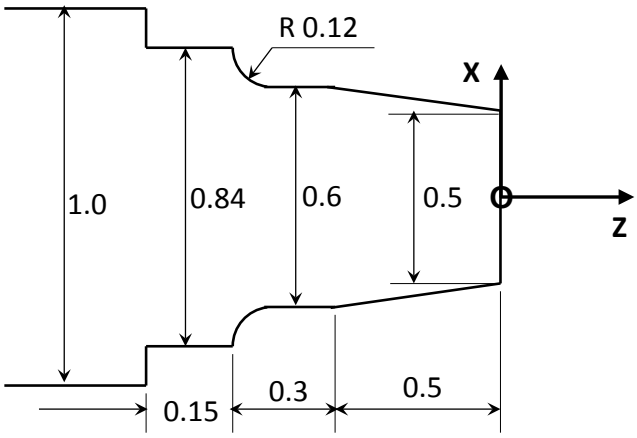
ME 363
[Lab #8] CNC Machining III (Turning)

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 8L lathe.

1. Using the Mastercam manual (if necessary), create a part program for the part shown in Figure 1. Use default machine type before defining tool paths. The workpiece is Aluminum 6061-T6, 1 inch in diameter and 2.25 inches in length. Use the following cutting conditions:
    - Tool number for part program: 1
    - Insert material: uncoated carbide
    - Insert :
        - diamond shaped
        - included angle of 55°
        - edge angle of 3°
        - nose radius of 0.015 in.
    - Speed for roughing and facing: constant cutting speed of 3.33 fpm (or 200 ipm) up to 2000 rpm
    - Feed for roughing: 0.008 inch per revolution
    - Feed for facing: 0.004 inch per revolution
    - Depth of cut for facing: 0.005 in. per path (to remove the material up to 0.010 in.)
    - Speed for finishing: constant cutting speed of 3.33 fpm (or 200 ipm) up to 2000 rpm
    - Feed for finishing: 0.004 inch per revolution
    - Depth of cut for roughing path: 0.02 in.
    - Allowance for finishing: 0.012 in.
    - Use 2 for x index and 1.5 for z index
2. For facing, a single pass cutting is sufficient. For roughing and finishing of longitudinal cutting, verify the tool path after tool path is generated. Specify the withdraw angle (other than 45° shown in the tutorial) to avoid the collision of the tool into the workpiece, if necessary. **NOTE**: The origin for this component has to be set at the point shown in Figure 1. Therefore, all the Z coordinates will be negative and all the X coordinates will be positive.
3. 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.
4. 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.
5. 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 of drawing from Mastercam and the generated part program. Indicate what portions of the part program correspond to each machining operation (such as facing, roughing, and finishing).



ALL DIMENSIONS ARE IN INCHES

Figure 1: Part dimensions for CNC turning.