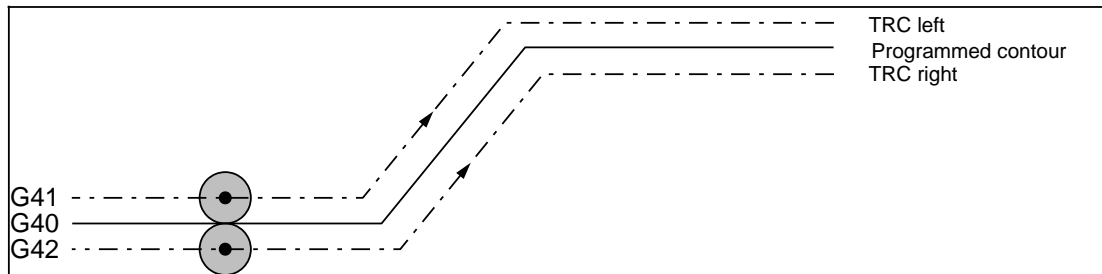


# 11 Tool Radius Compensation (G40/G41/G42)

The tool radius compensation (TRC) takes account of the tool radius for automatic compensation. The tool radius compensation calculates an equidistant path to the programmed contour. The compensation will take effect along the direction of movement to the left or to the right of the programmed contour, defined with G functions G41 or G42. The function is deselected with G40. These G functions are modal. G41/G42 refer to a right-handed rectangular coordinate system.

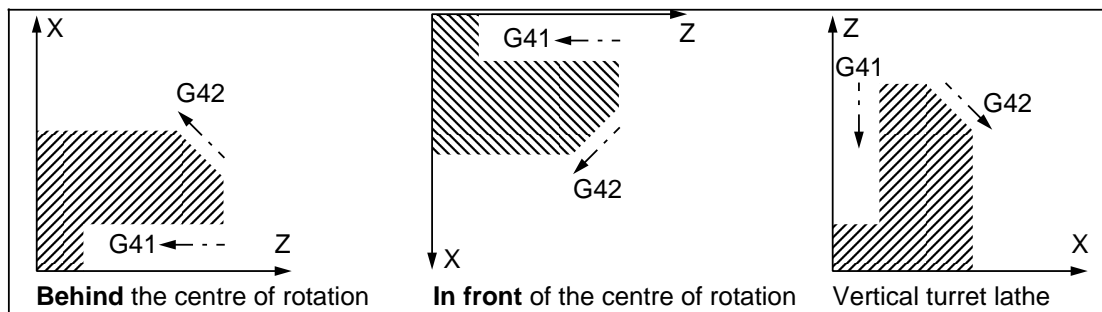


Tool radius compensation (G40, G41, G42)

## Use of tool radius compensation on turning machines

On turning machines, the direction of compensation appears to depend on whether machining is taking place in front of or behind the centre of rotation. When machining **BEHIND** the centre of rotation (clockwise coordinate system) the definition is as above.

When machining is **IN FRONT** of the centre of rotation (X axis mirrored around the Z axis), the direction of compensation is reversed for the operator. The part program and the tool offsets are the same whether the same workpiece is machined **IN FRONT** of or **BEHIND** the centre of rotation because there is rotation symmetry.



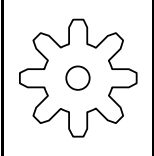
Using TRC (G41/G42) **behind/in front** of the centre of rotation

### Note:

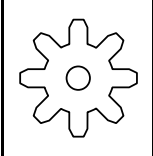
Please specify which tool is to be used in the machining program. The machining program should be tested with the smallest and the largest possible tool radius.

11.1 Behaviour at corners (G450/G451)

With the G commands G450 and G451, behaviour during transition from a contour element (straight line, circle) to a different contour element (behaviour at corners) at **external corners** when TRC (G41/G42) is selected can be controlled. These G functions are modal.

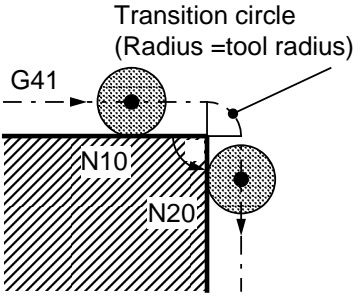
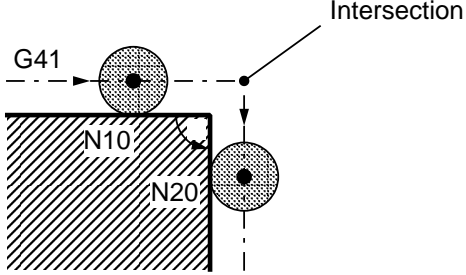


*The initial setting of G450/G451 can be set in the machine data.*



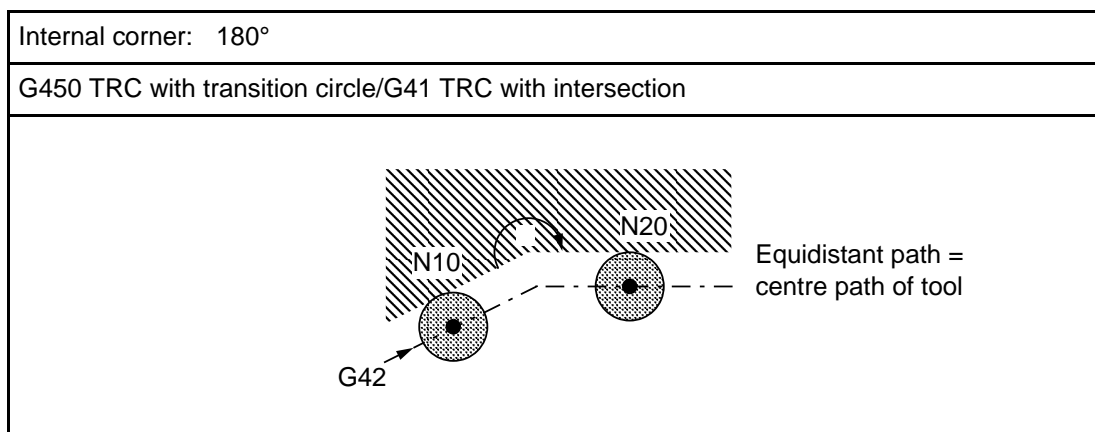
Comments on the following diagrams:

The thick line ( — ) represents the contour elements. The cutter centre line is shown by the dotted line ( - - - ).

External corner : 0 <180°	
G450: TRC with transition circle	G451: TRC with intersection
<div></div> <p>The centre point of the tool follows a circular path with the tool radius. This begins with the normal position (perpendicular to path tangent) at the end point of the previous path section and ends with the normal position at the starting point of the new path section.</p> <p>The transition circle is programmed before the next block (N20) and is declared as belonging to this block.</p>	<div></div> <p>Up to three intermediate blocks are inserted at the external corners (see following diagrams).</p> <p>The inserted intermediate blocks are also declared as belonging to the next block (here N20).</p>

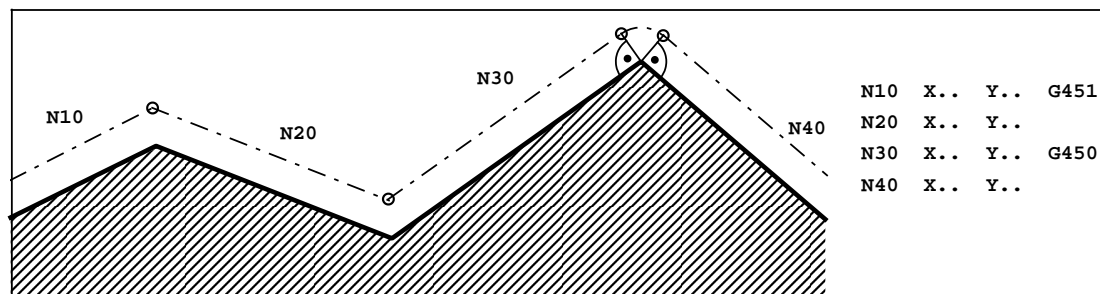
G450/G451 with TRC (G41) selected at an external corner

In the case of **internal corners**, the intersection of two equidistants of two successive path sections at a discontinuous transition are **always** calculated independent of G450/G451.



*G450/G451 with TRC (G42) selected at an **internal corner***

With the G commands G451/G450 it is always possible to program a change from "TRC with intersection to TRC with transition circle" or vice versa in the part program.



*Transition behaviour G451/G450*

### Comments on the following diagrams:

The contour elements are shown by the bold line ( — ). The tool centre line is shown by the dotted line ( - - - ). Programmed approach paths are shown by the thin line ( ——— ). All stopping points with individual blocks are marked with an S together with the associated block number in brackets. G42 is used in these examples.

11.2 Selection/deselection procedure (G455/G456)

When selecting/deselecting the function in the angular range "Behind the contour <180°" you can select whether the selection should be compensated for (G455) or directly executed in the "normal position" (G456). In the block following the selection block, a block starting vector (length R) is erected perpendicular to the programmed path. The G functions are non modal and must be programmed in a block with G40/G41/G42.

Selection/deselection with <180°	
G42/G455: "Compensated approach/departure"	G42/G456: "Approach/departure normal position"

G455/G456 with TRC selection/deselection

**Note:**  
If G455 is not programmed in the selection block, then selection with G455 is carried out automatically (G41↔G41 and G455). Approach with G456 makes it possible to approach the contour directly above the workpiece and then to lower the tool (e.g. with milling machines). It is also possible to directly approach a point outside the workpiece. Intermediate blocks are no longer necessary thus shortening program execution times.

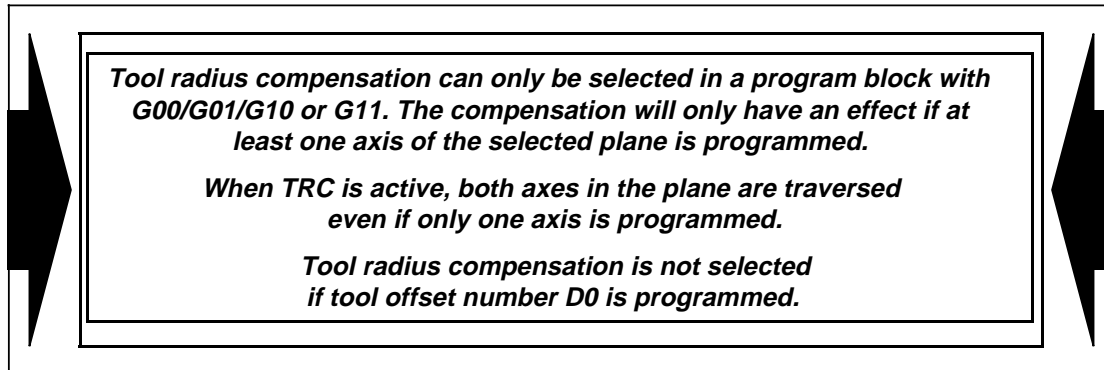
Selection with G42/G456 with <180°	
Approach with milling machines	Approach outside the workpiece

G456 with TRC selected

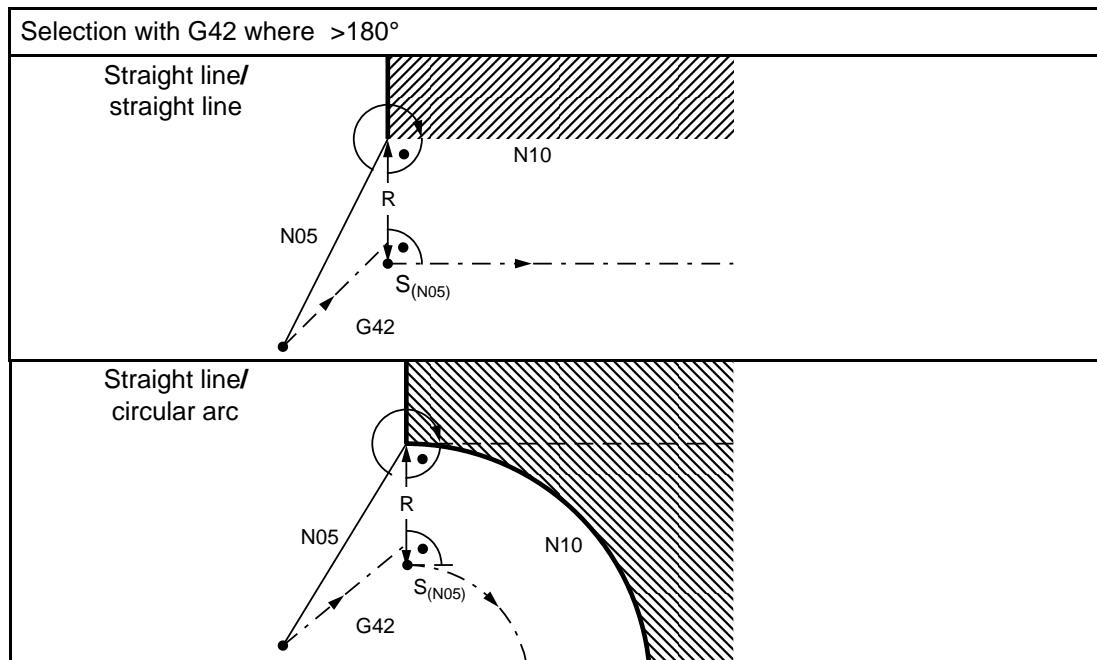
## 11.3 Selecting TRC (G41/G42)

Tool radius compensation is selected in the plane defined with G16 to G19 and with the preparatory functions G41/G42 together with the tool offset number D. It is not possible to change planes when tool radius compensation is active.

As soon as tool radius compensation is selected, two program blocks are read in to calculate the intersection.

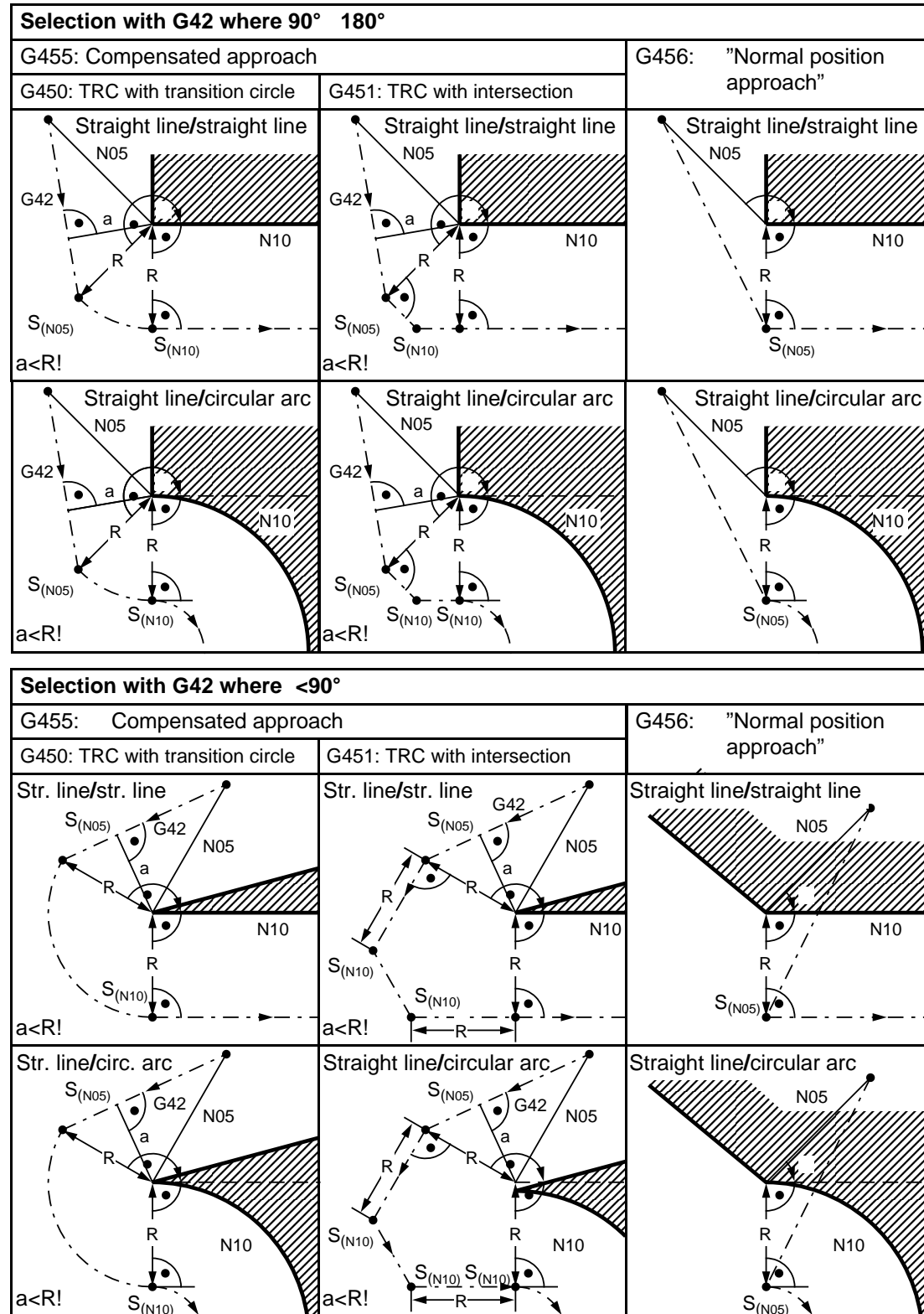


Tool radius compensation is usually selected in the angular range  $>180^\circ$  and is therefore independent of the set approach and corner behaviour (see Sections 11.1 and 11.2). The programmed point is approached directly in the "normal position" (perpendicular on approach point with distance tool radius R).



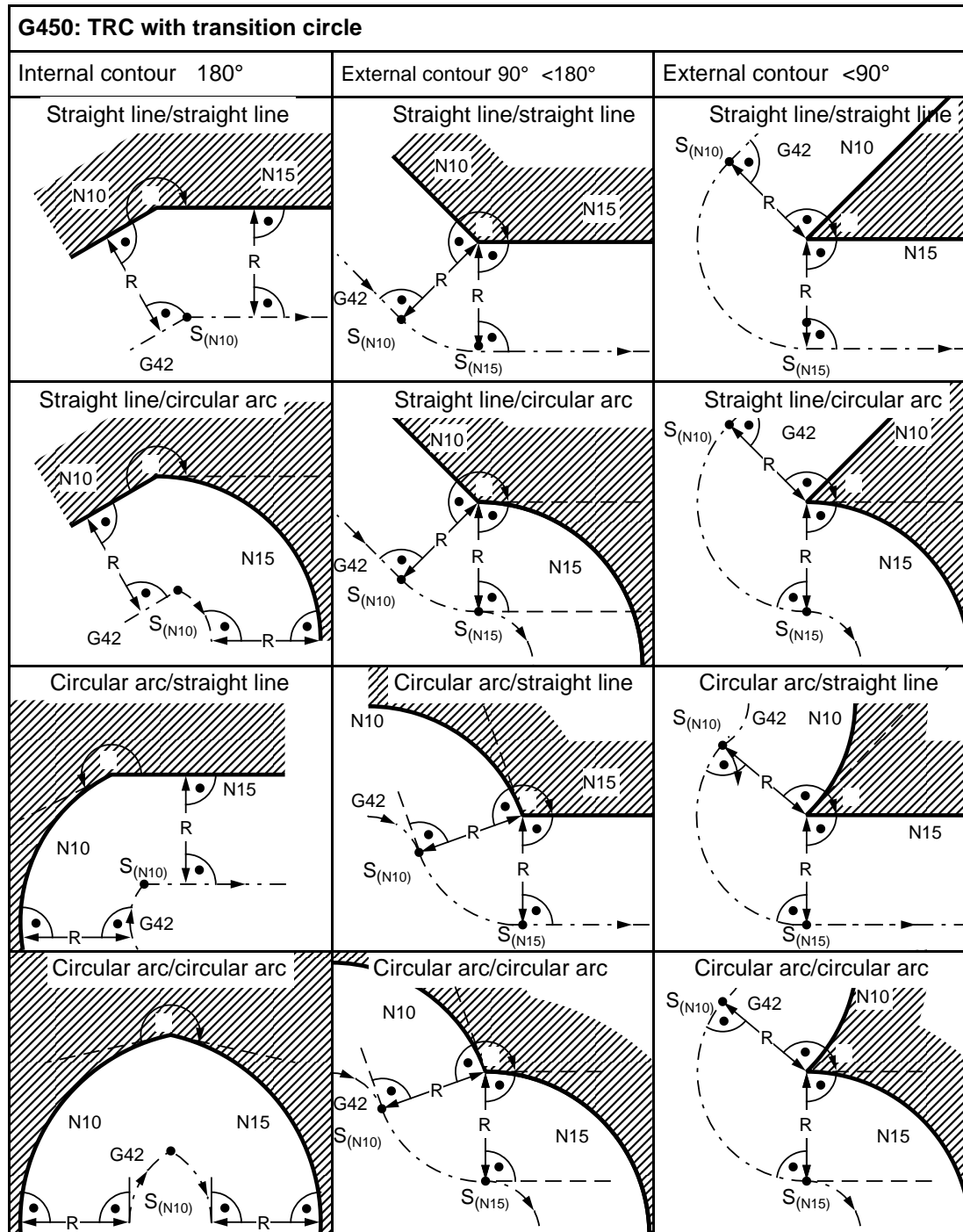
Selection of TRC with  $>180^\circ$

If the function is selected in the angular range  $180^\circ$ , the set selection procedure (G455/G456, see Section 11.2) is used. If compensation selection (G455) is programmed, the set behaviour at corners (G450/G451, see Section 11.1) is used. This results in a contour violation (see following examples,  $a < R$ !).



## 11.4 TRC in the program

When tool radius compensation is selected, the control reads in two further blocks in advance during processing of the current block and calculates the intersection of the compensated paths. During machining the tool path is independent of the TRC selection/deselection procedure (G455/G456), but dependent on the programmed behaviour at corners (G450/G451). The following diagrams show the behaviour of TRC with various transitions.



TRC with transition circle (G450)

G451: TRC with intersection		
Internal contour 180°	External contour 90° <180°	External contour <90°
<p>Straight line/straight line</p>	<p>Straight line/straight line</p>	<p>Straight line/straight line</p>
<p>Straight line/circular arc</p>	<p>Straight line/circular arc</p>	<p>Straight line/circular arc</p>
<p>Circular arc/straight line</p>	<p>Circular arc/straight line</p>	<p>Circular arc/straight line</p>
<p>Circular arc/circular arc</p>	<p>Circular arc/circular arc</p>	<p>Circular arc/circular arc</p>

TRC with intersection (G451)

#### Note:

If TRC is selected none of the following can be programmed: @714, G25, G26, G54 ... G57, G58, G59, G92, G74, G16 ... G19, C axis select./deselect. and transformation select./deselect.

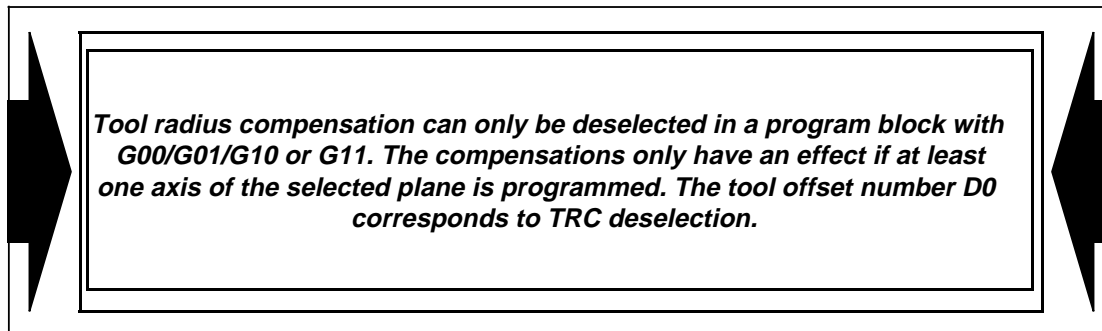
#### Remedy:

- Program these functions before selecting TRC.
- Deselect TRC, program functions, select TRC.

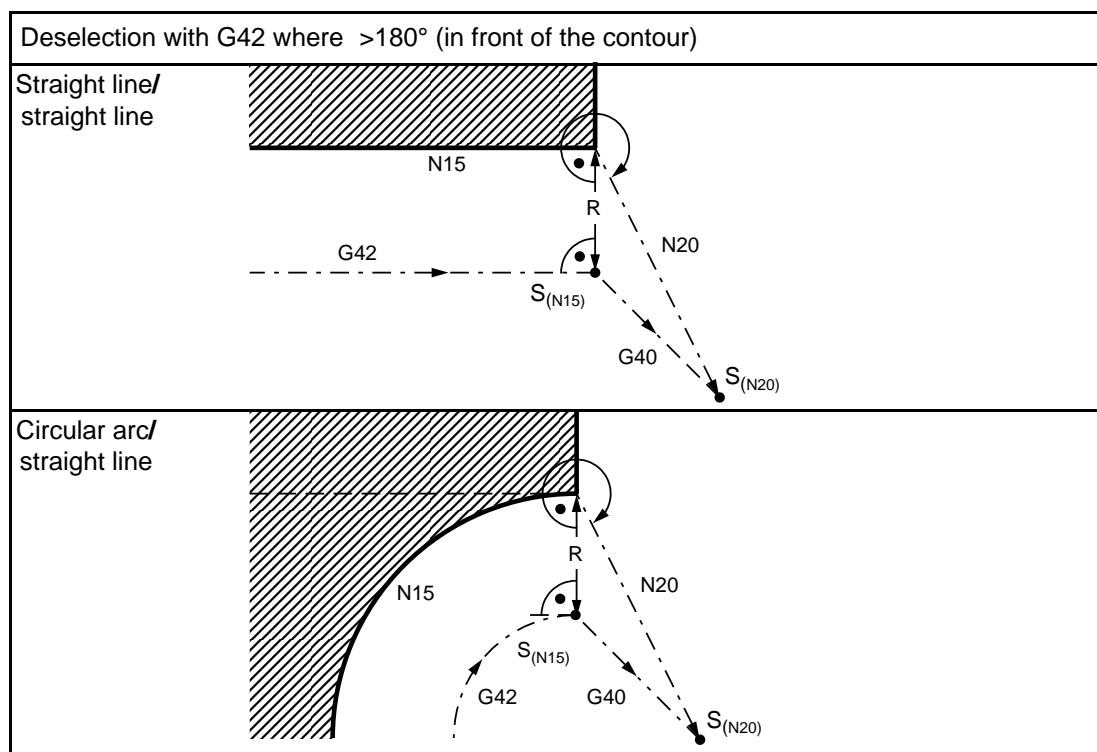


## 11.5 Deselecting TRC (G40)

Tool radius compensation (TRC) is deselected with the preparatory function G40.

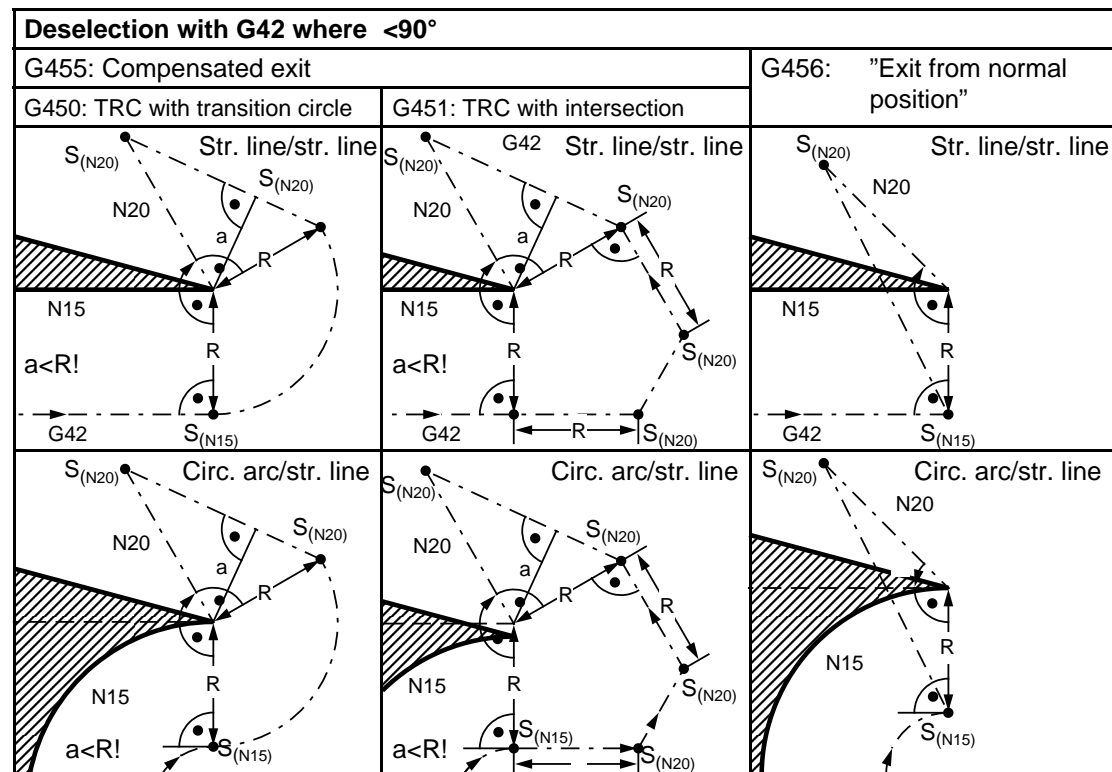
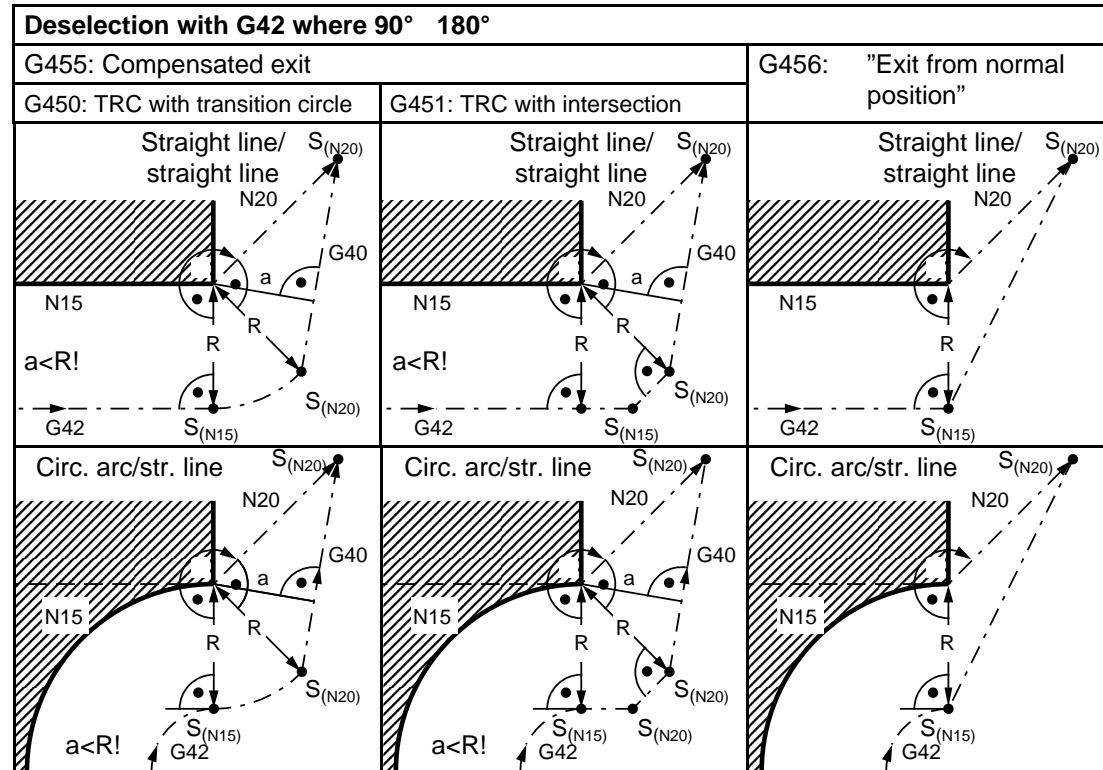


TRC is usually deselected in the angular range  $>180^\circ$  and is therefore independent of the set departure and corner procedure (see Sections 11.1 and 11.2). The programmed point is approached directly from the "normal position" (perpendicular on contour end point with distance tool radius R).



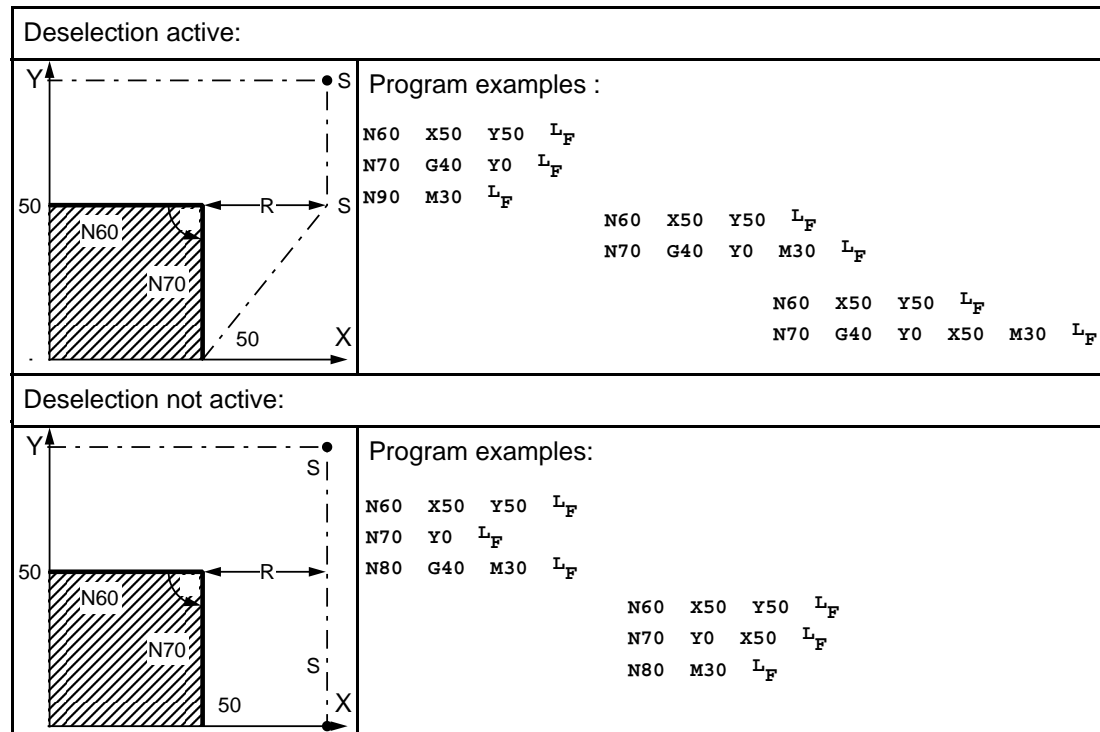
*Deselection of TRC where  $>180^\circ$*

If the function is deselected in the angular range  $180^\circ$ , the set selection procedure (G455/G456, see Section 11.2) becomes active. If compensated deselection (G455) is selected, the set behaviour at corners (G450/G451, see Section 11.1) is active. This results in a contour violation (see next example,  $a < R$ !).



### 11.5.1 Deselection of TRC in combination with program end (M30, M02)

Deselection of TRC in combination with program end will give varying results with different combinations (see following diagrams):



TRC in combination with M30, M02

The compensation is deselected with G40 in the last but one block together with at least one axis of the selected plane.

### 11.5.2 Special characteristics of selection and deselection

#### G40/G41/G42 without programmed path

G40, G41, G42 can be programmed in a block without programmed paths. However, they first become active when at least one axis of the selected plane has been programmed with a movement.

#### Example of selection:

```
N10  G01  G17  G41  D07  X..  Y..  L_F
```

At the end of this block the compensated path of the selected plane has been reached. Only the radius compensation value is calculated.

```
N15  Z..  L_F
```

Tool length compensation is calculated.

#### oder

```
N10  G17  L_F
```

Plane selection

```
N15  G41  D07  L_F
```

Compensation selection

```
N20  G01  X..  Y..  L_F
```

At the end of this block the compensated path in the selected plane has been reached. Only the radius compensation value is calculated.

```
N25  Z..  L_F
```

The tool length compensation is calculated.

### 11.6.1 Repetition of the active G function G41/G42

- the repetition is ignored if G41/G42 is programmed without G455/G456
- if G41/G42 is programmed with G455, TRC is deselected at the last point with G455 and TRC is selected with G455 at the programmed point
- if G41/G42 is programmed with G456, TRC is deselected at the last point with G456 and selected at the programmed point with G456

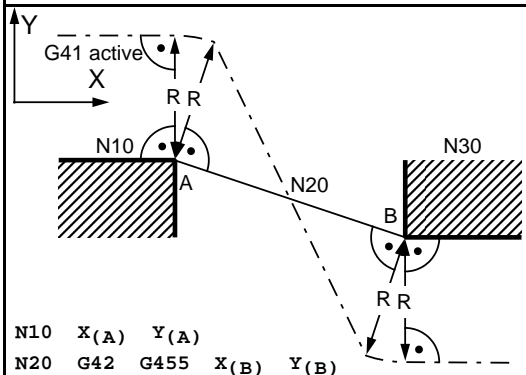
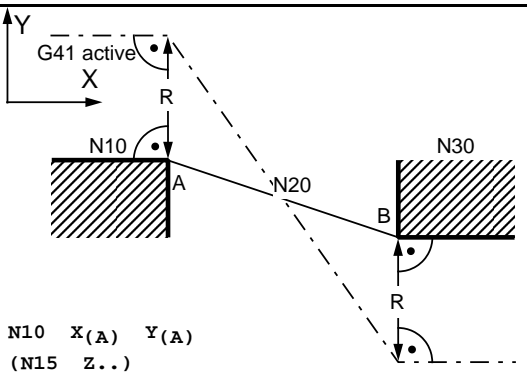
In this way machining at a contour can be terminated and machining of a new contour can be started without deselecting TRC. Selection/deselection procedure in various angular ranges is described in sections "Deselecting TRC" and "Selecting TRC". Behaviour at corners (G450/G451) is taken into account when selecting/deselecting.

Repetition of active G41/G42 command	
<p>G41/G455: Compensated traversing</p> <p>N10 X(A) Y(A)  N20 G41 G455 X(B) Y(B)  N30 X.. Y..</p>	<p>G41/G456: Traverse "normal position"</p> <p>N10 X(A) Y(A)  (N15 Z..)  N20 G41 G456 X(B) Y(B)  (N25 Z..)  N30 X.. Y..</p>

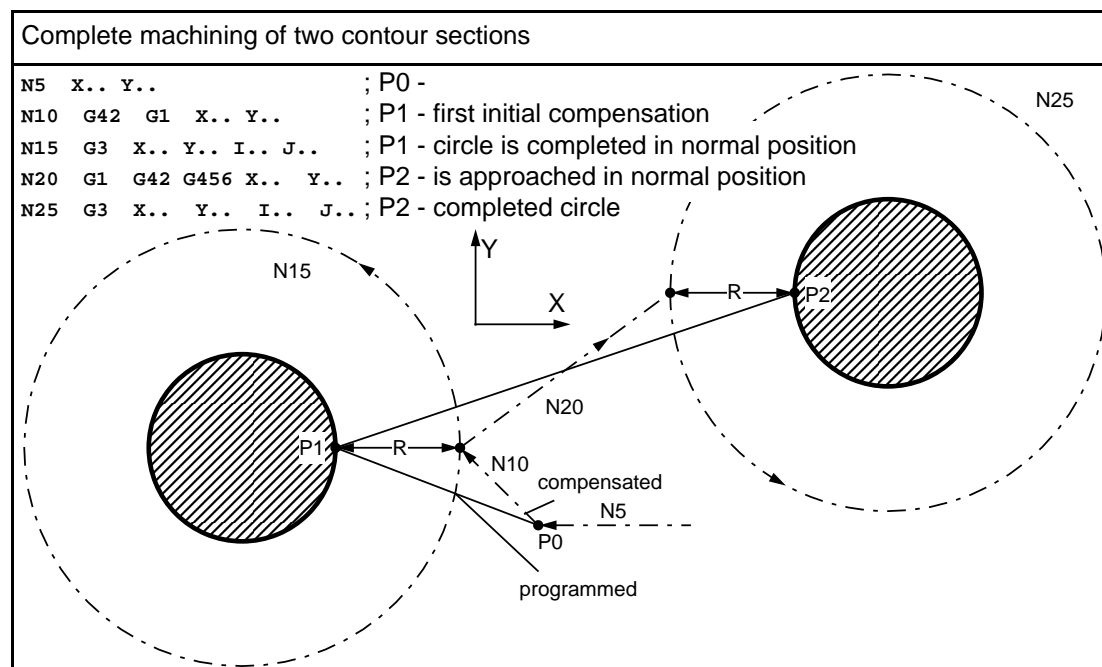
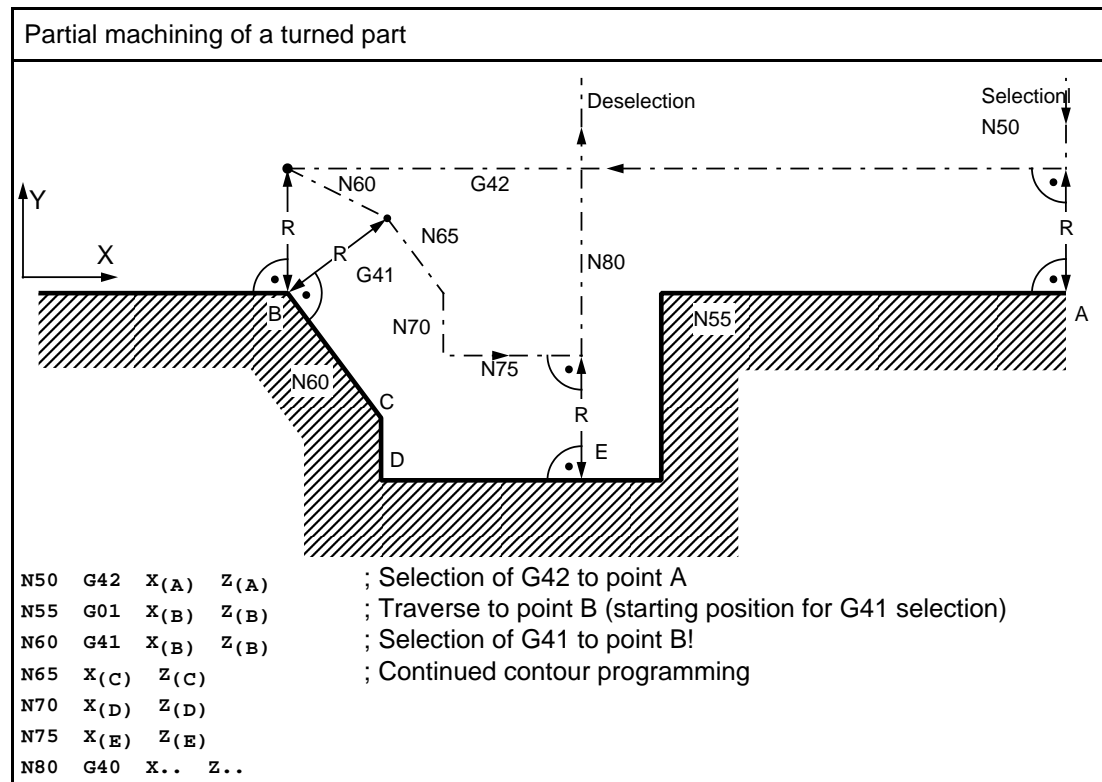
11.6.2 Changing the direction of compensation G41 G42

If G41/G42 is programmed when already active the following happens:

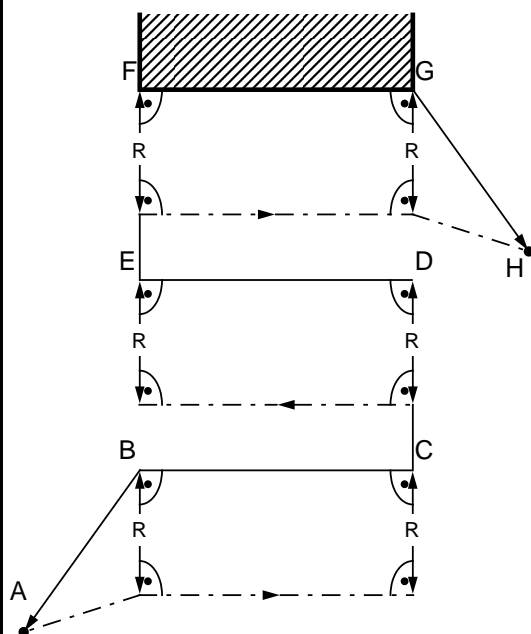
- If G42 is programmed without G455/G456, TRC is deselected at the last point with G456 and selected at the programmed point with G456
- if G42 is programmed with G455, TRC is deselected at the last point with G455 and selected with G455 at the programmed point
- if G42 is programmed with G456, TRC is deselected with G456 at the last point and selected with G456 at the programmed point. G456 need not be programmed.

Changing the direction of compensation	
G42/G455: Traversing with compensation	G42/G456: Traverse"normal position"
<div><p>N10 X(A) Y(A) N20 G42 G455 X(B) Y(B) N30 X.. Y..</p></div>	<div><p>N10 X(A) Y(A) (N15 Z..) N20 G41 G456 X(B) Y(B) (N25 Z..) N30 X.. Y..</p></div>

If the compensation is to be altered at a certain point this point must be programmed twice.



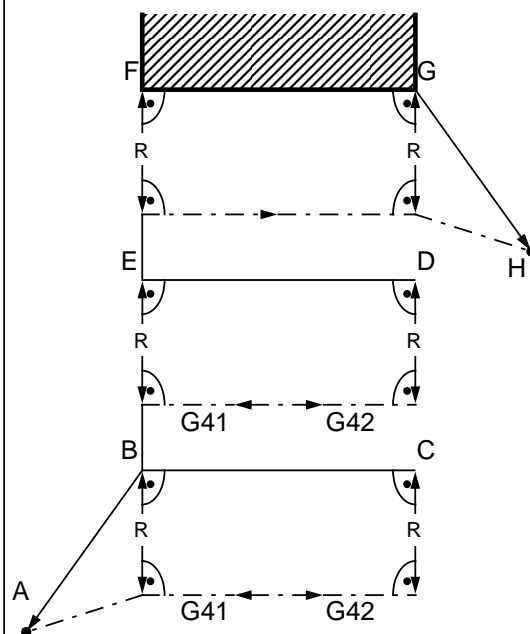
## Line by line milling with tool radius compensation



```

N10 X(A) Y(A) ;
N20 G42 X(B) Y(B) ; Compensation selection
N30 X(C) Y(C) ; Mill first line
N40 G41 X(D) Y(D) ; Infeed movement
N50 X(E) Y(E) ; Mill second line
N60 G42 X(F) Y(F) ; Infeed movement
N70 X(G) Y(G) ; Mill third line
N80 G40 X(H) Y(H) ; Deselect compensation

```

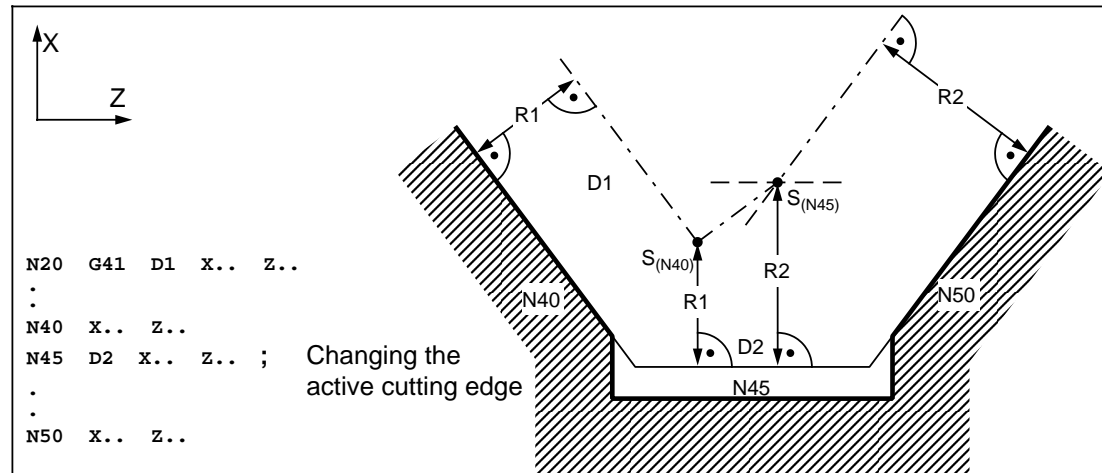


```

N10 X(A) Y(A) ;
N20 G42 X(B) Y(B) ; Compensation selection
N30 X(C) Y(C) ; Mill first line
N40 G41 X(B) Y(B) ; Exit
N50 G42 X(E) Y(E) ; Infeed movement
N60 X(D) Y(D) ; Mill second line
N70 G41 X(E) Y(E) ; Exit
N80 G42 X(F) Y(F) ; Infeed movement
N90 X(G) Y(G) ; Mill third line
N99 G40 X(H) Y(H) ; Deselect compensation

```

## 11.7 Changing the tool compensation number (D...)



Changing the compensation number

When the offset number is changed the following applies:

1. The block start intersection is calculated with the first tool radius.
2. The block end intersection is calculated with the second tool radius.

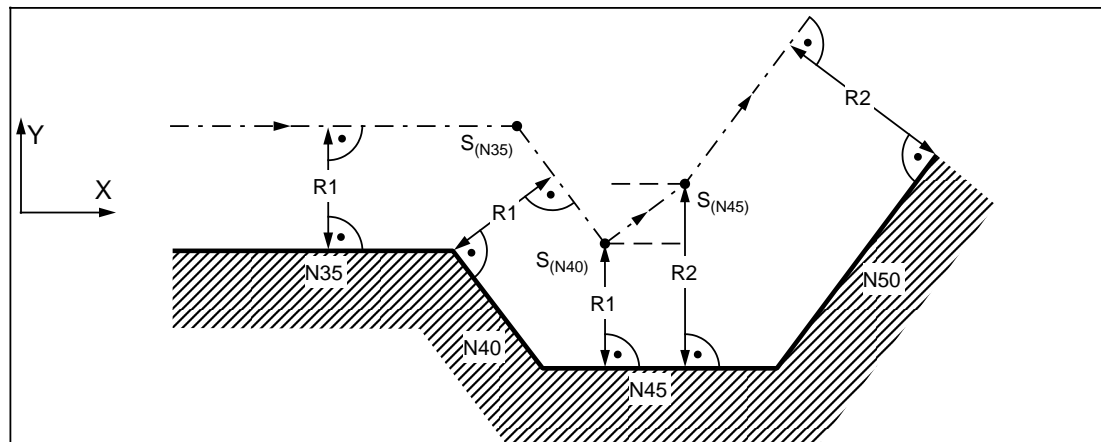
### Use:

- Tool with several cutting edges
- Cutting edge 1 (D1) is in cutting edge position 3
- Cutting edge 2 (D2) is in cutting edge position 4

**Note:** The offset number can only be altered in linear blocks.

## 11.8 Changing compensation values

The compensation values can be changed via the operator panel, the RS232 interface or via the PLC as an external tool offset. If the compensation values are changed in the program, they do not become active until two blocks after the compensation change has been programmed. If the changes are to become active immediately, the compensation values can only be entered in the NC stop state.



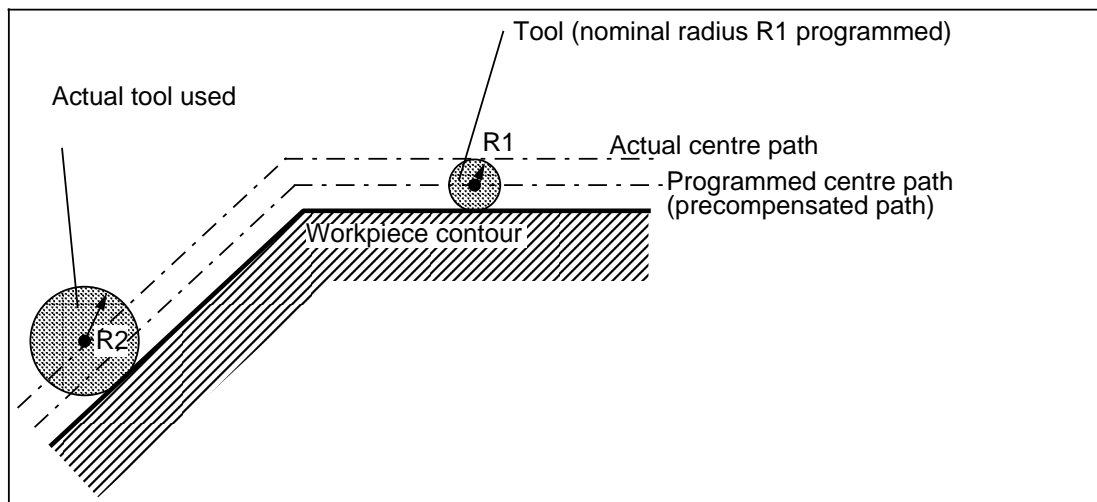


## 11.9 Procedure with precompensated contours

If the program has already been written with the nominal radius of the tool (centre point path), the tool radius compensation need only contain the difference between the nominal radius and the actual tool radius used.

Negative compensation values for the tool radius can also be used here.

### Example for milling:



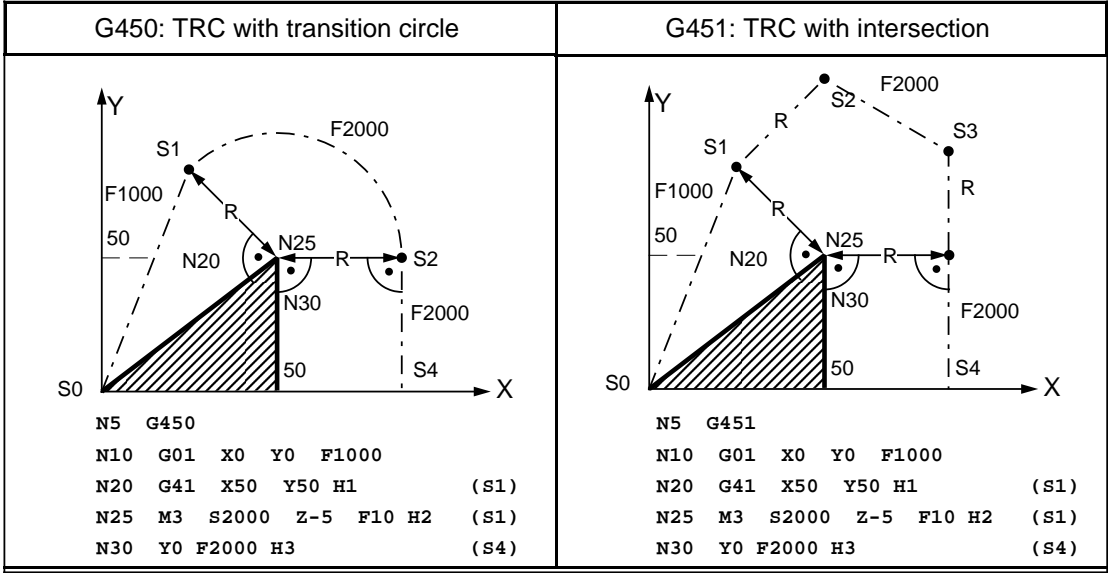
*Procedure with precompensated contours*

Radius value to be entered in the offset memory:

$R2 - R1$  ( Input positive, as  $R2 > R1$ )

The entered radius value is negative when  $R2 < R1$ .

11.10 Effect of G functions, M/auxiliary functions and feedrates



Effect of auxiliary functions and feedrates

The tool radius compensation generates one or several intermediate blocks for machining external corners. An additional block can be programmed outside the TRC plane (N25) between the blocks programmed in the TRC plane (N20 and N30). The auxiliary functions have the following effect:


- The auxiliary functions in the first programmed block of the TRC plane (H1 in N20) are output at point S0.
- The auxiliary functions or infeed movements in the programmed block outside the TRC plane (H2, M3, S2000, Z-5 in block N25) are output at point S1.
- No auxiliary functions are output in the intermediate blocks generated by the tool radius compensation.

They are traversed with the feedrate and the G functions of the next block (F2000 in N30).


If no feedrate is programmed in the next block, the last feedrate to be programmed is used. This can also be the feedrate programmed in a block outside the TRC plane (F10 in N25).

Any non-modal G functions in the next block (G09) are active in every intermediate block.

- The auxiliary functions in the second programmed block of the TRC plane (H3 in N30) are output after the generated intermediated blocks (S2 for G450, S3 for G451).



*You can define in the MD whether the auxiliary functions are to be output before or during the axis movement. Independently of this, there are M functions (M00, M01, M02, M18, M30) which do not become active until block end.*



## 11.11 TRC with blocks outside the compensation plane

When TRC is active, up to 3 blocks (N30, N35, N40) can be "programmed" outside the compensation plane and up to 2 blocks in the compensation plane (N25, N35).

If more than 3 blocks are programmed outside the compensation plane, alarm 3021, "Contour violation with TRC" is triggered.

**Note:** With SW 1 and 2 only one block can be "programmed" outside the compensation plane. If more than 1 block is programmed outside the compensation plane, alarm 3021, "Contour violation with TRC" is triggered.

One of the following can be a block outside the compensation plane:

- An "auxiliary function block" e.g. M08, H1
- A block with "path = 0"; a block in the compensation plane (X0) is treated like a block outside the compensation plane (Z0).
- Block not in the compensation plane e.g. Z-5
- A calculation block with comments

### Example:

N25 G91 X200 L<sub>F</sub>

(Block in compensation plane)

N30 M08 H1 L<sub>F</sub>

(Block outside the compensation plane)

N35 X0 L<sub>F</sub>

(Block in compensation plane with path = 0 is treated like a block outside the compensation plane)

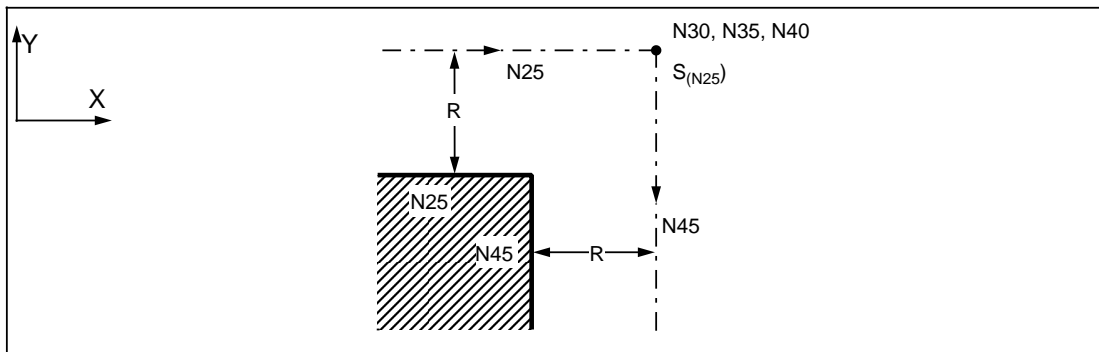
N40 Z-5 L<sub>F</sub>

(Block outside the compensation plane)

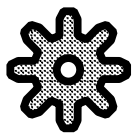
N45 Y-100 L<sub>F</sub>

(Block in compensation plane)

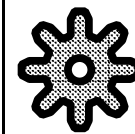
N30, N35, N40 are executed at point S<sub>(N25)</sub>.



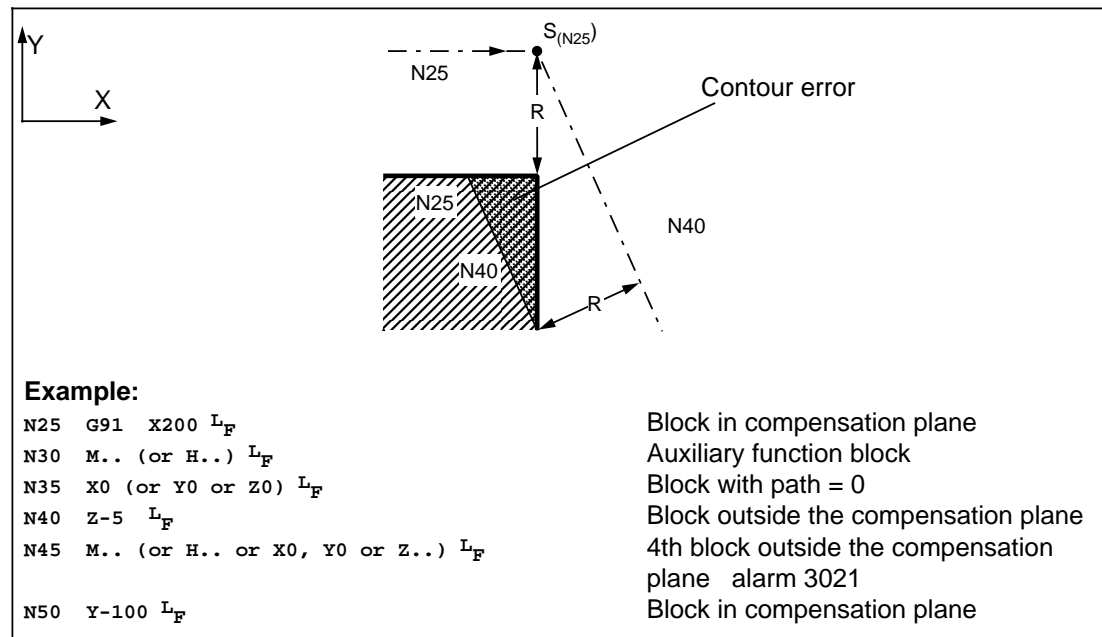
TRC with block outside compensation plane



**The effect of this alarm can be influenced in the machine data, contour violations may result if traversing is continued after the alarm.**



## 11.11 TRC with blocks outside the compensation plane

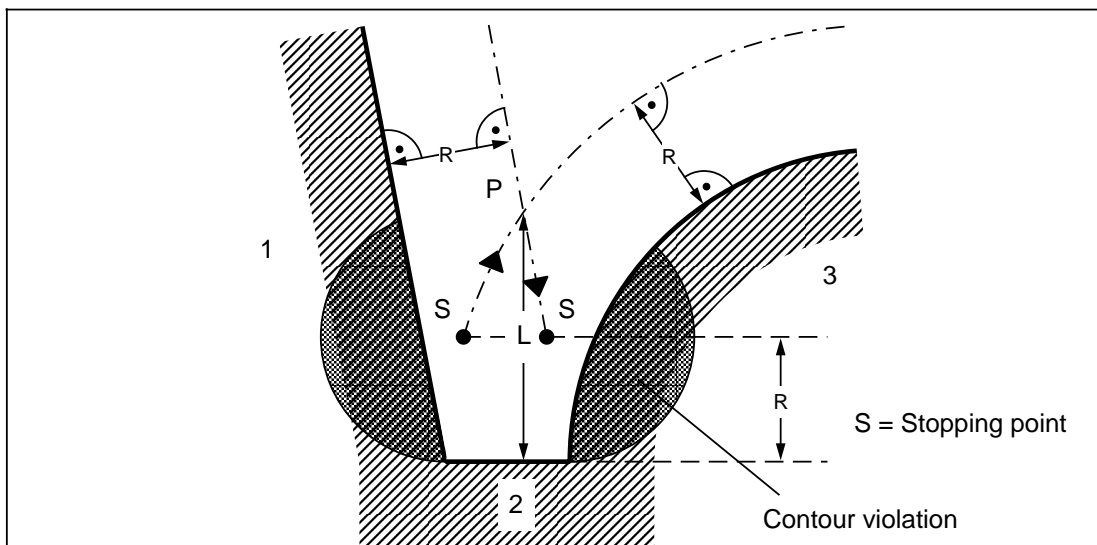
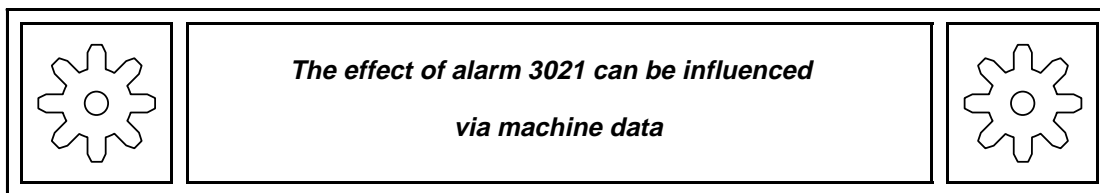


TRC with more than three blocks outside the compensation plane

## 11.12 Contour violation with tool radius compensation

Contour violations can occur with three successive contour elements and a tool radius.

The tool centre line is an equidistant with the distance tool radius ( $R$ ) to the programmed path. The equidistants from the first and third contour elements can form an intersection ( $P$ ). If the length of the perpendicular ( $L$ ) from the second contour element to intersection  $P$  is longer than the tool radius, contour violation will occur. The control outputs alarm 3021 "Contour violation with TRC" during processing.



*TRC special case: Intermediate block < compensation value*

## 11.13 Contour violation without TRC on turning machines

Turning tools (type P1=1 ... P1=8) are usually measured with reference to the theoretical tool tip P (see Section 10.4).

If TRC is not used, dimensional errors can occur because of the tool nose radius. Such errors can be avoided by selecting TRC.

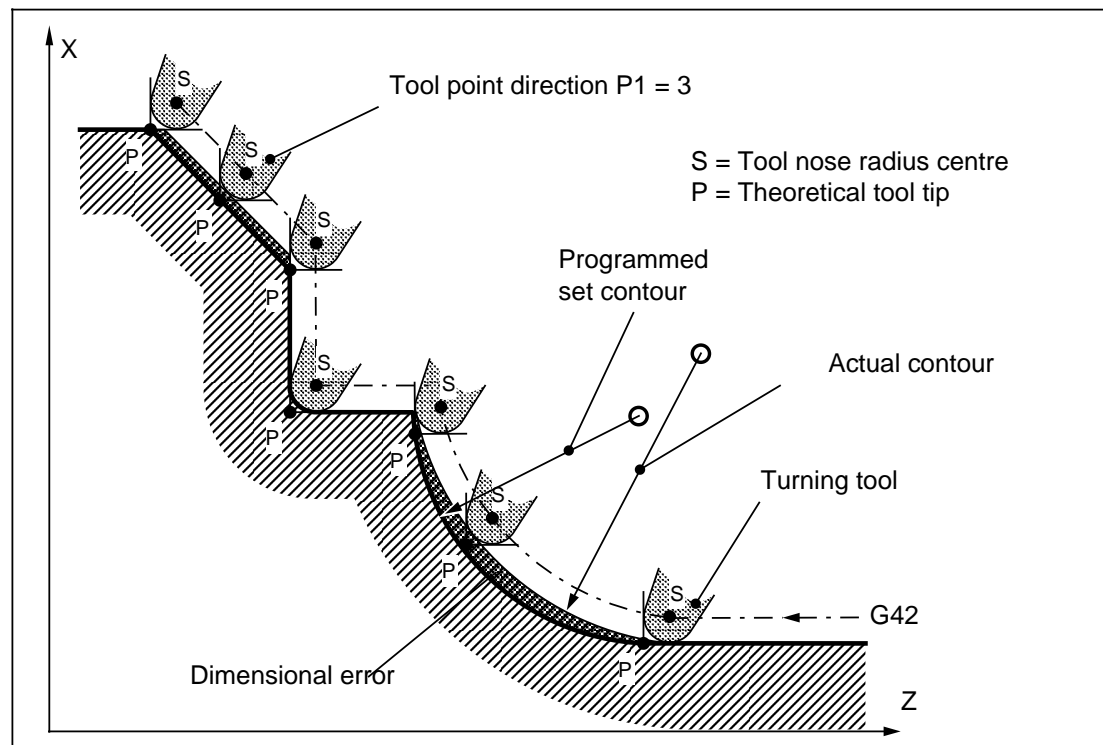


Fig.: Dimensional error resulting from the tool nose radius of the cutting tool **without** TRC

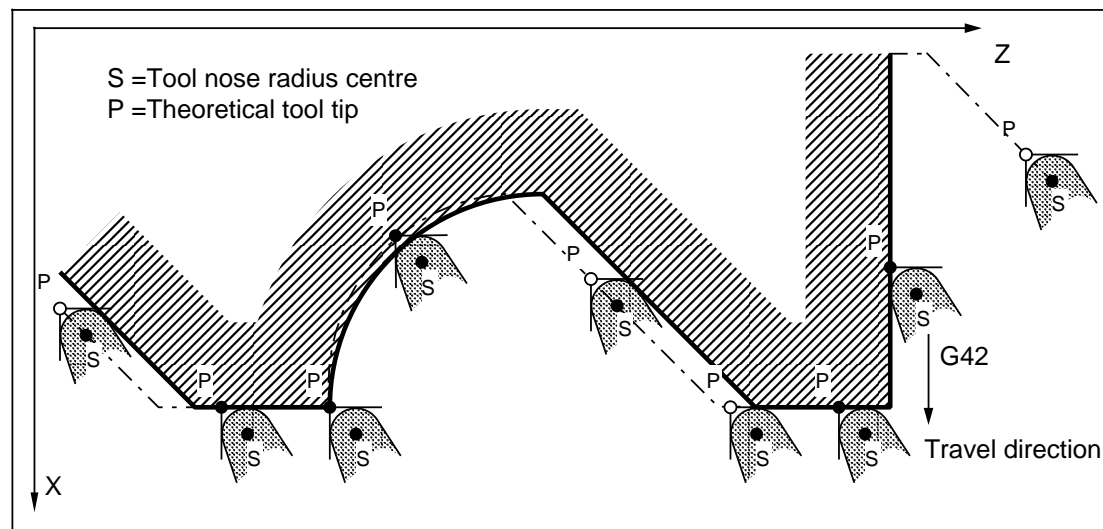
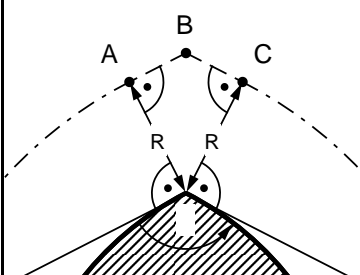
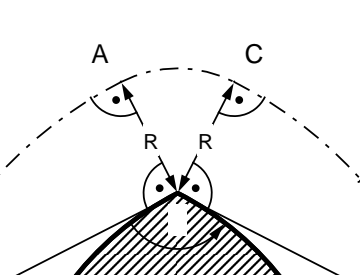
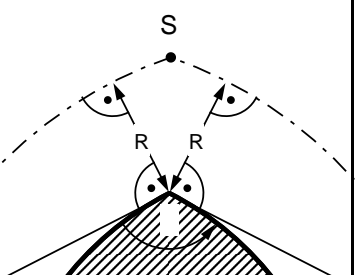
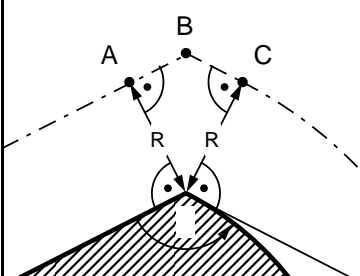
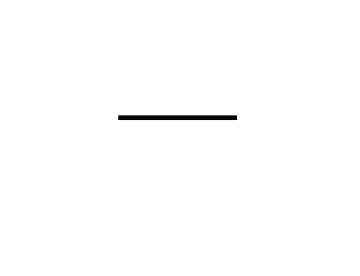
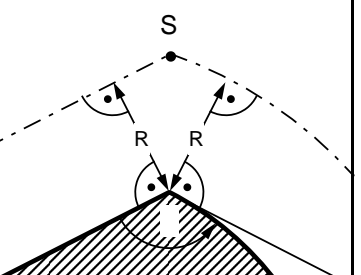


Fig.: **No** dimensional error resulting from tool nose radius of turning tool **with** TRC.

## 11.14 Circle transitions with obtuse angles

When programming exterior contours with circle transitions and obtuse angles ( $90^\circ < \alpha < 180^\circ$ ), one or two intermediate blocks are inserted at the circle transitions (straight line/circular arc, circular arc/straight line and circular arc/circular arc) (see Section 11.4, "TRC with transition circle" and "TRC with intersection"). If the angle is almost  $180^\circ$ , the traversed paths in the intermediate blocks become very small and can stop the continuous path mode. To avoid this, the control can suppress the intermediate blocks. It either switches from "TRC with intersection" to "TRC with transition circle" which avoids an intermediate block, or it approaches the intersection of both blocks. No intermediate block is generated if this is done.

Suppression of intermediate blocks at circle transitions with obtuse angles		
Normal case	Switchover to transition circle	Intersection approach
Circular arc/circular arc 	Circular arc/circular arc 	Circular arc/circular arc 
Straight line/circular arc 		Straight line/circular arc 

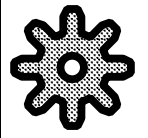
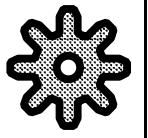
The distance from A to C is compared with the distance defined on start-up to determine whether switchover will take place. If the distance A to C is smaller than the defined distance, an intersection is traversed (applies to all circle transitions with "TRC with transition circle" G450 and "TRC with intersection" G451). If the distance A to C is less than double the defined distance, the control switches to "TRC with transition circle" G450 with "TRC with intersection" G451 and transition circular arc/circular arc.

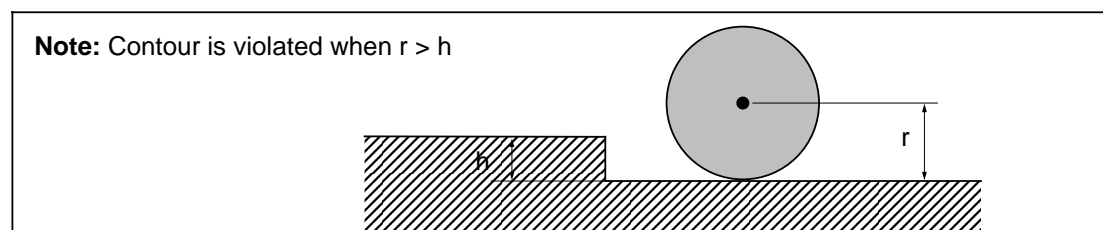
### Caution:

When an intersection is approached, the programmed circular arc(s) is/are lengthened. If the circle is lengthened to exceed  $360^\circ$ , only the section above  $360^\circ$  is traversed.

### Remedy:

Divide such a circular arc into several sections.

	<p><b>The distance for switchover</b></p> <p><b>is defined in the MD.</b></p>	
---	---	---

[illegible]

© Siemens AG 1992 All Rights Reserved 6FC5198- AA10  
SINUMERIK 840C (PG)