

G78 - Rough Contour Cycle

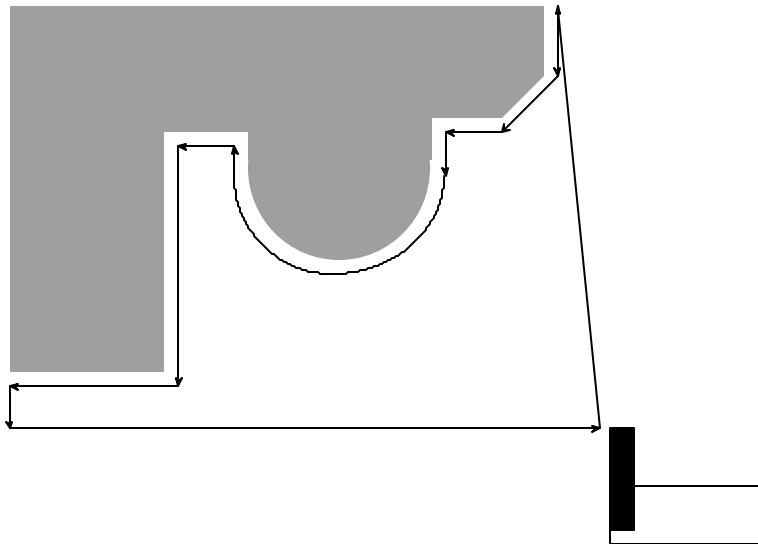
G78 is the start of a rough contour cycle. This cycle serves to rough a contour based on a section of program code describing a finish contour.

G78UnFnPn

Un is the amount to be left on the part for the a finish pass, (amount per side)

Fn is the feed rate

Pn (optional) is a subroutine number



The box cycle starts at the current position, then makes a cutting pass parallel to the final contour, but away from the part at a distance Un. At the end of the cycle, the tool is returned to the start point.

The rules and general usage commands are the same as with the G75 command. Please refer to that command for these notes.

Note about finish passes, subroutines, and tool nose radius:

Tool nose radius compensation can not be used within a subroutine. So if you need to use TNR comp for the finish pass you will have to copy the contour pass into the program for the last pass.

Worked example for G78

The following code is a finish pass for the same example for G75. Please refer there for the part layout. In this example we are using a different tool to take the finish contour pass, T2.

```
G90G94F300G73  
M03S2500  
T2(LH Finish TURN TOOL)  
X.8Z.1  
G95F.003  
G78U.01F.003  
X0Z0  
X.343C.125  
Z-.5  
X.406  
G02X.406Z-.687R.0935  
X.375  
Z-1  
X.75  
RF  
G00Z1  
M30
```

The same example shown using two tools, three passes with two types of cycle and subroutines.

The subroutine is used so that there is only one contour pass in the program. If there are any changes to the finish pass they can be done in one place and then the rest of the program is also changed.

```
G90G94F300G73
M03S2500
T1(LH TURN TOOL).....Call roughing tool
X0Z1
X.14Z.1
G95F.003
G75I.05U.04F.003P1 .....Box rough cycle
G00Z1
T2(LH FINISH TURN TOOL) .....Call finish tool
X0Z1
X0 Z.1
G95F.003
G78U.01F.003P1 .....Rough contour cycle
G00Z.1
X0
G95F.002
P1 .....Finish pass to depth with same tool
G00Z1
X-1
M30
}1
X0Z0
X.343C.125
Z-.5
X.406
G02X.406Z-.687R.0935
X.375
Z-1
X.75
M99
```