



JOMO KENYATTA UNIVERSITY OF AGRICULTURE AND TECHNOLOGY

Practical Exercise on CNC Milling Machine

1 Background

1.1 Safety

- i. Obtain instructor's permission.
- ii. Do not alter or modify any machinery, tooling or accessory unless you contact an instructor and obtain permission.
- iii. Before starting the machining, always warm the spindle by running it in steps of 500rpm for at least four (4) steps to lubricate the spindle
- iv. Review all the CNC milling machine set up and operating procedures provided.
- v. Review all the CNC milling machine programming instructions provided.
- vi. Prepare and review your program carefully.
- vii. Edit your program for safety, format, correctness and clarity before running it.
- viii. Always simulate the program before running it. Note any wrong tool paths and/or errors and edit them.
- ix. Securely clamp the workpiece on a vice or the table
- x. While loading tools on the spindle, hold the tool properly to avoid injury by pneumatic tool clamping mechanism
- xi. Do not use compressed air to clean the chips
- xii. Always start machining with minimum feed override, and increase it slowly as you ascertain that the program is machining as desired
- xiii. Always unplug the air hose from the supply once finished with the machining exercise

1.2 Introduction

This exercise involves four processes of milling a pocket. These are;

- i. facing using a facing tool
- ii. roughing using a flat end mill
- iii. finishing using a ball end mill
- iv. engraving using an engraving tool

The machining programs will be prepared on a CAM software in the computer laboratory. The student is expected to draw the part to be machined using the appropriate CAD software before generating the G-codes for machining. The programs are expected to be written as one continuous program. This will be done in order to demonstrate the tool changing process for the CNC milling machine.

2 Scope

2.1 Machine

This machining exercise shall be conducted on XYZ 1510 VMC 4-axes CNC milling machine.

2.2 Workpiece material

The XYZ 1510 VMC CNC milling machine has the capability of machining a wide variety of materials, but for the purpose of this exercise, a recycled aluminium blank that is cast in the foundry workshop will be used.

2.3 Size of the blank

The dimensions of the blank are 150 mm by 150 mm by 42 mm. The workpiece will be face milled for 2 mm to make it 150 mm by 150 mm by 40 mm.

NB *The face milling will be done in steps of 0.4 mm (5 steps)*

3 Procedure

- i. Connect air supply hose to the machine, power on the machine
- ii. Follow the machine start-up procedure (consult machine's operation's manual)
- iii. Clamp the workpiece on the machine vice or table
- iv. Load the tools as per the tool load procedure (refer to machine's operation manual)

- v. Measure the tool lengths
- vi. Measure the workpiece dimensions and set the workpiece offset (**remember to set the Z-zero at the top of the workpiece**). **This should coincide with the Z-zero position that is set when generating the G-code program for safety**
- vii. Make sure the coolant is turned on manually at the taps before machining starts
- viii. Load the program in the machine's computer
- ix. Edit the program to conform to the machines format (refer to operator manual)
- x. Simulate the program on the CNC machine, incase of any errors, edit the program
- xi. Turn the feed override to minimum and start the program execution once satisfied with the simulation
- xii. Increase the feed rate upwards slowly using the manual override switch as the machining progresses, i.e., after the first successful cut and z-feed, up to 100%
- xiii. After the machining, un-clamp the workpiece from the vice/table and unload all the tools (consult user manual)
- xiv. After machining process is over, clean the machine using a bloom or a soft cloth. Use the chip wash function of the machine to clean the chips that are far off. **Do not blow off the chips with air**
- xv. Press the emergency stop button and power down the machine.
- xvi. Disconnect the air hose

4 Results and Discussion

- i. Discuss the machining process stating any observations, challenges and merits of CNC machining.
- ii. Explain the choice of milling speeds and feeds taken for the machining process.
- iii. Explain the tooling chosen for each of the processes in section 1.2.
- iv. Discuss the surface finish for each of the processes undertaken.
- v. Discuss the various toolpath generation techniques and any other relevant aspect of the CNC machining exercise

5 References

- i. Fundamentals of CNC Machining by Autodesk: A Practical Guide for Beginners by Autodesk
- ii. XYZ 1510 VMC 4-axes CNC milling machine operator's manual
- iii. CNC Machining Handbook: Building, Programming, and Implementation (2010) by Alan Overby

