30-01 Solution

Toolpath Methods

When programming manually, the most common method of programming a toolpath is to program to the drawing dimensions, as if the cutter radius were zero. The programmer enters G41 or G42 preparatory command, along with the *D* offset number. At the machine - during setup - the CNC operator enters the actual tool radius amount into the offset register specified by the *D* number in the part program. This practice is represented by the project *30-01B*. If the *D* offset is omitted or set to zero, the center of the cutter will follow the drawing contour and the result is a scrap. See conclusion at the end.

➲ Project 30-01A:

```
N1 G21
N2 G17 G40 G80
N3 G90 G54 G00 X-10.0 Y-10.0 S1200 M03     (POSITION A)
N4 G43 Z2.5 H01 M08
N5 G01 Z-5.0 F200.0
N6 X3.0                                    (POSITION B)
N7 Y72.0                                   (POSITION C)
N8 X97.0                                   (POSITION D)
N9 Y3.0                                    (POSITION E)
N10 X-10.0                                 (POSITION F)
N11 G00 Y-10.0                             (POSITION A)
N12 G28 Z2.5
N13 G28 X-10.0 Y-10.0
N14 M30
%
```

1. The center of the cutter will follow the programmed toolpath.
   As no compensation for the cutter radius was applied, the part will be a scrap.
   D51 setting is irrelevant, as no cutter radius offset was programmed.
2. ADVANTAGES: NONE
3. DISADVANTAGES: Final contour will not be correct, part will be a scrap. Its length and width will be shorter by the cutter diameter

➲ Project 30-01B:

```
N1 G21
N2 G17 G40 G80
N3 G90 G54 G00 X-10.0 Y-10.0 S1200 M03     (POSITION A)
N4 G43 Z2.5 H01 M08
N5 G01 Z-5.0 F200.0
N6 G41 X3.0 D51                            (POSITION B - OFFSET APPLIED)
N7 Y72.0                                   (POSITION C)
N8 X97.0                                   (POSITION D)
N9 Y3.0                                    (POSITION E)
N10 X-10.0                                 (POSITION F)
N11 G00 G40 Y-10.0                         (POSITION A - OFFSET CANCELED)
N12 G28 Z2.5
N13 G28 X-10.0 Y-10.0
N14 M30
%
```

1. Although the program uses the same coordinates as in the 30-01A program, the finished part will be correct, because the cutter radius offset is used. The program coordinates apply to the drawing dimensions, so the offset setting must contain the radius of the cutter, which is 7.5 mm - D51=7.500
2. ADVANTAGES: Easy for manual programming, adjustments at the machine are possible
3. DISADVANTAGES: NONE

➲ Project 30-01C:

```
N1 G21
N2 G17 G40 G80
N3 G90 G54 G00 X-10.0 Y-10.0 S1200 M03     (POSITION A)
N4 G43 Z2.5 H01 M08
N5 G01 Z-5.0 F200.0
N6 X-4.5                                   (POSITION B)
N7 Y79.5                                   (POSITION C)
N8 X104.5                                  (POSITION D)
N9 Y-4.5                                   (POSITION E)
N10 X-10.0                                 (POSITION F)
N11 G00 Y-10.0                             (POSITION A)
N12 G28 Z2.5
N13 G28 X-10.0 Y-10.0
N14 M30
%
```

1. In this example, the XY coordinates apply to the center of the cutter. That means the cutter radius has been applied at the time of programming. Ideally, the finished part should be correct, because of the cutter radius application in the program. D51 setting is irrelevant, as no cutter radius offset was programmed.
2. ADVANTAGES: No need to worry about offset setting
3. DISADVANTAGES: No adjustment of dimensions or fine tuning is possible at the CNC machine, even with high precision tool diameter, there is no flexibility to maintain tolerances. XY coordinates do not reflect drawing dimensions

➲ Project 30-01D:

```
N1 G21
N2 G17 G40 G80
N3 G90 G54 G00 X-10.0 Y-10.0 S1200 M03     (POSITION A)
N4 G43 Z2.5 H01 M08
N5 G01 Z-5.0 F200.0
N6 G41 X-4.5 D51                           (POSITION B - OFFSET APPLIED)
N7 Y79.5                                   (POSITION C)
N8 X104.5                                  (POSITION D)
N9 Y-4.5                                   (POSITION E)
N10 X-10.0                                 (POSITION F)
N11 G00 G40 Y-10.0                         (POSITION A - OFFSET CANCELED)
N12 G28 Z2.5
N13 G28 X-10.0 Y-10.0
N14 M30
%
```

1. This last example also shows toolpath to the center of the cutter.
   Because the toolpath is already compensated, the amount of D51 will be zero, under ideal conditions - D51 = 0.000
2. ADVANTAGES: No need to worry about offset setting, adjustment of dimensions or fine tuning is possible at the machine, useful in CAD/CAM programming
3. DISADVANTAGES: Does not follow traditional format of manual programming, XY coordinates do not reflect drawing dimensions, but that does mean a disadvantage in this method

➲ CONCLUSION

There is a great difference whether the G41/G42 cutter radius offset is programmed or not. As these four projects show, project *30-01A* should be rejected immediately, because it does not represent correct program. Project *30-01C* is correct, but should also be rejected as an option, because it lacks possibility of adjustments at the machine. From the two remaining projects, project *30-01B* represents the traditional and very common method of manual programming, whereby project *30-01D* is often used in toolpath generation using CAD/CAM, although it can be used for manual method as well, to guarantee consistency.