



PA - SOFTWARE
Turning Cycles
PA 8000

401309

Edition

2.00

Revision

5

Copyright

PA

SUBJECT TO TECHNICAL MODIFICATIONS AND ERRORS



Contents

1 Stock removal cycles	1
1.1 General.....	1
1.2 G271 Stock removal in turning.....	1
1.2.1 Syntax.....	1
1.2.2 Example.....	2
1.2.3 Direction of allowance.....	4
1.2.4 Effective G-codes	4
1.3 G272 Stock removal in facing	5
1.3.1 Syntax.....	5
1.3.2 Example.....	6
1.4 Direction of allowance	7
1.4.1 Effective G-codes	8
1.5 G270 Finishing Cycle	8
1.5.1 Syntax.....	8
1.5.2 Example.....	9
1.6 Error messages	10
1.7 Part program display	10
1.8 Machine parameter table	12

1 Stock removal cycles

1.1 General

The stock removal cycles provide an easy way of roughly turning. The CNC programmer only has to program the desired shape. The CNC then creates a multiple repetitive cycle for stock removal and roughly turning of the shape.

The programmed tool path can also be used for the finishing cut with the help of the finishing cycle G270.

1.2 G271 Stock removal in turning

1.2.1 Syntax

G271 U... R...

G271 P... Q... U... W...

The cycle for stock removal in turning is prepared by the optional Block G271 U... R...

The **U** value gives the depth of cut for stock removal. The direction of cut is designated by the sign of the W value in the activating block.

The **R** value gives the escaping amount. Both values have to be programmed without sign and values are taken as radius programmed.

Both values are modal and if one of them or the whole preparing block is omitted, the applicated values in the machine parameters TurningDepthOfCut and TurningEscapeAmount are taken for the turning cycle.



The cycle is activated by the Block G271 P... Q... U... W...

The **P** value gives the number of the first block for the finishing shape.

The **Q** value gives the number of the last block for the finishing shape.
The blocks in between are replaced by the multiple repetitive cycle.

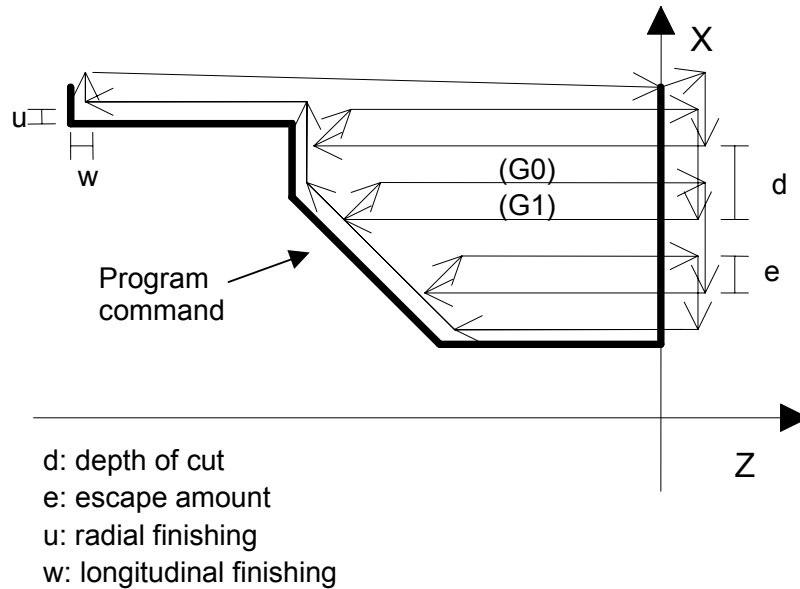
The **U** value gives the finishing allowance in radial direction (X). The sign of this value gives the direction of the allowance relative to the shape. The sign also designates the direction in which the levels of stock removal are changed. In the case of diameter programming the value is to be specified in diameter dimension.

The **W** value gives size and direction of finishing allowance in longitudinal direction (Z).

If a finishing allowance of zero is desired for U or W (or both), the sign has to be programmed together with the zero (for example: W+0 or W-0) in order to define the direction in which the levels of stock removal are changed. If a zero is programmed without sign, it is assumed as "positive".

1.2.2 Example

```
N50 G0 X45 Z0
N60 G271 U10 R5
N61 G271 P100 Q200 U.5 W1 S1200 F.8 M4
N100 G1 X10
N110 Z-30
N120 X30 Z-50
N130 X40
N140 Z-80
N200 X45 Z-80
```



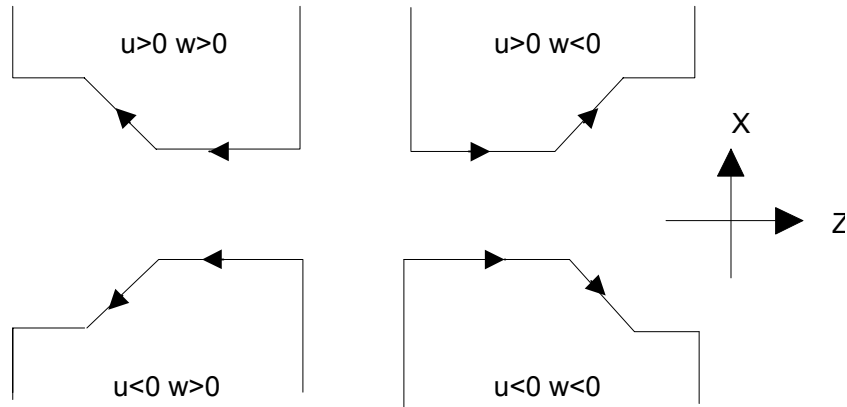
The turning cycle starts with the actual position before the block N100, i.e. with X45 Z0. The programmed allowances in positive X- and Z-direction, U.5 and W1, are added to the programmed positions.

For this roughly cutting shape three stock removal cycles on the X-levels 35.5, 25.5 and 15.5 are computed. First the tool moves to the start position X45.5 Z1.

The three cycles for stock removal are processed and then the shape for roughly cutting is processed with the positions X10.5 Z1; X10.5 Z-29; X30.5 Z-49; X40.5 Z-49; X40.5 Z-79; X45.5 Z-79.

The cycle ends at the starting position, i.e. the position before the block N100.

1.2.3 Direction of allowance



1.2.4 Effective G-codes

When G271 is active, only the programmed radial (X) and longitudinal (Z) position and the interpolation types G00, G01, G02, G03, G12 and G13 are taken into account.

All other programmed values, such as feed- or spindle speed values and all programmed G-codes are ineffective in stock removal.

Feed and spindle speed are constant with the before the first cycle block active rate.

If a finishing cycle is turned with the same blocks, no other G-codes than the ones for the interpolation type should be activated between the first cycle block and the finishing cycle G270. Else the shape could be destroyed.

In the blocks of the stock removal cycle, cycle programming *N... and subprograms are forbidden.

1.3 G272 Stock removal in facing

1.3.1 Syntax

G272 W... R...

G272 P... Q... U... W...

The cycle for stock removal in facing is prepared by the optional Block G272 W... R...

The **W** value gives the depth of cut for stock removal. The direction of cut is designated by the sign of the U value in the activating block.

The **R** value gives the escaping amount. Both values have to be programmed without sign and values are taken as radius programmed.

Both values are modal and if one of them or the whole preparing block is omitted, the applicated values in the machine parameters FacingDepthOfCut and FacingEscapeAmount are taken for the turning cycle.

The cycle is activated by the Block G272 P... Q... U... W...

The **P** value gives the number of the first block for the finishing shape.

The **Q** value gives the number of the last block for the finishing shape. The blocks in between are replaced by the multiple repetitive cycle.

The **U** value gives the finishing allowance in radial direction (X). The sign of this value gives the direction of the allowance relative to the shape. The sign also designates the direction in which the levels of stock removal are changed. In the case of diameter programming the value is to specified in diameter dimension.



The **W** value gives size and direction of finishing allowance in longitudinal direction (Z).

If a finishing allowance of zero is desired for U or W (or both), the sign has to be programmed together with the zero (for example: W+0 or W-0) in order to define the direction in which the levels of stock removal are changed. If a zero is programmed without sign, it is assumed as “positive”.

When G272 is active only the programmed radial (X) and longitudinal (Z) position and the interpolation types are taken into account. All other programmed values, such as feed- or spindle speed values are ineffective.

1.3.2 Example

...

```
N50 G0 X0 Z45
```

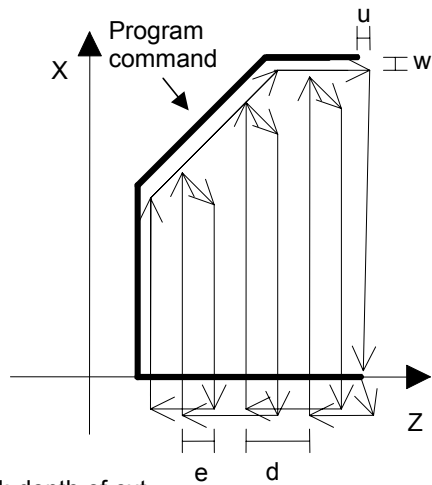
```
N60 G272 P100 Q200 U-.8 W1.3 S1100 F1 M3
```

```
N100 G1 Z10
```

```
N110 X30
```

```
N120 X50Z20
```

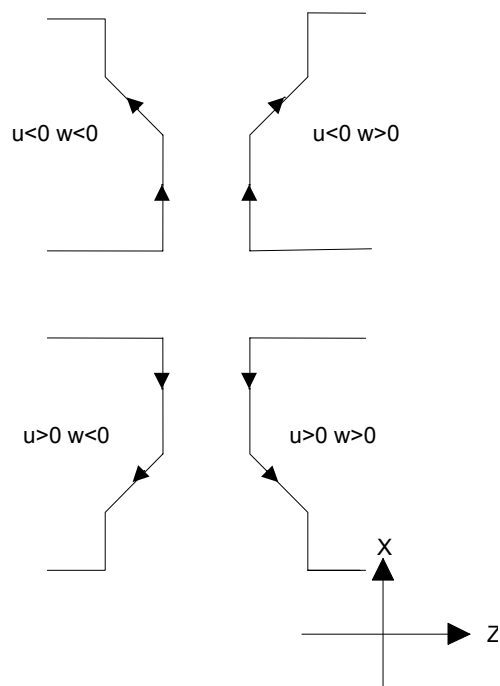
```
N200 X50 Z45
```

d: depth of cut
 e: escape amount
 u: radial finishing
 w: longitudinal finishing

The stock removal cycle starts at the actual position before block N100.
 Then three stock removal cycles are turned and afterwards the roughing shape is turned.
 The cycle ends at the starting position.

1.4 Direction of allowance



1.4.1 Effective G-codes

When G271 is active, only the programmed radial (X) and longitudinal (Z) position and the interpolation types G00, G01, G02, G03, G12 and G13 are taken into account.

All other programmed values, such as feed- or spindle speed values and all programmed G-codes are ineffective in stock removal.

Feed and spindle speed are constant with the before the first cycle block active rate.

If a finishing cycle is turned with the same blocks, no other G-codes than the ones for the interpolation type should be activated between the first cycle block and the finishing cycle G270. Else the shape could be destroyed.

In the blocks of the stock removal cycle, cycle programming *N... and subprograms are forbidden.

1.5 G270 Finishing Cycle

After roughly turning the programmed blocks can be used for a finishing cut.

1.5.1 Syntax

G270 P... Q...

The P value gives the first block for the finishing cut.

The Q value gives the last block for the finishing cut.

For an exact finishing cut after a stock removal cycle, the numbers of first and last block in G270 and G271/G272 must be identical. The start position of the finishing block must be identical to the position before the first block in stock removal. This can be achieved by programming G270 directly after the last block of stock removal.

All G-codes and other instructions in the finishing cycle blocks are effective.

1.5.2 Example

```
...  
N50 G0 X45 Z0  
N61 G271 P100 Q200 U.5 W1 S1200 F.8 M4  
N100 G1 X10  
N110 Z-30 F1  
N120 X30 Z-50 F1.5  
N130 X40  
N140 Z-80  
N200 X45 Z-80  
  
N210 G270 P100 Q200  
  
N220...  
...
```

After the stock removal cycle G271, the exact finishing shape is turned, i.e. the CNC moves from the start point (X45 Z0) to the positions in the blocks N100 to N200. Then it returns to the start point of the cycle (X45 Z0) and then continues with the next block (N220).

1.6 Error messages

- 315 Machine parameter TurningGCodeAppl faulty
The function Turning Cycles is not available in the system. Please contact your machine builder
- 708 Turning Cycles: Parameter wrong
If depth of Cut ≤ 0 or escape amount < 0 . If U- or W-value is not programmed in the activating block G271/G272.
- 709 Turning Cycles: Block number wrong
If P or Q in the activating block G271/G272 is not programmed.
- 710 Turning Cycles: Block not found
If there exists no block with the in P or Q programmed number.
- 711 Turning Cycles: Cycle programming *N not allowed
No Cycle programming *N... is allowed in the turning cycles.
- 712 TurningCycle: Circular level not allowed
The activated plane (G17, G18, G19) includes other axes than X and Z (the applied radial and longitudinal axes).

1.7 Part program display

The part program display is modified while turning cycles are active in order to clarify the progress of the turning cycles to the user. On principle the following is valid:

Line 1: shows the block which has caused the actual turning cycle.

Line 2: The second line shows the block which defines the contour element which is actually processed.



Line 3: This line shows the element or block which is to be processed after the actual contour element.

At entering turning cycles, the block with G271 (or G272) is displayed a second time. At this time the CNC moves to the start point with additional finishing allowance.

At the end of turning cycles (G271 or G272) the block with G271 (or G272) is displayed again. With this block active, the CNC moves back to the start point.

When finishing is active (G270) the block with G270 appears a second time at the end of the finishing cycle. With this block active the CNC moves to the point from which turning cycles were started.

Blocks which define contour-elements parallel to the tool moving axes are not displayed.



1.8 Machine parameter table

The tool management is optional and not available in all systems. It must be activated from your tool machine producer.

The required machine parameters are:

FacingDepthOfCut

Depth of cut for stock removal in facing

FacingEscapeAmount

Escape amount for stock removal in facing

TurningDepthOfCut

Depth of cut for stock removal in turning

TurningEscapeAmount

Escape amount for stock removal in turning

TurningGCodeAppl

G-Code of the first turning cycle (e.g. G270)

CharacterApplTab

The transverse axis (e.g. X axis) and the longitudinal axis (e.g. Z axis) are to define in CharacterApplTab

Additional the NC Address formats of "U" and "R" have to define like the axis NC Address format.