

This week's laboratory assignment involves using the CAM software (Mastercam X4) to create a part program for the part shown in Figure 1, post-processing the program and running it on the Emco-Meier CNC milling machine.

1. Using the Mastercam X manual and help files (if necessary), create a part program for the part shown in Figure 1. The workpiece is Aluminum 6060-T6, 2×2×0.5 inches in size with the origin located on the upper left corner of the top surface. Use the cutting conditions below for the following three operations: contouring, pocketing, and drilling.
 - Common parameters
 - Clearance plane: 0.05 in.
 - Tool change location: X=-1.0, Y=1.0, Z=1.0
 - Coolant: off
 - Material: ALUMINUM inch—6061 (Available under “Mill—library” in the tool settings tab.)
 - Machine type: MPPC100M.MMD (EMCO PC100 Mill)
 - Contouring and Pocketing:
 - Cutting tool: a 2-flute flat end mill with a diameter of 0.25 in. (use tool ID T2)
 - Cutting speed: 4714 RPM
 - Feed rate: 14 in./min.
 - Plunge feed rate: 4 in./min.
 - Max depth of cut: 0.125 in.
 - Run roughing and finishing in two opposite directions for a better surface finish.
 - Drilling:
 - Cutting tool: a 2-flute flat end mill with a diameter of 0.1875 in. (use tool ID T3)
 - Cutting speed: 1500 RPM
 - Feed rate: 1.4 in./min.
2. Once the geometry and part program is created, you must store it on your network drive space or personal storage media. To save the file, click the File menu and highlight “Save as”. When the submenu is displayed, type in the file name.
3. Post processing will create ISO or EIA codes in an ASCII format for the CNC you specify. Execute the CNC post processing. After the CNC data is processed, you need to save the .NC file onto a floppy disk for transfer to the Emco mill using a name with the structure: OXXXX (letter O followed by 4 digits—the program number—with no file extension.) If you haven't got any error message, exit from Mastercam X4 and print out the .NC file.
4. Take the part program to room 58, load it onto the Emco mill (by TA or Mike Sherwood) and perform a simulation. If the program works, make an actual cut. If not, bring the program back and perform debugging.

Write a report describing the overall procedure used in Mastercam X4, changes made in the part program and your observation. Also include in the report the printout from Mastercam X4 drawing and the generated part program.

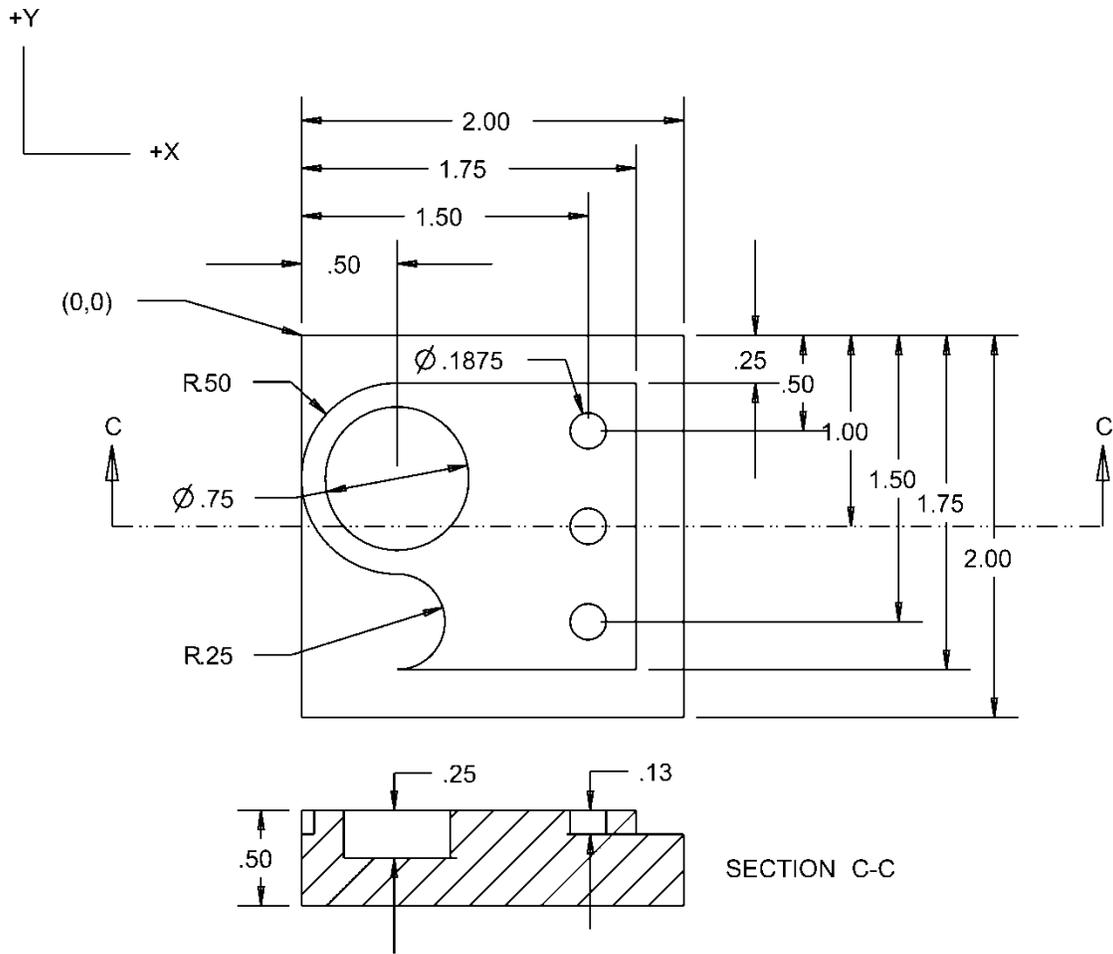


Figure 1: Part drawing.