ME 363
[Lab #7] CNC Machining II

This week's laboratory assignment involves using the CAM software (Mastercam 2022) to create a part program for the part shown in Figure 1, post-processing the program and running it on the Tormach 770M CNC milling machine.

1. Using the Mastercam manual (if necessary), create a part program for the part shown in Figure 1. Use Tormach VMC Machine type before defining tool paths. There are three features on this part: middle island generation, pocketing, and drilling (use chip break cycle for drilling). The workpiece is Aluminum 6061-T6 and the dimension is 2 inches (w) by 2 inches (d) and 0.5 inches (h) in size. Use the following cutting conditions:
   - Cutting speed: 2000 rpm
   - Feed rate: 12 inches/min.
   - Cutting tool: a carbide end mill with a diameter of 0.25 in and 3 flutes (use tool ID T2).
   - Stock to be left for finishing pass: 0.01 in.
   - Speed to be used for drilling: 2000 rpm
   - Feed to be used for drilling: 5 in/min
   - Depth increment to be used for drilling: 0.1 in.
2. Once the geometry and/or 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 process the part program using Tormach VMC machine definitions AND post processors. Post processing will generate a G-code program in the form of an .NC file. Execute the CNC post processing.

Take the saved Mastercam file to the Tormach Mill via USB flash-drive to have it checked. Once the program is cleared for cutting, machine your workpiece. If not, bring the program back to Mastercam'22 and perform debugging. Write a report describing the overall procedure used in Mastercam, changes made in the part program and your observation. Also include in the report the printout from Mastercam drawing and the generated part program. Indicate the portion of the program corresponding to each major feature of the part (such as roughing, finishing, pocketing and drilling).

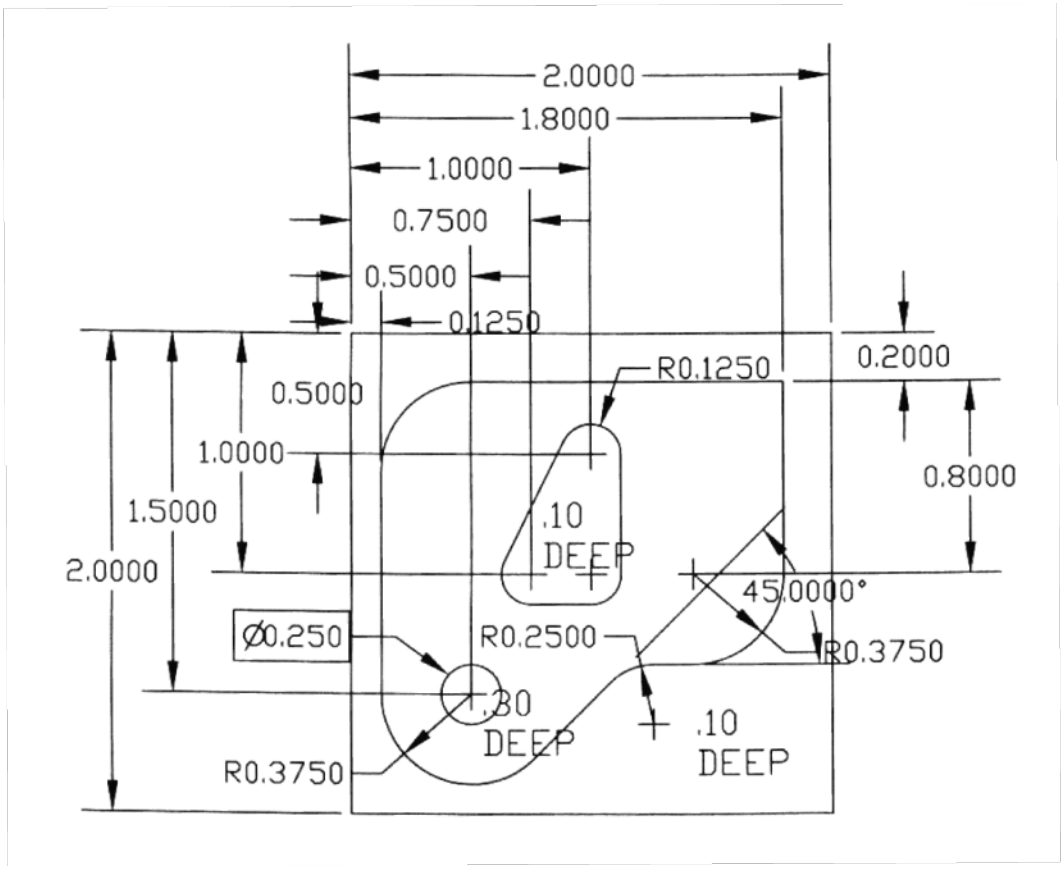

Figure 1: Part dimensions for CNC milling.

NOTE: For the session on milling, the workpiece has to be oriented in Mastercam according to the layout shown in Figure 2 below. In other words, the top left corner of the workpiece will be at the origin of the Mastercam screen with all the Y coordinates being zero or negative.

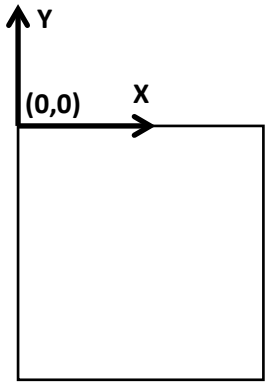

Figure 2: Coordinates for the CNC milling part.