

## Multi-Step Pocket

- ➡ The complete program solution uses one main program and one subprogram - one tool illustrated:

```
(39-02.NC)
(T01 - 1.0 DIA CARBIDE END MILL - CENTER CUTTING)
(X0Y0 CENTER OF POCKET - Z0 TOP OF PART)
(T01 IN SPINDLE AFTER SETUP)

N1 G20                                (ENGLISH UNITS)
N2 G17 G40 G80                        (COMMON CANCELLATIONS)
N3 G90 G54 G00 X0 Y0 S1500 M03        (MOTION TO THE POCKET CENTER + SPEED)
N4 G43 Z1.0 H01 M08                  (STARTING CLEARANCE + OFFSET + COOLANT)
N5 G01 Z0.05 F50.0                   (SAFE TOOL CLEARANCE)
N6 Z0 F5.0                            (MANDATORY START LOCATION FOR SUBPROGRAM)
N7 M98 P3952 L12                      (12 TIMES 0.025 DEPTH = 0.3 FULL DEPTH)
N8 G90 G00 Z1.0 M09                  (REINSTATE ABSOLUTE MODE)
N9 G28 Z1.0 M05                       (MACHINE RETURN Z AXIS)
N10 G28 X0 Y0                         (MACHINE RETURN XY AXES)
N11 M30                               (END OF PROGRAM)
%
```

```
O3952                                (SUBPROGRAM FOR 0.025 DEPTH OF CUT)
N101 G91 G01 Z-0.025 F5.0             (INCREMENTAL SINGLE DEPTH OF 0.025)
N102 G41 X-0.75 Y-0.125 D01 F12.0    (LINEAR LEAD-IN TO 0.75 RADIUS)
N103 G03 X0.75 Y-0.75 I0.75 J0       (0.75 LEAD-IN RADIUS USED)
N104 J-0.875                          (FULL CIRCLE POCKET)
N105 X0.75 Y0.75 I0 J0.75            (0.75 LEAD-OUT RADIUS)
N106 G40 G01 X-0.75 Y0.125           (LINEAR LEAD-OUT FROM 0.75 RADIUS)
N107 M99                             (END OF SUBPROGRAM)
%
```

- ➡ Answers to questions:

- The minimum tool diameter must be one third (1/3) of the pocket diameter, therefore  $1.75/3 = 0.58333$ . This is the smallest cutting tool that should clean the bottom of the pocket, at least in theory. An end mill  $\varnothing 5/8$  or larger will be a better choice.
- The type of the cutting tool used must be a **center cutting end mill** (also known as the **slot drill**). As this is a solid pocket, a standard end mill will not be able to penetrate into the depth by plunging along the Z-axis. Ramping or helical entry is also possible with non-centercutting end mills.
- Yes, the  $\varnothing 1.75$  circle can be programmed in a single block on most CNC machines. However, the address I and/or J must always be used. Even if other blocks in the program use the R-address, R cannot be used for a circle (360° arc).
- The minimum number of depth passes is twelve - 0.3 total pocket depth divided by 0.025 each depth = 12. The subprogram has to be repeated twelve times. Note the L12 (or K12) repetition count in the block N7.
- If the start position for the circular pocket is Z0.1, and the depth of each cut is 0.025 with the final depth of 0.3, the number of repeats would be sixteen. The reason is that the tool starts at Z0.1 and ends at Z-0.3. The total distance between these points is 0.4. As each depth of cut is only 0.025, that means the total number of depth passes is  $0.4/0.025 = 16$ . Starting above Z0 often mean cuts in the air !
- Starting at Z0, the number of cutting passes with the 0.05 depth will be  $0.3/0.05 = 6$ . Starting at Z0.1, the number of cutting passes will be  $0.4/0.05 = 8$ . Starting above Z0 may mean cuts in the air !