MeshCAM and CNC Bitmap Approach

Opening the Bitmap and setting its dimensions 1-8

1. Open MeshCam
2. Under File, open the desired Bitmap file
3. A window called "Set Bitmap Dimensions" should appear
4. For the X, Y, and Z sizes you must keep in mind of your stock and remember these numbers are in inches
   (Example, Soap bars are approximately X=2.2 Y=3.75 Z=.90; it is recommended you set X size to 2, Y should resize proportionally, and then change z to 0.5)
5. You should now uncheck "Don't Resize Bitmap" and "Keep Proportional" allowing you to edit "X and Y stepover" and make sure "White is Z+" is on
6. "X and Y stepover" must be at least 1/10 to 1/3 of the drill bit's size (the bit should be .125 and so 1/10 is .0125)
   Note: The stepover sizes should be equal and increasing stepover speeds up time spent drilling
7. Hit OK.
8. Should you want to change the dimensions you would have to reload the Bitmap

Defining Stock, Build Z map, and Setting Retract Height 9-17

9. Under CAM, click Define Stock
10. A window should appear showing "Set Absolute" and "Set Relative" each over one picture
11. You do not need to edit anything related to "Set Relative" and they should be all 0
12. For X and Z they should be close to what you set initially (With X being 2 and Z being near .5)
13. For Y you must change it to 3.60 at least but do not go over 3.70 or the milling machine will carve into the clamps
14. Hit OK, you will notice the white square around the image increased through the Y axis
15. Under CAM, click "Build Z Map" and you will be prompted to choose "Bottom Level" or "Top Level"
16. Choose "Bottom Level" and hit OK
17. Under CAM, click "Set Retract Height" and, when prompted, leave it at 0.1 and hit OK (this may not be necessary as 0.1 is its default)

Generating a Toolpath 18-28

18. Under CAM, click Generate Toolpath
20. "Tool Diameter" should be 0.125 and "Tool Type" as End; set "Plunge rate" to 10 is recommended but increasing it would speed up drilling slightly (to a maximum of 20)
21. Make sure "Generate Roughing Pass" is checked
22. "Depth per Pass" should be around 0.15 (maximum of 0.20) but increasing does decrease drilling time just keep in mind the length of the bit
23. Set "Feedrate" to its maximum of 20 for quicker results and leave Skin Thickness alone
24. Make sure "Generate Cleanup Pass" is checked
25. You do not have to change anything under "Cleanup" but you can set "Max Skin Thickness" to 0
26. Make sure "Generate Finish Pass" and all three "Cut Along..." are not enabled
27. Set Feedrate to 20
28. Recheck the "Tool Type" to be end and hit OK

Sherline CNC (Milling Program) 29-

29. When the milling machine computer boots up, start "Sherline CNC (inch)"
30. Click the a horizontal button at the top left called, "Manual"
31. Click a vertical button at the bottom right called, "All Zeros"
32. Click the button left of "Manual" called "Auto"
33. Near the bottom middle click "Open" and all the toolpaths should be in the folder "gcode"
34. Click and open the desired toolpath
35. Turn on the milling machine and the switch on the left of the computer
36. Now click "Run"
Note: Assuming you have set the Milling Machine at the bottom left corner everything should go well!