

Four Bolt Circles

There are two common solutions to this project. One uses the incremental method, which is commonly used when repeating a machining pattern transferred from one location to another location. The other method is to write the subprogram in absolute mode, which is not as common, but provides identical results with a better program structure. The choice is only a matter of preference by the programmer. Both methods of solution are presented here for comparison and further study. Other projects in this chapter offer additional methods.

```
O3901 (39-01.NC - VERSION 1)
(T05 - 0.5 DIA SPOT DRILL)
N1 G20
N2 G17 G40 G80
N3 G90 G54 G00 X1.5 Y0.75 S2139 M03 T19      (MANDATORY MOVE TO HOLE H1 OF PATTERN A)
N4 G43 Z1.0 H05 M08
N5 G99 G82 R0.1 Z-0.0745 P200 F6.0          (SPOT DRILL HOLE H1 OF PATTERN A)
N6 M98 P7001                                (SPOT DRILL 5 MORE HOLES OF PATTERN A)
N7 X3.5 M98 P7001                            (SUBPROGRAM CALL FOR PATTERN B)
N8 Y2.25 M98 P7001                            (SUBPROGRAM CALL FOR PATTERN C)
N9 X1.5 M98 P7001                            (SUBPROGRAM CALL FOR PATTERN D)
N10 G80 Z1.0 M09
N11 G28 Z1.0 M05
N12 M01

(T19 - 0.125 DIA DRILL)
N13 T19
N14 M06
N15 G90 G54 G00 X1.5 Y0.75 S2139 M03 T05     (MANDATORY MOVE TO HOLE H1 OF PATTERN A)
N16 G43 Z1.0 H19 M08
N17 G99 G81 R0.1 Z-0.3375 F6.0               (SPOT DRILL HOLE H1 OF PATTERN A)
N18 M98 P7001                                (SPOT DRILL 5 MORE HOLES OF PATTERN A)
N19 X3.5 M98 P7001                            (SUBPROGRAM CALL FOR PATTERN B)
N20 Y2.25 M98 P7001                            (SUBPROGRAM CALL FOR PATTERN C)
N21 X1.5 M98 P7001                            (SUBPROGRAM CALL FOR PATTERN D)
N22 G80 Z1.0 M09
N23 G28 Z1.0 M05
N24 G91 G28 X0 Y0
N25 M30
%

O7001 (SUBPROGRAM FOR O3901 - INCREMENTAL MODE)
(TOOL EXPECTED AT HOLE H1)
N701 G91 X-0.25 Y0.433                      (FROM H1 TO H2)
N702 X-0.5                                  (FROM H2 TO H3)
N703 X-0.25 Y-0.433                        (FROM H3 TO H4)
N704 X0.25 Y-0.433                        (FROM H4 TO H5)
N705 X0.5                                  (FROM H5 TO H6)
N706 G90                                    (REINSTATE ABSOLUTE MODE)
N707 M99                                    (RETURN TO THE MAIN PROGRAM)
%
```

The previous solution is a common one, but has a small flaw - *the subprogram structure is fragmented*.

Study the program carefully. The first hole *H1* is machined within the main program, whereby the remaining five holes are machined from within the subprogram. This is only an acceptable method, not a preferred method. An improvement can be made by moving the tool to the center of each bolt circle in the main program, and use the *L0* feature in the fixed cycle call. This method is not demonstrated here, but well worth further study. Programming the *L0* method is used in the absolute version of the exercise 39-01. Note that some controls use *K* instead of *L*.

The absolute method offers a much improved structure, since the *X* and *Y* coordinates indicate tool locations, not tool motions. Storing all six hole locations in a subprogram is easier to manage, but also more demanding. The programmer should be familiar with the *G52* command (*Local Coordinate System*), and the *L0* (or *K0*) fixed cycle parameter, which are the keys to the absolute version of the solution.

```
O3901 (39-01.NC - VERSION 2)
(T05 - 0.5 DIA SPOT DRILL)
N1 G20
N2 G17 G40 G80
N3 G90 G54 G00 X1.0 Y0.75 S2139 M03 T19      (CENTER OF PATTERN A - MANDATORY MOVE)
N4 G43 Z1.0 H05 M08
N5 G99 G82 R0.1 Z-0.0745 P200 F6.0 L0        (DATA SAVED - NO MACHINING)
N6 M98 P7001                                  (SPOT DRILL 6 HOLES OF PATTERN A)
N7 G52 X3.0 Y0.75                            (TEMPORARY PART ZERO SHIFT TO PATTERN B)
N8 M98 P7001                                  (SUBPROGRAM CALL FOR PATTERN B)
N9 G52 X3.0 Y2.25                            (TEMPORARY PART ZERO SHIFT TO PATTERN C)
N10 M98 P7001                                 (SUBPROGRAM CALL FOR PATTERN C)
N11 G52 X1.0 Y2.25                           (TEMPORARY PART ZERO SHIFT TO PATTERN D)
N12 M98 P7001                                 (SUBPROGRAM CALL FOR PATTERN D)
N13 G52 X0 Y0                                (CANCEL LOCAL COORDINATE SYSTEM)
N14 G80 Z1.0 M09
N15 G28 Z1.0 M05
N16 M01

(T19 - 0.125 DIA DRILL)
N17 T19
N18 M06
N19 G90 G54 G00 X1.5 Y0.75 S2139 M03 T05     (MANDATORY MOVE TO HOLE H1 OF PATTERN A)
N20 G43 Z1.0 H19 M08
N21 G99 G81 R0.1 Z-0.3375 F6.0 L0            (DATA SAVED - NO MACHINING)
N22 M98 P7001                                  (DRILL 6 HOLES OF PATTERN A)
N23 G52 X3.0 Y0.75                            (TEMPORARY PART ZERO SHIFT TO PATTERN B)
N24 M98 P7001                                  (SUBPROGRAM CALL FOR PATTERN B)
N25 G52 X3.0 Y2.25                            (TEMPORARY PART ZERO SHIFT TO PATTERN C)
N26 M98 P7001                                  (SUBPROGRAM CALL FOR PATTERN C)
N27 G52 X1.0 Y2.25                           (TEMPORARY PART ZERO SHIFT TO PATTERN D)
N28 M98 P7001                                  (SUBPROGRAM CALL FOR PATTERN D)
N29 G52 X0 Y0                                (CANCEL LOCAL COORDINATE SYSTEM)
N30 G80 Z1.0 M09
N31 G28 Z1.0 M05
N32 G91 G28 X0 Y0
N33 M30
%

O7001 (SUBPROGRAM FOR O3901 - ABSOLUTE)
(G52 IS MANDATORY IN THE MAIN PROGRAM)
N701 X0.5                                     (HOLE H1 LOCATION)
N702 X0.25 Y0.433                           (HOLE H2 LOCATION)
N703 X-0.25                                  (HOLE H3 LOCATION)
N704 X-0.5 Y0                                (HOLE H4 LOCATION)
N705 X-0.25 Y-0.433                         (HOLE H5 LOCATION)
N706 X0.25                                   (HOLE H6 LOCATION)
N707 M99                                     (RETURN TO THE MAIN PROGRAM)
%
```

There are additional variations available, not possible to include here. The two programming methods described here are the most common. For example, the *L0/K0* method can also be combined with the incremental method, and the overall program structure would be improved even more. Also keep in mind that ultimately, it is the CNC programmer who determines the particular programming method over an alternative.

Subprograms are very powerful and - more often than not - offer more than one method of determining the final programmed toolpath.