36-02 Solution

Repeating Grove

The instructions for this project specifically asked for the grooving to be done in the incremental mode. Programming the grooves in the absolute mode is certainly possible, but not justifiable in any circumstances. The first program is for the groove only - note that the tool motions are identical for all three grooves (common tool motions). When compared to a subprogram version, the subprogram stores these identical (common) motions, and the main program just calls the subprogram at the proper groove location. In both cases, each groove must start at the same relative position.

```
(36-02 - REPEATING GROOVE - GROOVE ONLY PROGRAM)
(GROOVE CUTTING FROM CENTER OF GROOVE AND 0.05 ABOVE GROOVE)
(T06 - 0.125 WIDE GROOVING TOOL - LEFT CORNER TIP)

N1 T0600
N2 G96 S500 M03
N3 G00 X1.4 Z-0.2835 T0606 M08          (Z IS 0.135+0.172/2+0.125/2)
N4 G01 X1.1 F0.02                       (X IS 0.05 ABOVE THE FIRST GROOVE)
(FIRST GROOVE)
N5 G01 U-0.395 F0.002
N6 G00 U0.395
N7 W-0.0935
N8 G01 U-0.14 W0.07 F0.001
N9 U-0.255 F0.003
N10 U0.395 W0.0235 F0.02
N11 G00 W0.0935
N12 G01 U-0.14 W-0.07 F0.001
N13 U-0.26 F0.003
N14 W-0.047
N15 U0.4 W0.0235 F0.02                  (FIRST GROOVE)
N16 G00 X1.225                          (X IS 0.05 ABOVE THE SECOND GROOVE)
N17 Z-0.6835                            (Z IS 0.535+0.172/2+0.125/2)
(SECOND GROOVE)|
N18 G01 U-0.395 F0.002
N19 G00 U0.395
N20 W-0.0935
N21 G01 U-0.14 W0.07 F0.001
N22 U-0.255 F0.003
N23 U0.395 W0.0235 F0.02
N24 G00 W0.0935
N25 G01 U-0.14 W-0.07 F0.001
N26 U-0.26 F0.003
N27 W-0.047
N28 U0.4 W0.0235 F0.02
N29 G00 X1.35                           (X IS 0.05 ABOVE THE THIRD GROOVE)
N30 Z-1.0685                            (Z IS 0.92+0.172/2+0.125/2)
(THIRD GROOVE)
N31 G01 U-0.395 F0.002
N32 G00 U0.395
N33 W-0.0935
N34 G01 U-0.14 W0.07 F0.001
N35 U-0.255 F0.003
N36 U0.395 W0.0235 F0.02
N37 G00 W0.0935
N38 G01 U-0.14 W-0.07 F0.001
N39 U-0.26 F0.003
N40 W-0.047
N41 U0.4 W0.0235 F0.02
N42 G00 X1.7                            (CLEAR ABOVE THE PART)
N43 G00 G40 X5.0 Z4.0 T0600 M09         (TOOL CHANGE POSITION)
N44 M30
%
```

➲ The complete program adds a few more tools and uses a subprogram for the grooves:

```
(36-02 - REPEATING GROOVE - COMPLETE PROGRAM)
(FANUC 10T CONTROL)

(T03 - 55 DEG FACE AND TURN TOOL - 0.031 TLR - TIP NUMBER 3)
(T01 - 35 DEG FINISHING TOOL - 0.015 TLR - TIP NUMBER 3)
(T06 - 0.125 WIDE GROOVING TOOL)
(T05 - 0.125 WIDE PART-OFF TOOL - LEFT CORNER TIP)

N1 G20 G50 S3500 T0300                  (T03 - 55 DEG FACE AND TURN TOOL)
N2 G96 S500 M03
N3 G00 X1.7 Z0 T0303 M08
N4 G01 X-0.07 F0.005
N5 G00 Z0.1
N6 G42 X1.55
N7 G71 P8 Q17 U0.04 W0.005 D1000 F0.012 (CAN BE CHANGED TO A TWO-BLOCK INPUT)
N8 G00 X0.76
N9 G01 X1.0 Z-0.02 F0.002
N10 Z-0.4 F0.004
N11 X1.125 K-0.02 F0.002
N12 Z-0.8 F0.004
N13 X1.25 K-0.02 F0.002
N14 Z-1.225 F0.004
N15 X1.375 K-0.02 F0.002
N16 Z-1.625 F0.004
N17 X1.7
N18 G00 G40 X5.0 Z4.0 T0300 M09
N19 M01

N20 G50 S3500 T0100                     (T01 - 35 DEG FINISHING TOOL)
N21 G96 S650 M03
N22 G00 G42 X1.7 Z0.1 T0101 M08
N23 G70 P8 Q17
N24 G00 G40 X5.0 Z4.0 T0100 M09
N25 M01

N26 T0600                               (T06 - 0.125 WIDE GROOVING TOOL)
N27 G96 S500 M03
N28 G00 X1.4 Z-0.2835 T0606 M08         (Z IS 0.135+0.172/2+0.125/2)
N29 G01 X1.1 F0.02                      (X IS 0.05 ABOVE THE FIRST GROOVE)
N30 M98 P5001                           (FIRST GROOVE)
N31 G00 X1.225                          (X IS 0.05 ABOVE THE SECOND GROOVE)|
N32 Z-0.6835                            (Z IS 0.535+0.172/2+0.125/2)
N33 M98 P5001                           (SECOND GROOVE)
N34 G00 X1.35                           (X IS 0.05 ABOVE THE THIRD GROOVE)
N35 Z-1.0685                            (Z IS 0.92+0.172/2+0.125/2)
N36 M98 P5001                           (THIRD GROOVE)
N37 G00 X1.7                            (CLEAR ABOVE THE PART)
N38 G00 G40 X5.0 Z4.0 T0600 M09         (TOOL CHANGE POSITION)
N39 M01

N40 T0500                               (T05 - PART-OFF TOOL)
N41 G97 S1700 M03
N42 G00 Z-1.63 T0505 M08
N43 X1.55
N44 G01 X1.1 F0.003
N45 G00 X1.475
N46 G01 Z-1.555                         (G01 W0.075 CAN BE USED INSTEAD)
N47 X1.335 Z-1.625 F0.002               (U-0.14 W-0.07 CAN BE USED INSTEAD)
N48 X-0.02
N49 G00 X5.0
N50 Z4.0 T0500 M09
N51 M30
%
```

```
(SUBPROGRAM FOR GROOVE)
O5001 (GROOVE CUTTING FROM CENTER OF GROOVE AND 0.05 ABOVE GROOVE)
N1 G01 U-0.395 F0.002
N2 G00 U0.395
N3 W-0.0935
N4 G01 U-0.14 W0.07 F0.001
N5 U-0.255 F0.003
N6 U0.395 W0.0235 F0.02
N7 G00 W0.0935
N8 G01 U-0.14 W-0.07 F0.001
N9 U-0.26 F0.003
N10 W-0.047
N11 U0.4 W0.0235 F0.02
N12 M99
%
```

Note that the complete program with a subprogram is only marginally longer than the grooving tool alone in the first example. Additional benefits of using subprogram include easier changes and groove relocation, if necessary.