G75 - Box Contour Roughing Cycle

G75 is the start of a box contour cycle. This cycle serves to rough out an area bounded in part by a contour defined in the part program
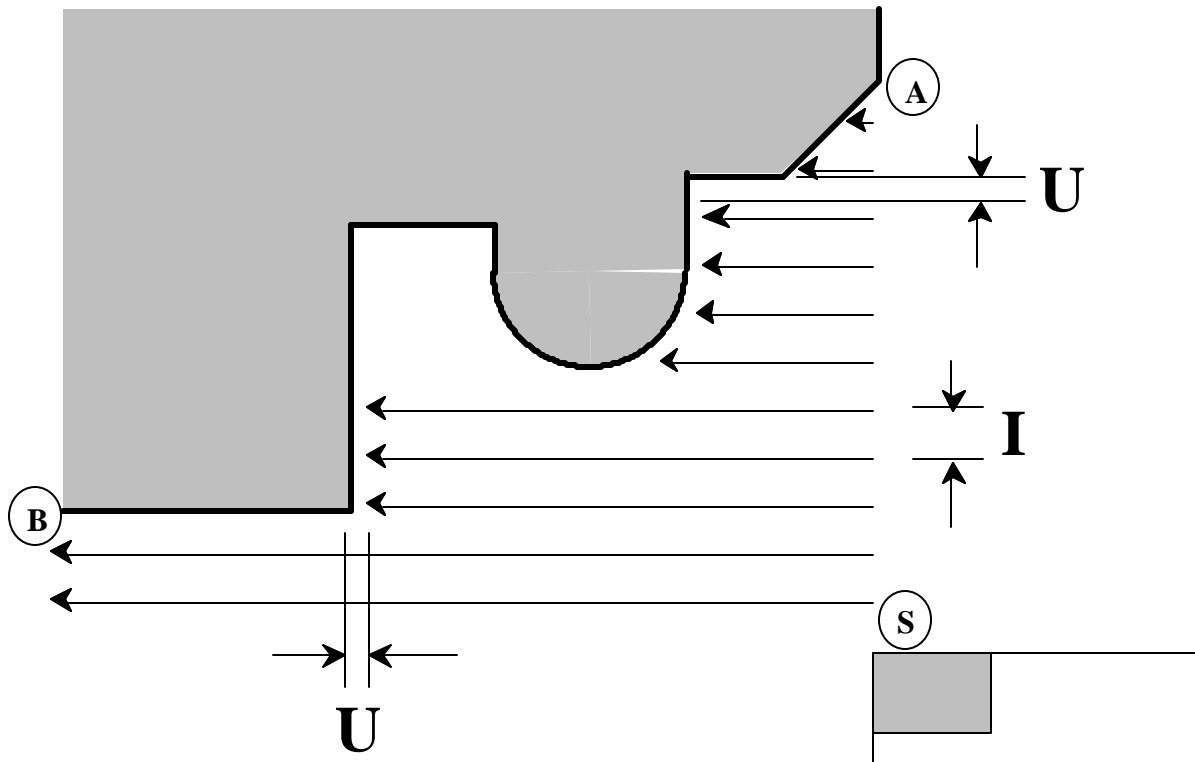
G75InUnFnPn

Un is the amount to be left on the part for the a finish pass
In is the maximum amount to be roughed per pass, defined as the depth of cut per side
Fn is the feed rate
Pn (optional) is a subroutine number

The box cycle starts at the current position, then makes cutting passes parallel to the Z axis at a cutting depth no greater than the I value until the last pas which is a U amount off the part. The area of material that is removed is bounded by the Z and X axis through the tool start point and the contour from A to point B. At the end of the cycle, the tool is returned to the start point.

The location of the contour code to rough to:

- *If the P code is present*, the contour is defined in a subroutine n.
- *If the P code is not present*, the contour is defined in the following blocks until a block with an "RF" code is encountered

The feed rate is FPM (G94) or IPM (G95), depending on the mode of the control when the cycle is started.

The RF code must be on a line by itself.

The return passes are at a fixed clearance distance (.02") from the last cutting pass.

The G75 cycle can be used for either internal or external removal, and from the back.

Internal format   External from rear format
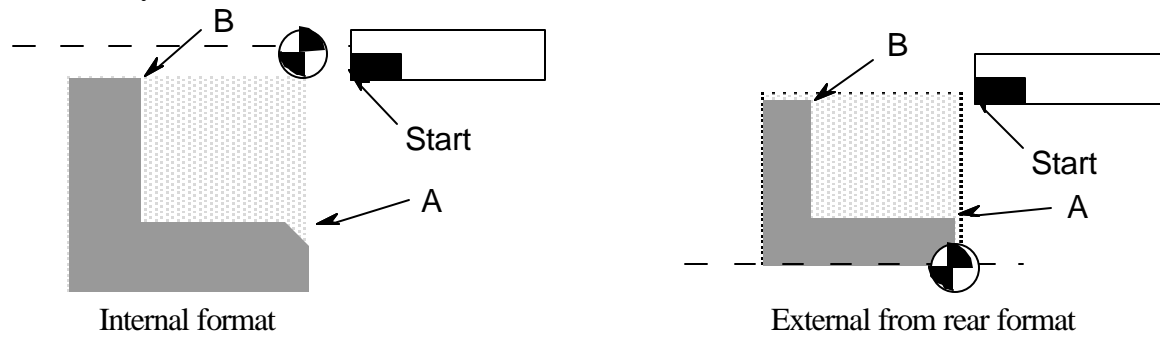
The G74 cycle can be used in radius (G73) or diameter (G72) mode. In both cases the In value is the amount to be taken off on a side. An In value of .05 would take .1 off the diameter of the part with each pass.

Tool nose radius compensation **can not** be active when using the roughing cycle.

In calculation of the contour, the control honors those codes which affect the geometry of the contour (presets, absolute/incremental, etc) but ignores any other information which might be present (T & D codes, M codes, feedrates, etc)

An error will be declared if the start and end point of the contour (A & B) are not in the same quadrant relative to the start point (S)

A

This is not allowed. The A point is
not in the same quadrant as B relative
to the start point.

B

S
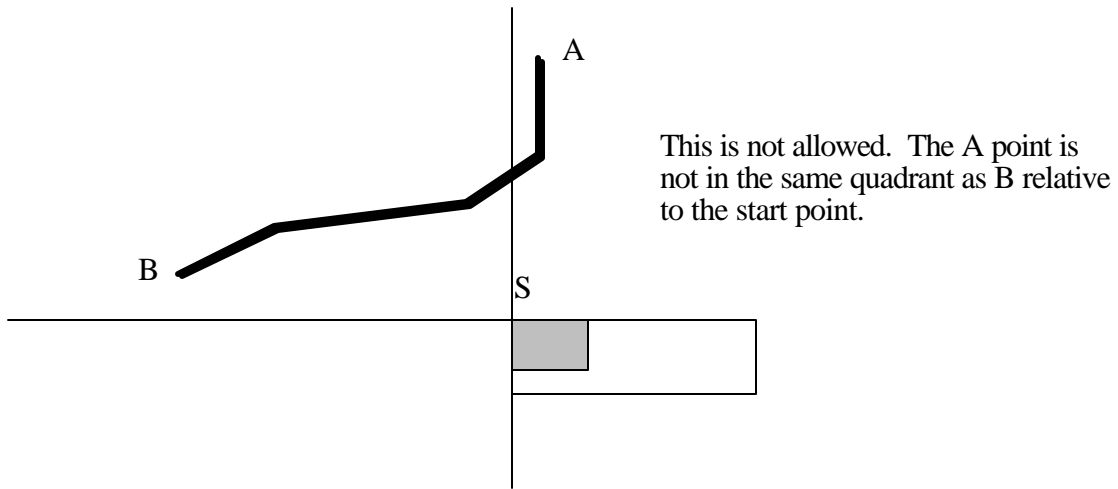
If the start point is moved over to include both A and B the cycle will work.

A

This is allowed. The A and B are in the
same quadrant relative to the start point

B

S
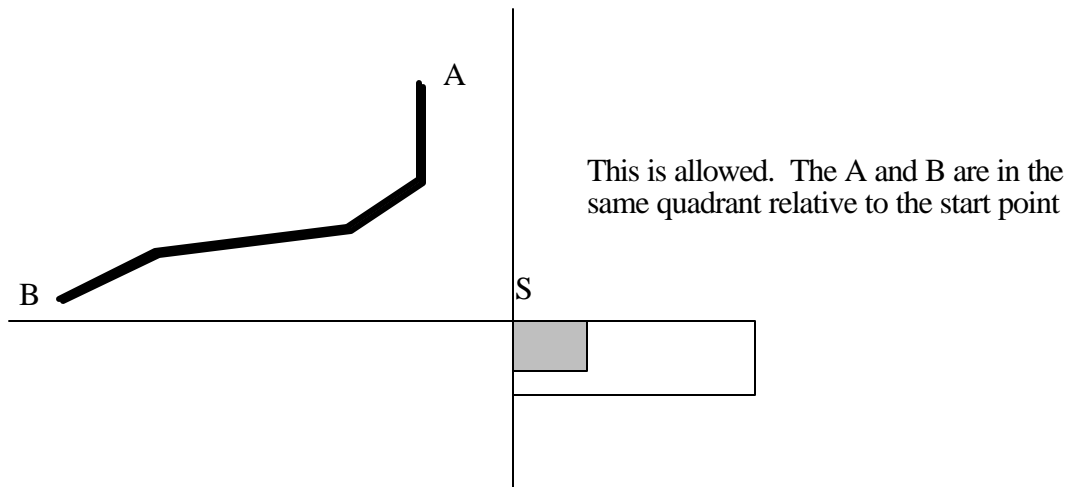
Worked external examples for G75
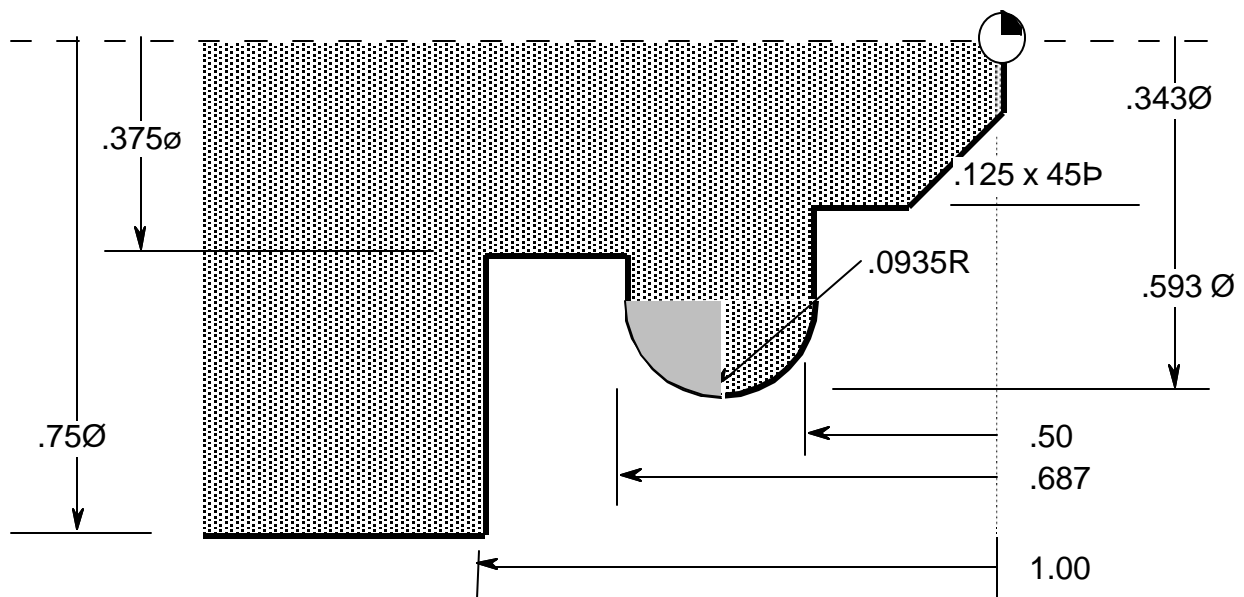
Above is a part that can be roughed. The part is defined in radius values in X. The code that can be used to generate the contour could be:

X0Z0
X.343C.125
Z-.5
X.406
G02X.406Z-.687R.0935
X.375
Z-1
X.75

The first example has the finished contour to be roughed immediately after the G75 command. This program would just rough the part and stop.

Written in Diameter mode	Written in Radius mode
G90G94F300G72	G90G94F300G73
M03S2500	M03S2500
T1(LH TURN TOOL)	T1(LH TURN TOOL)
X0Z1	X0Z1
X.8Z.1	X.4Z.1
G95F.003	G95F.003
G75I.05U.02F.003	**G75I.05U.02F.003**
X0Z0	X0Z0
X.343C.125	X.1723C.125
Z-.5	Z-.5
X.406	X.203
G02X.406Z-.687R.0935	G02X.203Z-.687R.0935
X.375	X.1875
Z-1	Z-1
X.75	X.375
RF	**RF**
M05	M05
G00Z1	G00Z1
X-1	X-.5
M30	M30

In this example the contour code is given as a subroutine. You can use the same subroutine for the G75 and G78 roughing passes. If you have to make a correction to the contour, it can be done in one place instead of for each cycle. You can also use it for the finishing pass if you are not going to use TNR compensation. If you want to use the compensation then you must leave the finish pass out of the subroutine since they will not accept the compensation.

G90G94F300G72
M03S2500
T1(LH TURN TOOL)
X0Z1
X.4Z.1
G95F.003
G75I.05U.02F.003P1
M05
G00Z1
X-1
M30
}1
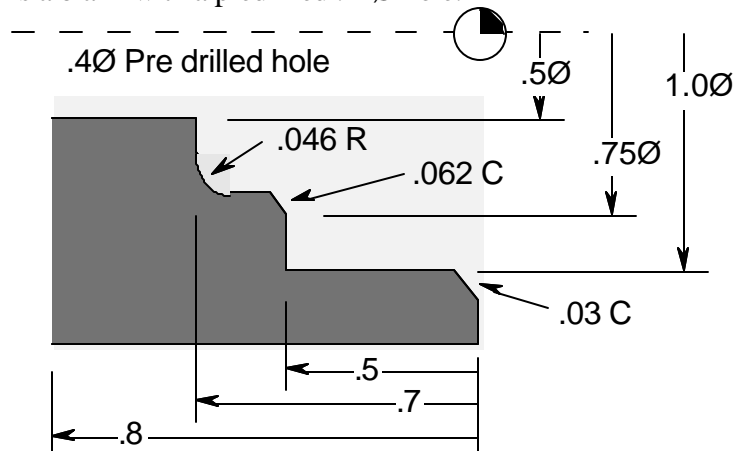X0Z0

X.343C.125
Z-.5
X.406
G02X.406Z-.687R.0935
X.375
Z-1
X.75
M99

Worked internal example for G75

In this example there is a blank with a predrilled .4"Ø hole.

.4Ø Pre drilled hole
.046 R
.062 C
.5Ø
1.0Ø
.75Ø
.03 C
.5
.7
.8

Notice the starting point is at the minor diameter of the finished bore, and the A and B points are at the starting and ending points of the finish contour. Also notice that the finish contour does not use C for the .03" chamfer. If this were needed in this box cycle you could add a facing move and chamfer to the contour pass.

B
Start
A
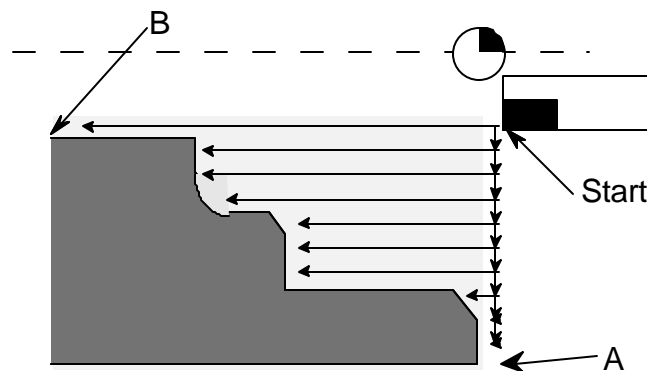
G90G94F300G72
M03S2000
T1
X.4Z.1
G95F.003
G75I.05U.02F.003
X1.2Z0
X1.06
X1Z-.03
Z-.5
X.75C.062
Z-.704
G03X.658Z-.75R.046

```
X.5
Z-1.
RF
G00Z1
M30
```