

AC 2008-297: IMPLEMENTATION OF CAD/CAM/CNC CURRICULUM USING MASTERCAM X SOFTWARE IN TECHNICAL PROGRAMS

Farzin Heidari, Texas A&M University, Kingsville

Implementation of CAD/CAM/CNC Curriculum Using MastercamX Software in Technical Programs

Introduction

One of the major contributions to the growing acceptance of Computer Numerical Control (CNC) has been the development of Computer Aided Manufacturing (CAM). A Computer Aided Manufacturing (CAM) program uses a CAD drawing, and then adds the cutting parameters to generate the CNC program.

MastercamX is PC-based CAD/CAM software, which integrates the geometry, toolpath, and the G-code program together. The CAD section of the Mastercam software generates a dimensioned drawing using basic entities. The geometry file is saved as a CAD file and then turned into CNC code. The CNC code is used to machine parts on a milling machine. Mastercam software ties the CAD/CAM process together to generate a CNC program to make a part, using a machine tool.

CAD/CAM Process

Mastercam uses levels to identify different parts of the same geometry on different layers. Each layer or combination of layers can be selected to be visible or invisible. It is also possible to delete, move, or copy entities from level to level. This feature can be used to create part programs for complex 2D drawings, by breaking down the geometry into different sections and creating each section of a drawing on a specific level according to its toolpath requirements. Each level can be selected at a time for certain toolpath operation. Combination of all toolpaths created for each level will produce the complex toolpath for the drawing.

There are five different toolpath options provided by Mastercam for 2D geometry to create a part program. They are face toolpath, contour toolpath, drill toolpath, pocket toolpath, and engrave toolpath. The five basic 2D milling operations toolpath supported by the Mastercam software are used to create part programs for most parts. The face toolpath removes materials across the top surface of the work piece. Contour toolpath, removes material to a certain depth along a specified CAD drawing. Drill toolpath; identify selected points on a drawing to drill holes to a specified depth. Pocket toolpath, removes all materials within the outline of an enclosed geometry. Pocket has the useful feature of island, which allows any shape within the enclosed geometry stay intact, and remains, as an island. The island feature of the pocket toolpath is used in many complex 2D parts. Engrave toolpath, creates toolpath to engrave art and letters.

During this process three different steps are taken, first part geometry is created, the CAD file, then toolpaths are selected by assigning one of the above toolpaths to a section of the drawing. Each selected toolpath has its own parameters related to tool definitions and cutting parameters including feed, speed, plunge rate, depth of cut and size of the cutter. The result, which is a visible toolpath, can be verified and checked for accuracy and correctness. This is the CAM process, which the geometry is used to create a visible toolpath. Finally the CNC program can be created using the appropriate postprocessor for milling machine controller, which is then used to cut the part.

Example

The following drawing was created as an example to represent the application of levels in Mastercam software. The drawing is divided to six different parts. Each part presents a level and a toolpath (Figure 1). The combinations of all toolpaths results in the creation of CNC program for the part shown in figure 2.

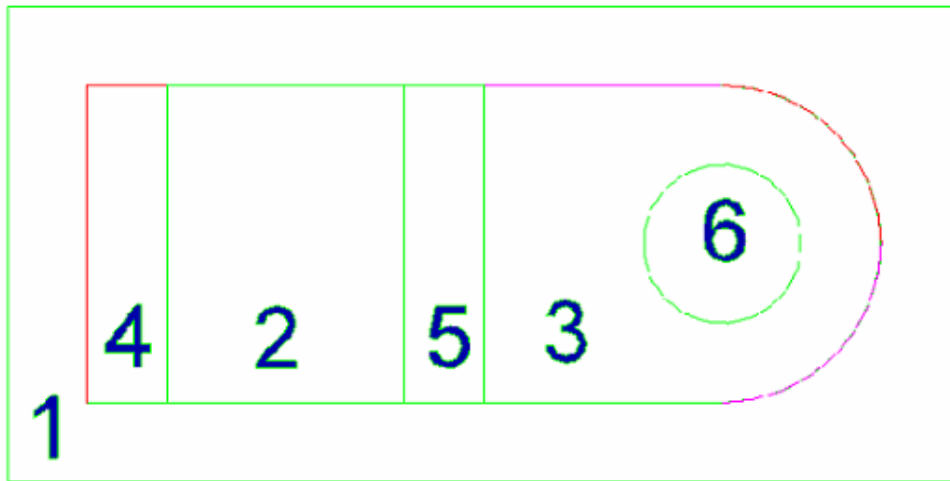


Figure 1. 2D drawing with six different level saved as CAD drawing.

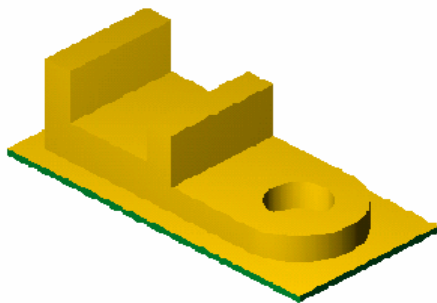


Figure 2. Finished machined part combining all six level toolpaths.

The drawing in figure 1 is divided into six levels. Level 1 includes the border and the shape profile of the part. The first toolpath is a pocket with the profile as the island. The depth of the

cut is set at Z -1.75. The toolpath verification and the resulted part shape after the first operation are shown in figure 3.

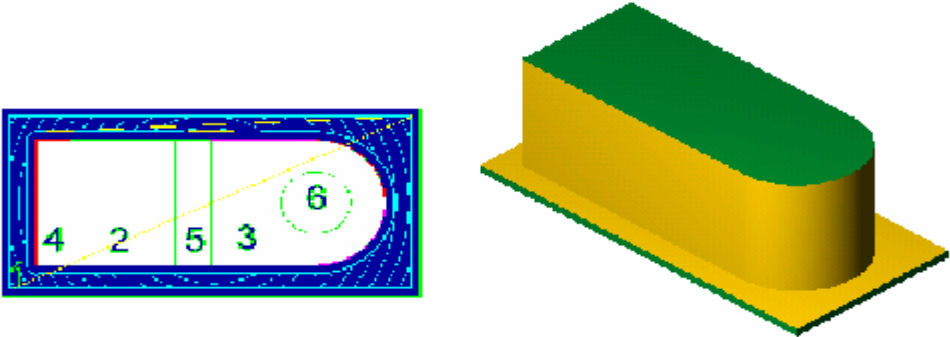


Figure 3. Toolpath verification and resulted part for level 1.

Toolpaths for level 2 and 3 are pockets set at Z-1 for level2 and Z-1.25 for level3.

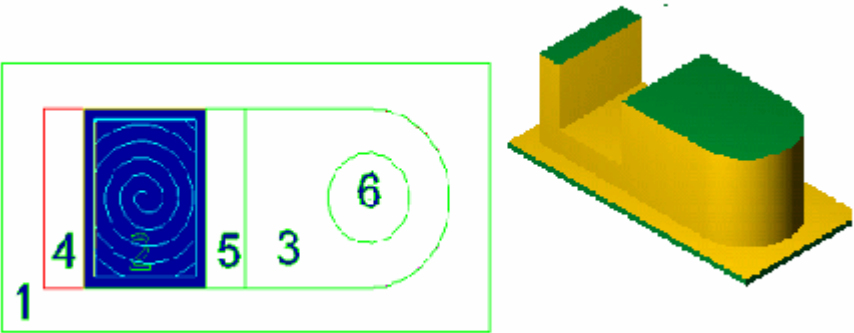


Figure 4. Toolpath verification and resulted part for level 2.

Toolpaths for level 4 and 5 are face operations. The depth of cut for level 4 is set at Z-.25 and for level 5 is set at Z -.5.

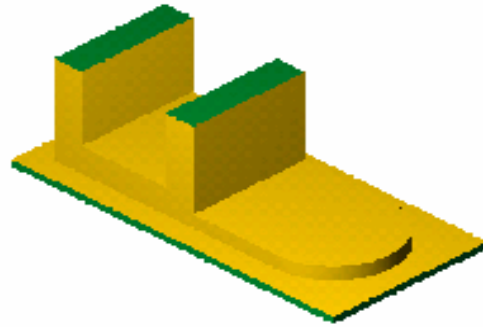
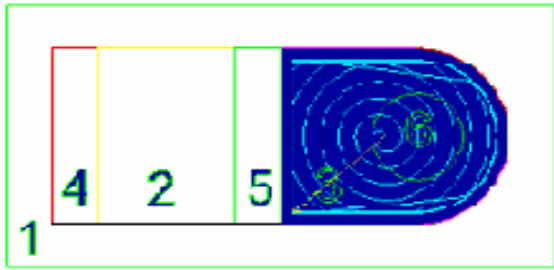


Figure 5. Toolpath verification and resulted part for level 3.

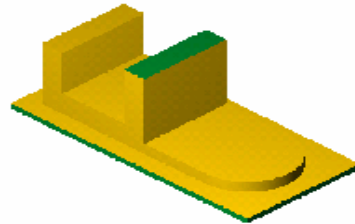
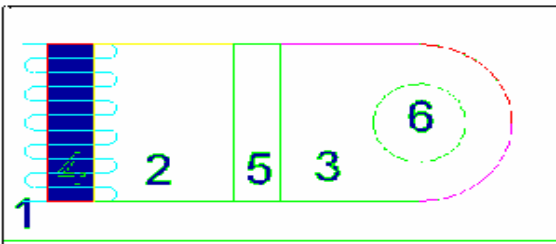


Figure6. Toolpath verification and resulted part for level 4.

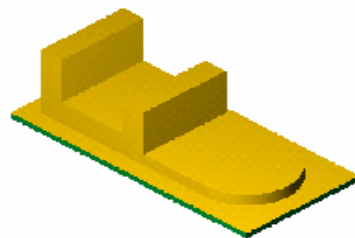
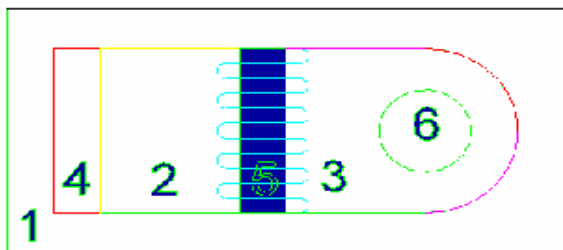


Figure 7. Toolpath verification and resulted part for level 5.

Toolpath for level 4 and 5 are face operations set a Z-.25 for level 4 and Z-.5 for level 5. The last toolpath level 6 is a pocket. The circle geometry is pocketed to generate the hole. A drill toolpath can be used instead of a pocket. The depth of the pocket is set at Z-1.75.

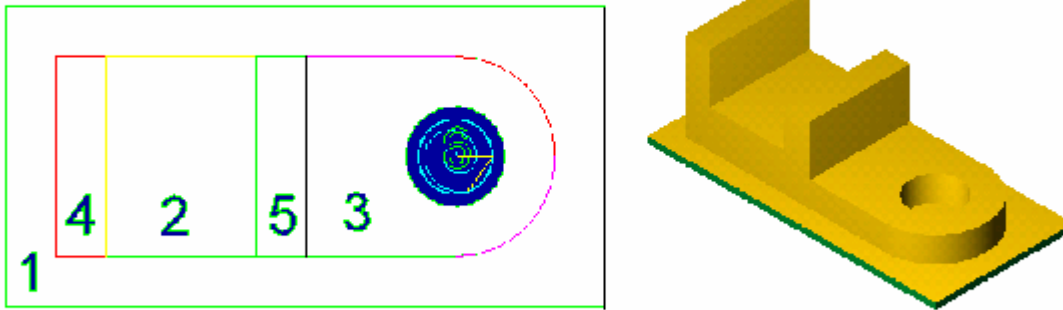


Figure 8. Toolpath verification for level six and the finished part.

This approach is only used for creating CNC programs that are not simple and present a degree of complexity in generating the part program. If the toolpath is simple and the areas that need to be machined are not connected to each other, the toolpath can be created on the same drawing without changing the levels. For simple parts different section of the drawing can be selected one at a time for each toolpath to form the part. Creation of different levels for the purpose of toolpath generation is an effective and efficient method for generating a challenging part. This method allows the programmer to simplify the toolpath creation process and combine individual toolpaths to create a part program.

Bibliography

1. Heidari, F., (November, 2006). Mastercam X Update and Demonstration. Paper presented at the National Association of Industrial Technology (NAIT).
2. Heidari, F., (November, 2001). Converting Bitmap Images to Vector Files for CAD/CAM Integration. Paper presented at the National Association of Industrial Technology (NAIT).
3. Lin, Jonathon, 2006. Mastercam X Mill and Solids. Ann Arbor, MI: Scholar International Publishing Corp.
4. Walker, John, 1998. Machining Fundamentals. Tinley park, IL: Goodheart-Willcox Publishing Company.