Mastercam
Engraving With Contour

Mastercam is a CAD/CAM (Computer Aid Design/ Computer Aid Manufacturing) program. This Worksheet walks you through Engraving Using Contour

To understand how to use Contour to engrave you need to understand how the machine reads the code. The machine follows the line that is specified in the code. The Center of the cutter follows this line. Thus if the Cutter is a .5 in diameter, .25 of the cutter is on either side of the line. If you want to cut a 2’X 2’ square you need to program so that the cut path is .25 larger in all dimensions to end up with a 2x2. MasterCam will automatically offset the cutter path for you so that you get a 2x2 block.

When using Contour to engrave you want the cutter to follow the line without offsetting in either direction. This worksheet will walk you through to use contour to engrave.

BEFORE GETTING STARTED

1. Under Toolpaths Select Contour.
2. Make sure NC name is the same as your file name.
3. The Chaining Window will open. A is C-plane (Construction Plane) you want to work only in the C-plane to make sure it is marked. B is loop, this will select a group of lines that make a complete circle. C is a single line this allows you to select just one line at a time. E is partial loop this will select a group of line that are segmented one after the other. D is unselect this will remove the last line, loop or partial loop you selected.
4. The Green arrow indicated where the cutter will start and the red arrow were it will end as well as the direction the cutter will move along the line. To change the direction of the arrow us reverses or F.
5. IMPORTANT NOTE When selecting the line, select them in a way that they flow together (all arrow point in the same direction and as linked as possible). Try not to jump around the part.
Mastercam
Engraving With Contour

6. Once all the line segments you want to cut in this operation are selected, Green Check.

7. The 2D Tool Pathing-Contour page will appear. Check that Contour is highlighted, then Double Click on Tool.

8. Select Tool Library.

9. There are thousands of possible tools, so to help a filter is on that allow you to select form categories of tools. For this project we have TWO tool options, one is the 3/32 end mill or a engraver. To only see these type of tools click on the filter.

10. On the Tool List Filter page, select only Flat End Mills and Engraving Tools.

11. The two possible tools for this project are tool # 231 or tool 520. Choose one that will work best for your project. And

12. On the Tool page, mark Tool # as 1. If you are using more tool the mark subsequent tool in the order that you will used them.

13. This toolpath parameters tab only needs to be done once per tool. If you use the same tool for multiple contours, all the setting on this page should stay the same.

14. Switch to Cut Parameters

15. Change the Compensation type to OFF

16. Under Lead In/Out

17. Unckeck the box in the upper left corner

18. Click on Linking Parameters

19. Depending on the cutter you are using the depth will change. For an engraver (tool 520) the depth should be -0.03. If you are using the end mill (tool 231) you can typicily cut up to half the cutter diameter. Give that a 3/32 cutter is .09375 set the depth to -.05.

20. MAKE SURE YOU MAKE IS A NATIVE NUMBER