a layer the bit will cut as it mills down the part. Rules of thumb: *Foam: Max StepDown = cutting edge length Wood: Max StepDown = 1/3 diameter of bit Metal: Max StepDown = 1/4 diameter of bit*

Step Over: 2/3 diameter of bit for softer Materials; 1/3 for denser materials and faster feed rates.

Direction: The direction of the passes the bit will make as it cuts.

Climb: Used for cutting wood.

Conventional: Used for monolithic materials.

Safe Clearance: Set to 0.5

Ramp Angle: Set to 10 deg.

Click **OK** to calculate toolpath.





Completed Toolpath.

Step 8: Set up a Profile Curve Toolpath.

Click the Create 3P Toolpath button. Choose Profiling.



The prompt instructs you to select the profile curves and press **Enter**. Note: Try to select curves in clockwise order.

Once you hit Enter, you will be prompted to select the side of the line to cut on and if you want to do a climb or conventional cut.

Climb Cutting-Used for material with a grain like wood and some plastics. Conventional cutting-used for monolithic materials such as metal and MDF



Profiling	
Stock to leave	
* •	٢
Step Size	
Material top	Step Down
	0.125
Z1 Material bottom	Kin Cut
75	0
<u> v</u>	Break Through
	0
SafeClearance Rad	Radius
OK Cano	el Help

Step 9: Profile Toolpath Setup:

Material Top: Set to 0 by default.

Material Bottom: The thickness of the material. In this case we set to -.75 (3/4 plywood)

StepDown: How thick of a layer the bit will cut as it mills down the part. Rules of thumb:

Foam: Max StepDown = cutting length of bit Wood: Max StepDown = 1/3 diameter of bit Metal: Max StepDown = 1/4 diameter of bit

Choose Radius Lead Approach

Safe Clearance: Set to 0.5.

Click **OK** to calculate toolpath.

In this case I joined each profile and created a toolpath for each profile. This makes milling more efficent on the mill as it will cut each profile completely before moving to the next profile.



Step 10: Simulate Cutting Job. Click the Simulate button. The Cut Simulator Window will open.



The buttons at the top of the window are used for controlling view, cut simulation and cut simulation settings.

Press Play to start simulation.

What you see here is what you will get!



Step 11: Post the toolpath to the Yale Post Processors.



Click the **Postprocess** button. The MadCAM Post window will appear.

Step 12: Verify the post processor and cutter library settings are correct.

D	Post Process – 🗆 🗙
File	
Name / Program number	i 🗋 😂 🗙 🛃 📾 📔 🔛 🙀
1	
	Choose a Post Processor for the machine you want to use:
Machine	YSOA Large Mill or YSOA Small Mill.
3-axis_Machine	
Post Processor	Click on the Post Process button :
YSOA Large Mill	For the YSOA Small Mill , you can give it any name and save it to
YSOA Large Mill YSOA Small Mill	your Box account.
Home Position	
X 9 Y 12 Z 0.13	Ear the VSOA Large Mill you need to name the file using at the
	For the YSOA Large Mill , you need to name the file using at the
	most 6 numbers (no letters or spaces). Save the file to a thumb
	drive.
Post Process >>	Note: the thumb drive needs to be formatted with a "fat 32" file
	system in order to be recognized by the mills. See a DM staff
Class	member for assistance with this if needed.
Close	

Click this button for viewing or editing the output file.

	mC	Post Process	_ 🗆 🗙
Click on the Post	File		
Process button to	Name / Program number	i 🗋 📂 🗙 🛃 🛸 🗍	#
write your file.	123456	madCAM report: Created on: 9/25/2015 12:27:50 PM	
	Machine 3-axis_Machine	======= Programmer notes ======== Programmed by: ARCH-PC-402 Maximum Tool Length: Approximate work piece dimensions: L=18; W=2	
	Post Processor YSOA Large Mill	====== Toolpath Specifics ========	
		Tool: T1 (D0.250_r0.000_L1.375) .25 - Flat - F Path: Pocketing	oam
N	Home Position	File successfully written as D:\Box\yale (eberhart Post Processor:C:\ProgramData\5XCNC\madCA	@yale.edu)\teaching resi M\5.0\Machines\Yale\Y
		Tool changes: not specified in post processor Rapid traverses: 25 seconds Cut in material: 4 minutes 2 seconds	
		Total machining time: 4 minutes 27 seconds	Outputted Fi
	Post Process >>		
	Close	<	>