

Drexel Autonomous systems lab

- [Home](#)
- [About](#)
- [Tutorials](#)
- [Co-Op](#)
- [Resume](#)
- [Contact](#)

Links

[DASL](#)

[Dr. Paul Oh](#)

[HUBO](#)

[PCNC Mill](#)

[MasterCAM](#)

[Confero](#)

[Makezine](#)

Lesson 5
Mastercam x2 basics

[Return DASL -131 Main](#)

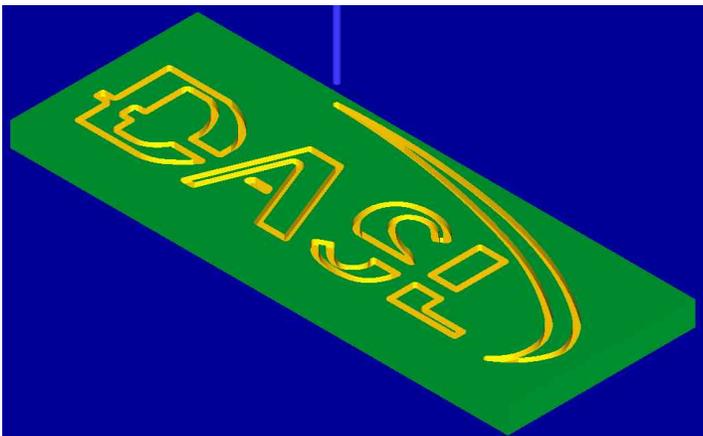


FIGURE 1

☞ Lesson 5 introduces NC code generation with Mastercam X2. Computer-aided Manufacturing (CAM) mixes Computer-aided Design (CAD) with manufacturing techniques to create an efficient and automated NC code generator.

[Section 1 - Overview](#)

[Section 2 - Menu Navigation and Part Setup](#)

[Section 3 - Path Layout](#)

[Section 4 - NC Simulation and Generation](#)

Section 1 - Overview

☞ Mastercam X2 software is a CAM product made to make NC code generation faster and more efficient. This tutorial and following project requires CAD experience. Reference DASL software library to acquire proper CAD software. This tutorial will supply STEP files that are to be used to generate the examples.

If MasterCAM X2 requires HASP USB dongle to run the software the DASL single seat. The seat is installed on the the computer located in the Office in Bossone 107. Plugged into the computer is this HASP dongle, this dongle stays with this computer unless is cleared to be moved by Robert Ellenberg or Roy Gross. The dongle must be plugged into the computer upon boot and if removed during operation will lock the use of Mastercam X2.

Under START menu Mastercam is listed under Mastercam X2 MR2

Section 2 - Menu Navigation and Part Setup

☞ Click [Menu Navigation](#) to view a screen shot and labels of the Main dashboard

Follow the following steps to set up the Fourth Example that is the same from lesson 1. These steps should be followed each time for setting up a STEP file.

Along the top are the generic menus (file, edit...etc.). Find Machine Type > Mill > Default. Clicking this a new feature will appear in the Toolpaths Manager "Machine Group-1". Click here to download Example1.STEP. File > Open > Set "files of type" to STEP Files (*.STP;*.STEP) . Select file from download location and click the green check mark. Now the wire frame diagram located at Origin. Under Views click the green wire cube that is not filled in (Isometric View). Now click the blue sphere that was labeled as Wire/Solid and then click the Center icon, both of these are labeled in "Menu Navigation" Then Click Properties - Generic Mill > Stock setup. This sub-menu " Machine Group Properties" should be used to define the region or "stock" dimensions so Mastercam knows it's working bounds. Click the All Solids button, check to see if the dimensions are correct and then press the green check button. Notice the stock bounds will show up as a red dotted Rectangle around the CAD. Remember this represents the "real" un-machined stock that the machined part will be created from.

Section 3 - Path Layout

⊕ This section will outline the basics to create the CAM tool paths to machine Lesson 1 example 4 part . In the introductory example 4 a 1/2inch Flat end mill was used. For this example a 1/4inch Flat end mill will be used. The NC code from lesson 1 example 4 only cuts out the "island" but does not take care of the reaming stock to make a complete part. Rerun NCPlot example 4 code to check this out. To make this a complete part there will be two groups of tool passes. One will Contour the curve in the Island and the other will eliminate the reaming material.

To start go to the menu bar and find Toolpaths > Contour Toolpath... > a dialog box will appear, click the green check > "Chaining" sub menu should appear. The following steps will be circled in red in the figure 2.



FIGURE 2

Click the Red wire frame that is filled in blue (Solids). The sub menu should change then click the blue box with the red edge button. Now click the round edge that is shown below in figure 3. The edge of the should turn blue and a green arrow should also appear on the edge and click the green check mark to complete the contour setup. A new menu should appear "Contour (2D)". Now find "Select library tool..." button located on the "Toolpath parameters" tab and another menu should appear make sure MILL_INCH.TOOLS is selected. Find tool 235, click to highlight blue and then click the green check mark . Find feed rate, plunge speed, Spindle speed, and retract rate. Leave as the Default F 6.4176 and S 2139; these are the DRO inputs that will be changed to adjust the feeds and speeds depending on material. Click the "Contour parameters" tab and uncheck the the box next to "Retract...". Retract is used to lift the tool above the part after each pass, removing cuts down machining time. "Top of stock..." should read 0.0 and "Depth.." should read -0.1 if not then click the

"Depth.." button and position the + cursor over a corner that joins the base of the island and the remaining stock surface shown in figure 3. To test this cut to see if it was constructed correctly click the Toolpath Manager > "Verify the selected toolpaths" (purple solid icon). The screens should change and a window called "verify" should appear. There is a slider bar between a black person walking and a red person running, place the slider in the center to control the speed. Then press the black triangle (>) to run the machining simulator. There is rewind option to view again (double back arrow <<). To exit press the green check. If everything looked like it worked as planned move on to next path. If it did not, to edit click the toolpaths manager and click on "Parameters". To set up the next tool path follow the same steps to get to Contour sub menu. After each edit the toolpaths become "dirty" and cannot run the simulation until fixed. To fix it click the toolpath manager and click the button that looks like a tool with a red X next to it (Regenerate all dirty operations). Next Click the red wire frame that is not filled in shown in figure 4. Click the button that has a single red diagonal line show in figure 4.

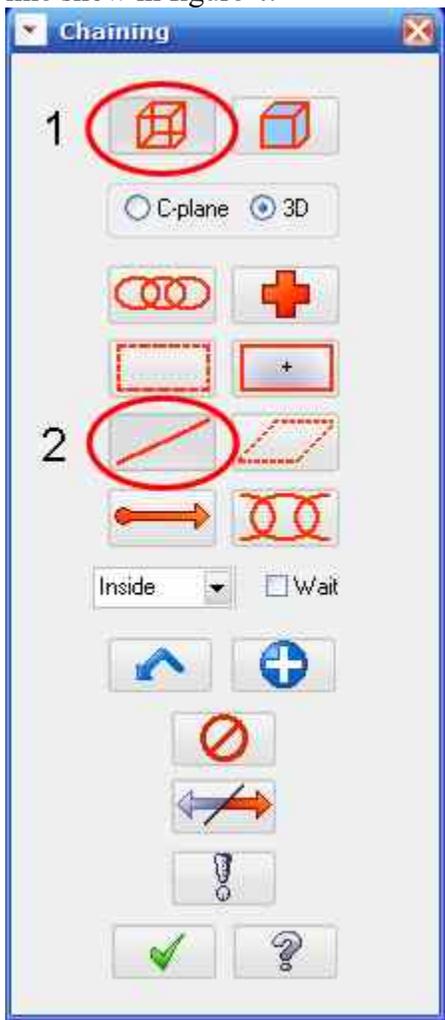


FIGURE 4

Now click each edge of the block as shown in figure 5 the edges should turn yellow. Click green check mark and "Contour 2D" should appear. Use the same selected tool. Go to the second tab and set the same as last contour but change the drop down menu "Compensation Direction:" from Left to Right. No run the simulation as previously stated. Notice that it only runs the edge material removal (last path generated) . To show manufacturing of entire part, click the toolpath Manager and click "select all operations" button, it is a green check mark with a green arrow next to it. After you verify the correct paths are generated, Complete CAM, the next step is to generate the usable NC code. Click

the toolpath manager and click the button that is a blue G1. A window named "Post Processing" appears and press the green check mark. Then the "Save As" window appears, rename (Example4) and location is based on user. After saving the Mastercam X Editor should maximize to full screen. Now the code is now in raw NC code to be edited. Line number 162 and 164 should be deleted, issues arise when running Mach3 interpreter. Also if this was going to be machined M8/M9 would have to be added as taught in Lesson 1. Copy completed code and paste it into CNC simulator and verify with the code below.

Section 4 - NC Simulation and Generation

[Return DASL -131 Main](#)

[Valid XHTML](#) | [Valid CSS](#) | Copyright © Roy J. Gross | Design by [SmallPark](#)