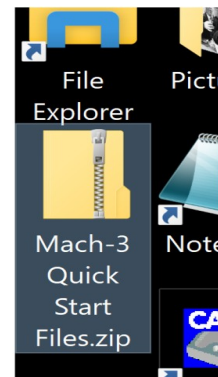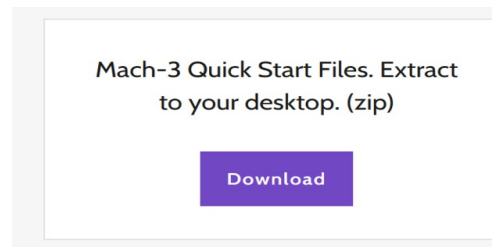# Mach-3 Quick Start guide

 This guide will walk you thru creating a G-code file for a CNC Foam Cutting Machine with Mach-3 or Mach-4 controller software. This guide is for the inexperienced Windows user as well.

   ShopCAM has all the tools needed to create the geometry. (lines, arcs, circles points) as well as editing tools (trim, break, copy, delete, scale etc). Rather than walking you thru the geometry creation, we will load an existing DXF CAD file

Step one is getting the files you will need

- ➤ Minimize Shopcam if it is open

- ➤ Open www.Shopcam.com in your favorite browser

- ➤ Go to the 'Support' page

- ➤ Scroll down to 'Mach-3 Quicks Files' and download to you desktop.

- ➤ Close or minimize your browser

- ➤ Right-click the ZIP file you just downloaded and select 'Extract All'. This will create a folder with all the files needed.
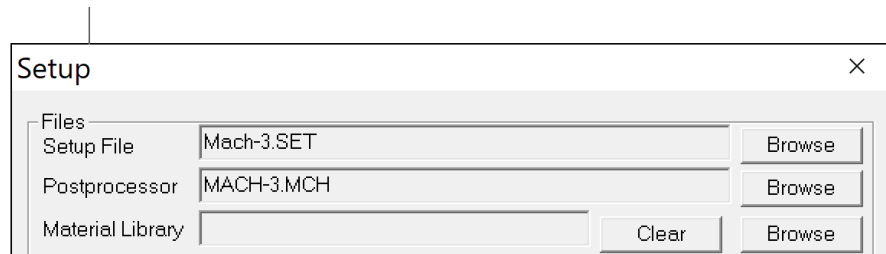
Now we can start.

- ➤ Maximize or open Shopcam.

- ➤ You should have a blank screen.

- ➤ Let's make sure Shopcam knows what machine you are using. Note, once you save a setup file, that will become the default setup file for every new part.

Mach-3 Quick Start guide

➢ Go to [Info} → [Setup File]

➢ Make sure the setup file is Mach-3.SET. If not, click on the [Browse] button and select the Mach-3

Setup   ✕

Files
Setup File   Mach-3.SET   Browse
Postprocessor   MACH-3.MCH   Browse
Material Library [   ]   Clear   Browse

Now we are ready to read in the DXF

➢ Goto [Files] → [Import] → [DXF File]. Note the system will look in the path listed in [Info] → [Default Directories]

Enter a dxf Filename

Look in: Mach-3 Quick Start Files ▼ ⇐ 🗁

Nam: dan@shopcam.com (dan-ryzen)
          Desktop
Ma   Mach-3 Quick Start Files
          3D Objects
          Documents
          Music

➢ At the top of the files dialoge box you will notice a drop-down arrow. Browse to the desktop and select the folder 'Mach-3 Quick-Start files'

➢ Select the DXF file.

➢ Click the [OK] button to read all the layer.

➢ Click the [OK] button for the warning screen. Since we know this DXF doesn't have any unsupported object, there is no need to save-as then reopen.
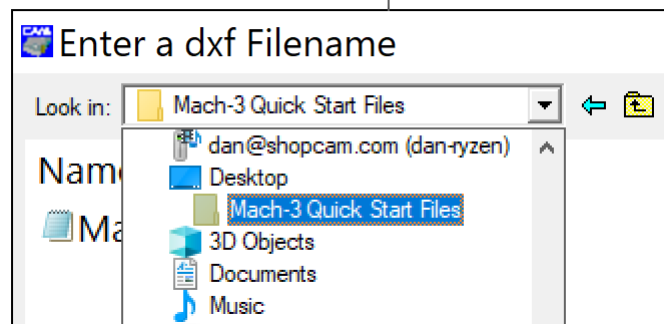
 Now that the hard part is done, we can do the actual part. If you will be making your own DXF files, I suggest going to [Info] → [Default Directories] and change the default CAD folder (via the [Browse] button to the folder your CAD program uses.
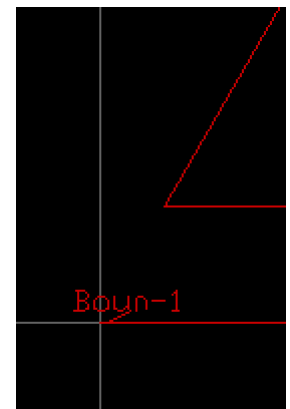
Mach-3 Quick Start guide

With the DXF loaded, we can make a boundry for the cutting order and a toolpath.

➢ Select [Group] → [Boundry]

➢ The default pick method is *Chain.* You should leave this. As long as contiguous geometry is found, it will make the group. If the entire part turns blue, you have good clean geometry. If it stops midway or doesn't chain at all there is a reason. See pages 18 and 26 of the *'Technical Reference & User guide'*
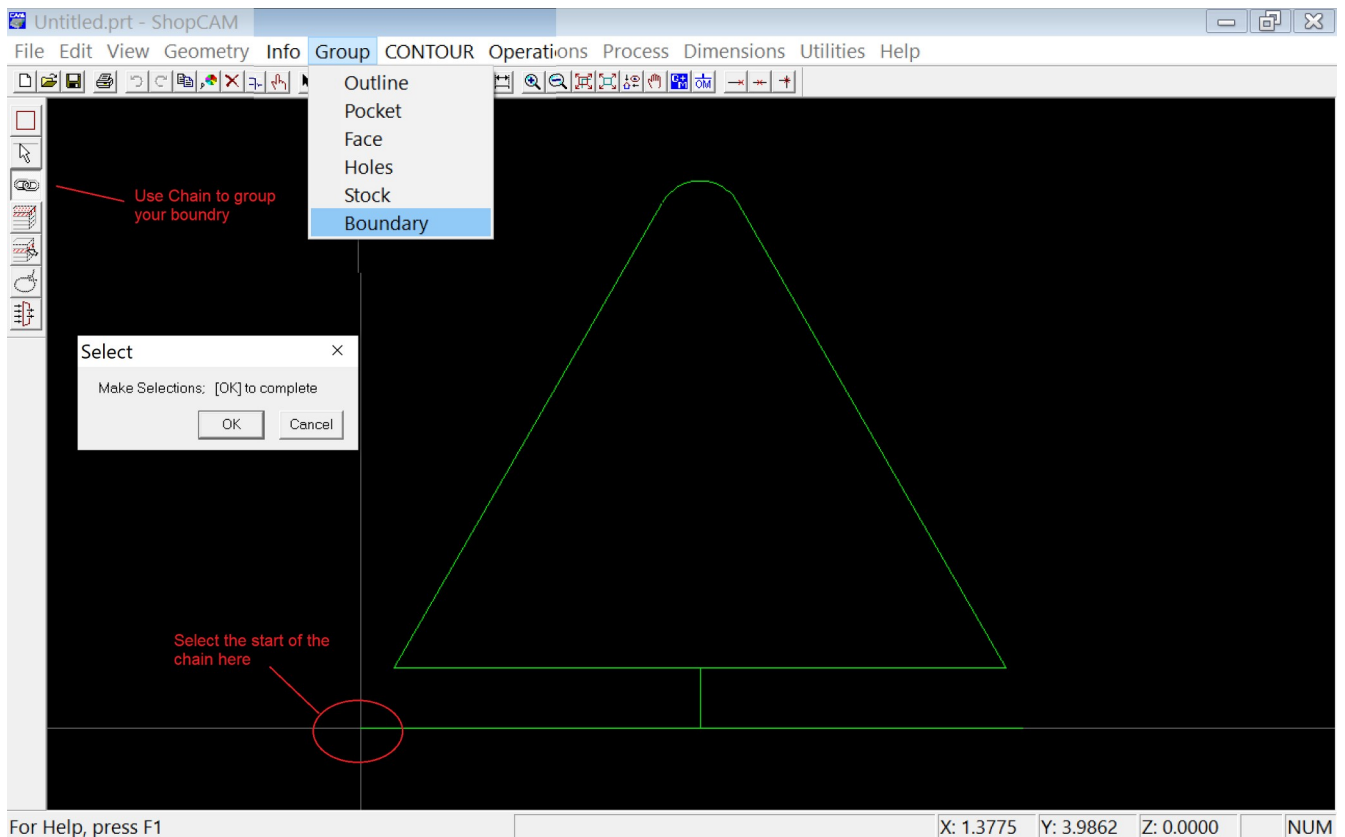

➢ Select near the starting point. The part will turn blue. Hit the [OK] button.

➢ The system will create a boundry layer on the next available on layer or layer 1 in this case. Notice Boun-1

Mach-3 Quick Start guide

At this point we can add our toolpath.

➢ Select [Operation] → [Finish]

➢ Either click on screen or the check box next to '1 Boun'

➢ At this point you may change parameters* to make the G-code appear different. See Mach-3.PDF vocated in the *'Mach-3 Quick Start Files'* folder, on your desktop.

➢ Hit the [Make Tool path] button and the yellow toolpath will be put on layer-2 as Path-2

➢ Hit the [Close] button.

➢ Select [Process] → [Post Process] then either [NC code only] or [Draw 2D Animation]

➢ You will be prompted to save the file. This is the Shopcam PRT file. Not the G-code file

➢ If 2D animation is selected, you can control the draw speed with the up-arrow or down-arrow on your keyboard.
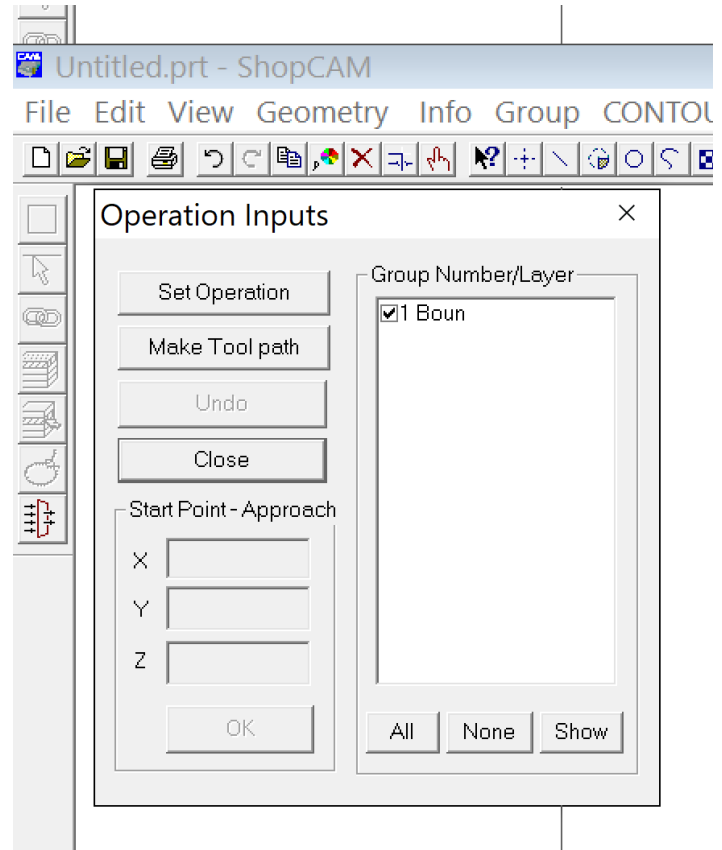
When posting is complete a 'posting complete' window will appear. At this point, the G-code has been created.   Select the [Editor / DNC] button to view

*The G-code will look different based on the parameters selected. See a detailed description located in the Mach-3 helpfile. Print the Mach-3.pdf file, located in the *'Mach-3 Quick Start Files'* folder on your desktop.

The G-code file will have a tap extension and will be located in the default tape directory.