

## Sub-subject: Manufacturing Processes

### Topic: CNC Milling and G-Code Programming

#### Given and Introduction

The task is to write a G-code program using a CNC milling machine for machining the part shown in the figure. Sub-programming commands M98 and M99 will be used for pocket cutting, with a maximum depth of cut set at 5 mm.

#### Steps and Explanations

```
O0001
G00 G90 G21 G17 G40 G49 G80 ; Initialize system
T1 M06 ; Tool change to Tool 1
G54 ; Use work coordinate system 1
G43 H01 Z50.0 M08 ; Tool length offset and coolant on
```

#### Explanation:

- O0001 starts the main program.
- G00 G90 G21 G17 G40 G49 G80 set various modes:
  - G00 for rapid positioning
  - G90 for absolute positioning
  - G21 for metric units
  - G17 for XY plane selection
  - G40 for cutter radius compensation off
  - G49 for cancel fix tool length offset
  - G80 for cancel canned cycle
- T1 M06 changes to Tool 1.
- G54 selects work coordinate system 1.
- G43 H01 Z50.0 M08 applies tool length offset, positions tool to Z50.0, and turns coolant on.

#### Main Contour (Primary Machining)

```
G00 X0.0 Y0.0 ; Move to starting position
G01 Z-5.0 F200 ; Linear interpolation to depth of cut
G01 X100 Y0.0 F200 ; Linear interpolation to next point
G01 X100 Y50.0 ; Linear interpolation to next point
G01 X50 Y50.0 ; Linear interpolation to next point
G01 X50 Y100.0 ; Linear interpolation to final point
G01 X0.0 Y100.0 ; Linear interpolation to next point
G01 X0.0 Y0.0 ; Linear interpolation to start / closing the contour
```

#### Explanation:

- G00 X0.0 Y0.0 moves to starting position.
- G01 Z-5.0 F200 linearly interpolates to depth of cut at feed rate 200.
- G01 commands move the tool along the specified path to machine the part contours.

#### Sub-Program Call for Pocket Cutting

```
M98 P0002 L2 ; Call sub-program 00002 twice for pocketing
G00 Z50.0 ; Retract the tool to Z50.0
M09 ; Coolant off
G91 G28 Z0.0 ; Move tool to reference point
M30 ; End of program

; Sub Program for Pocket Cutting
O0002 ; Start of sub-program
G00 X40 Y60 ; Move to starting position of pocket
G01 Z-5.0 F100 ; Linear interpolation to depth of cut
G01 X90 Y60 F200 ; Machine the rectangle pocket
G01 X90 Y90
G01 X40 Y90
G01 X40 Y60
G00 Z50.0 ; Retract the tool to Z50.0
M99 ; Return from sub-program
```

### Explanation:

- M98 P0002 L2 calls sub-program O0002, twice for pocketing.
- G00 Z50.0 retracts the tool after pocket cutting.
- M09 turns off coolant.
- G91 G28 Z0.0 moves the tool to the reference point (tool change position).
- M30 ends the main program.
- O0002 starts the sub-program.
- G00 X40 Y60 moves to the starting position of the pocket.
- G01 Z-5.0 F100 plunges to pocket depth.
- G01 commands move the tool along paths to cut the pocket.
- G00 Z50.0 retracts the tool after machining the pocket.
- M99 returns to the main program after sub-program execution.

---

### Final Solution

The correctly structured G-code for machining the given part using a CNC milling machine with sub-program calls for pocket cutting is provided above. This includes all the set-up commands, primary contour milling, sub-program calls, and proper tool movements and configurations.

...