3. Programming
3.1 Basics

What is programming?
As opposed to a conventional lathe, series machines are not controlled by hand, but by a CNC control system, the **Gildemeister ELTROPILOT L-2.** CNC means Computerized Numerical Control. In order to be able to instruct the machine as to which machine functions are to be performed, a machining cycle must first be divided into individual steps.

For example:
Spindle and feed on,
coolant on,
rapid traverse to contour,
engage first contour point,
engage second contour point,
etc.
These individual steps must then be translated into a language comprehensible to the control system. This procedure is called programming.

NC words
A so-called NC word must be programmed for each function which the machine is to perform. An NC word consists of a letter, the **address**, and following digits, e.g.:

F 0.5

F is the address for feed.
G95 F0.5 means feed 0.5 mm/rev.

Example: 3-word part of block:

...G1 X35.7 Z-29.25...

G1 means feed along a straight line.

X and Z determine the final position of the traverse path.

Block
A block consists of several words. For identification purposes, a block always begins with a block number. It is programmed under address N and consists of max. four digits (N9999).
The block number has no effect on the processing sequence - the blocks are processed in the order in which they stand in the program. For example:

...
...
N 110 G...
N 130 G...
N 120 G...
...
...

Block N 130 is processed **before** block N 120.

Example of a block

N10	G1	X50	M7
Block no.	Straight line	Target pos.	Coolant

This NC block contains:

the **block number**, address N,

the **path conditions**, how the tool is to reach target position (circle, straight line), address **G**,

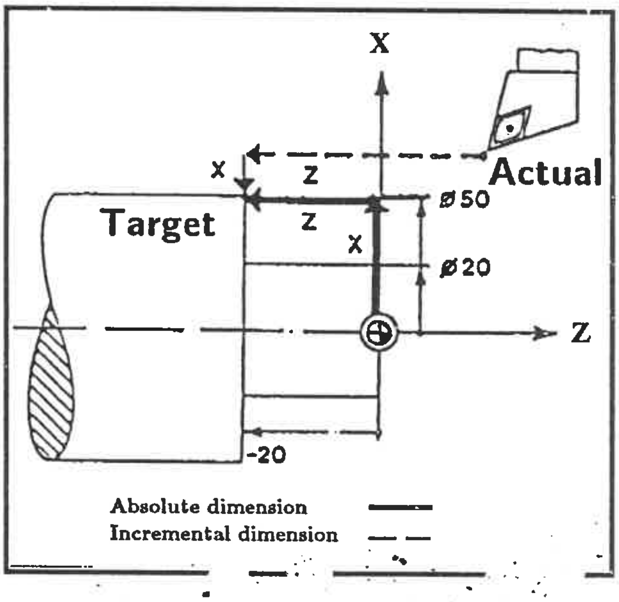
The **path instruction** (target position), to which position the tool is to move, address **X** and **Z**,

The **switch function** (coolant etc.) address M.

Program number (%)
Control system memory has capacity for many programs. A program number is entered in order to differentiate the programs; it may consist of as many as 8 digits (e.g. 99999999). The program number is requested by the control system when the program is entered (see section 4).

G and M functions
G and M functions are important components of the programming language. Their meanings are partially standardized.
G functions serve primarily for programming the geometry of the workpiece. M functions are frequently required for machine functions, e.g. for switching procedures (coolant on/off, spindle gear stage etc.).
The G functions are described in detail, accompanied by examples, in section 3.2.
The M functions are described in detail in section 3.6.

Path information (X, Z)
Target positions, i.e. data concerning the position to which the tool is to move, are programmed under address X for cross and Z for longitudinal movements (see also section 2, geometrical principles).



Address X:
- In absolute dimension programming, the number following X always indicates diameter (target point).
- In incremental dimension programming, the number following X indicates radius difference (path length), see example after G1.

Address Z:
- In absolute dimension programming, the number following Z indicates the distance from the workpiece zero point.
- In incremental dimension programming, the number following Z indicates the distance from the previous point.

When starting the program automatically absolute dim. (G90) and feed mm/rev (G95) are active.

The following pages contain a list of the various machining steps together with the corresponding G-functions and the most important M-functions.

Tool movements without machining operations being performed

G0	Positioning of the tool tip at a defined point (no machining of the workpiece)
G14	Engaging tool change point

Straight lines

G1	Moving the tool along a straight line (maching cylindrical or conical elements)
G88	Chamfers of 45°

Circles

G2	Circle in clockwise direction with incremental indication of the center coordinates
G12	Circle in clockwise direction with indication of the center in absolute dimension
G3	Circle in counter-clockwise direction with incremental indication of the center coordinates.
G13	Circle in counter-clockwise direction with indication of center in absolute dimension.
G1	Roundance at the end of a straight line.
G87	Transition radius of 90°

Threads (preprogrammed maching cycles)

G31	Longitudinal thread
G32	Transversal thread
G33	Special thread
G35	Metric ISO-thread

Special machining operations

G74	Cycle of drilling deep
G77	PCD on frontface of the workpiece
G770	Angular circle cycle
G78	PCD on circumference of the workpiece
G79	Milling keyways

Zero point shifts

G53	Allowance 1
G54	Allowance 2
G55	Allowance 3
G56	Allowance 4
G59	Programmable zero point shift
G152	Zero point shift for C-axis 1

Allowances

G51	Programmable stock allowance
G57	Stock allowance for contour cycles
G58	General stock allowance

Groove

G86	Cycle grooving
G861	Cycle transversal groove (with programmable contour)
G862	Cycle longitudinal groove (with programmable contour)
G863	Transversal keyway finishing (together with G861)
G864	Longitudinal keyway finishing (together with G862)
G85	Undercut E/F

Feed

G94	Feedrate in millimeter (inch) per minute
G95	Feedrate dependent on rotation (main spindle)
G193	Feed as a function of tool geometry (auxiliary spindle 1)
G195	Feed as a function of revolutions (auxiliary spindle 1)

Speed

G96	V-constant (main spindle)
G97	Speed of main spindle
G196	Constant cutting speed (auxiliary spindle 1)
G197	Speed of auxiliary spindle 1
G296	V-constant (auxiliary spindle 2)
G297	Speed of auxiliary spindle 2
G26	Speed limitation of main spindle
G126	Speed limitation of auxiliary spindle 1
G226	Speed limitation of auxiliary spindle 2

Cycles

G81	Cycle longitudinal
***G817**	Cycle longitudinal roughing
***G818**	Cycle contour longitudinal without plunging into "valleys"
***G819**	Cycle falling contour with plunging into "valleys"
G82	Cycle transversal
***G827**	Cycle transverse roughing
***G828**	Cycle contour transversal without plunging into "valleys"
***G829**	Cycle falling contour with plunging into "valleys"
***G83**	Cycle contour

*G836 Cycle contour parallel

G80 Cycle end

*) Cycles with freely programmable contours

Tool nose radius compensation (German abbrev. SRK) /
Milling tool radius compensation (German abbrev. FRK)

G41	SRK/FRK left-hand side. In direction of traverse movement tool is on the left side of contour.
G42	SRK/FRK right-hand side. In direction of traverse movement tool is on the right side of contour.
G40	Cancel SRK/FRK

Other functions

G04	Period of dwell
G09	Precision stop
G60	Function protection zones
G61	Jump function
G64	Intermittent feed
G90	Calculation of traverse path in absolute dimension.
G91	Incremental calculation of traverse path.
G92	Tool file
G900	Calculation of return point for inspection cycle
G901	Transfer of actual values into the variables memory
G902	Transfer of complete current zero point shift to variables memory
G907	Blockwise deactivation of speed monitoring
G908	Setting feedrate values as 100 %
G909	Interpreter stop
G921	Convert system of dimensioning to slide position
G940	Switch off block display

G941 Switch on block display

G943 Switch on tool service life surveillance

G944 Switch off tool service life surveillance

G981 Set system of dimensioning back to tool-specific shiftings

Graphics representation

G38 Erasure of graphics

G970 Sector boundaries

G971 Dimensions of blank

G972 Clamping area length

G973 Scope of graphic representation

Working on the frontface (C-axis)

G100 Positioning the driven tool in rapid traverse to a defined position in front of the frontface of the workpiece (no machining of the piece)

G101 Moving the driven tool along a straight line on the frontface of the workpiece

G102 Circle in clockwise direction on the frontface of the workpiece

G103 Circle in counter-clockwise direction on the frontface of the workpiece

Working on the circumference

G110 Positioning the driven tool in a defined position above the circumference of the workpiece (no machining of the part)

G111 Moving the driven tool along a straight line on the circumference of the workpiece

G112 Circle in clockwise direction on the circumference of the workpiece

G113 Circle in counter-clockwise direction on the circumference of the workpiece

Functions for controlling the measuring process

G910	Activating the input of the probe for measurement of the workpiece (collision surveillance)
G911	Activating the input of the probe for measurement of the tool (collision surveillance)
G912	Switchover of the input of the probe, activated by G910 or G911, to detection of the actual value
G913	Terminating the measuring process, with error evaluation
G920	De-activating the active zero point shift for tool measurement
G980	Re-activating the zero point shift de-activated by G920

SPS functions

G600 to 699 The meaning of these functions can be defined by appropriately programming the machine interface (SPS).

Summary of the most important M-functions

M0	Absolute stop
M1	Optional stop
M3	Spindle movement in clockwise direction (as seen from operator's positions)
M4	Spindle in counter-clockwise direction (as seen from operator's position)
M5	Spindle off
M7	Coolant 1 on
M8	Coolant 2 on
M9	Coolant 1+2 off
M19	Spindle stop in defined end position
M30	End of program with return to program start
M99	End of program with return to start of program and automatic restart

3. Programming
3.2 Geometry Functions

Detailed Description of Geometry Functions

Rapid traverse movement G0

Rapid traverse, G0
If the slides are to move in rapid traverse to the target position, G0 must be programmed.

Required addresses
After selecting path condition G0, the control system requests the following inputs:

DIAMETER X:
Target position in X (diameter dimension)

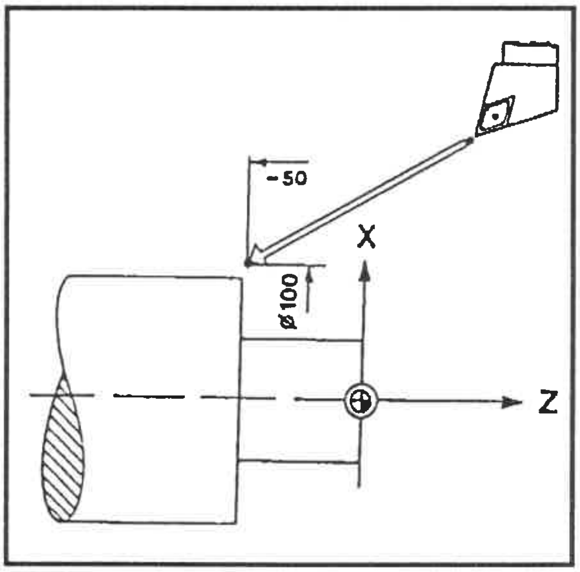
LENGTH Z:
Target position in Z

Rapid traverse movements are also possible with stationary spindle. After the rapid traverse movement, the machine returns to the previously programmed feed rate.

Example

N 10 G0 X100 Z-50

The tool moves in rapid traverse on both axes simultaneously to point X100 Z-50.



Note
The rapid traverse speeds are stored in the parameter store and can be altered under PARAMETERS operating mode.
In the modes AUTOMATIC, SINGLE BLOCK and MANUAL CONTROL, rapid traverse can be altered using the handwheel.

Feed override is limited to 100% even if a value higher than 100% is displayed.

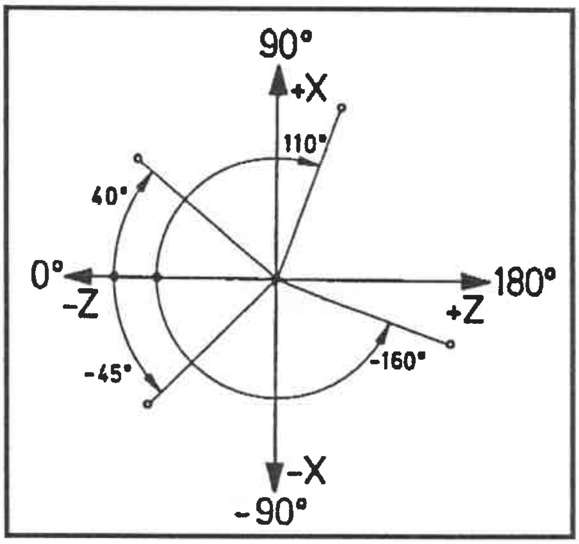
Linear movement
G1

Linear, G1
If the tool is to move in a straight line, i.e. to turn a cylinder or cone, path condition G1 must be programmed.

Required addresses:
After selecting path condition G1, the control system requests the following inputs:

DIAMETER	X:
LENGTH	Z:
ANGLE	A:
CURVE	
N. TANG./CHAMFER	B:
SPECIAL FEED	E:
INTERSECT. SELECTION	Q:

Meaning of A:
The sign of address A can be seen in the drawing on the right.



Note: The origin of the coordinates in the drawing must be regarded as the starting point of the programmed straight line.

Meaning of B:
In addition, it is possible to program a straight line with a curve or chamfer.
B +: radius of the curve
B -: width of the chamfer

If the Straight line has a circular contour element,but no tangential transition, it is imperative to program b = 0

Meaning of E:
A Special feed can be defined under the address E for the execution of the curve or the chamfer (address B). Depending on the type of feed previously defined in the program run, the value entered under E is executed in mm/min (feed in minutes) or in mm/rev (feed in revolutions)

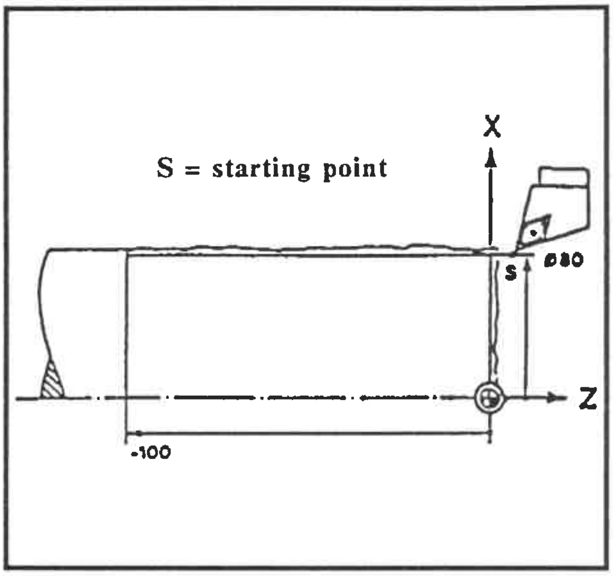
Meaning of Q:
If the calculation of the end point leads to two possible solutions, the first (nearest) point of intersection is chosen by programming Q = 0 or by striking the confirmation key. The second (distant) point of intersection can be chosen by programming Q = 1.

Examples for programming with A, B and Q can be found in section 3.3

G1

Example
G1 with absolute dimension programming.
A blank rod is to be turned to 80 mm diameter and 100 mm length.

N 1 G96 G95 F0.5 S180 T1 M4 M7
N 2 G0 X80 Z5
N 3 G1 Z-100
N 4 G1 X82
N 5 G0 Z0
N 6 G1 X-1.6
N 7 G0 X100 Z50 M30



Explanation:

N1 The first block contains the "starting conditions".
Feed im mm/rev., feed 0.5 mm/rev., F constant 180 m/min, spindle counter-clockwise, coolant on, tool 1
(this is to have the cutting dimensions I+0.8 K+0.8).

N 2 Rapid traverse to starting point.

N 3 Straight line interpolation, the tool traverses at programmed feed to position Z-100. As the X-value X80 is to be retained, no X is programmed.

N 4 Straight line interpolation G1, tool feed to X82.
It is then positioned 1 mm from the workpiece.

N 5 Rapid traverse to Z0.

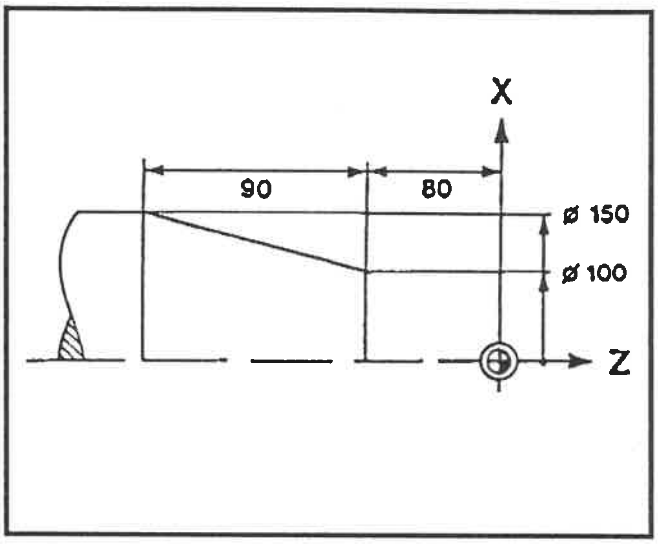
N 6 Straight line interpolation, tool feed to X-1.6,
i.e. the cross surface is being turned.

N 7 Return in rapid traverse, program end.

G1

Example
G1 with incremental dimension programming.

N 0 G90 G96 F0.5 S180 T1 M4 M7 M41
N 1 G0 X100 Z1
N 2 G91 G1 Z-81
N 3 G1 X25 Z-90
N 4 G90 G0 X200 Z50 M30



Explanation:

N 0 Start conditions.

N 1 Traverse to start point in absolute dimension.

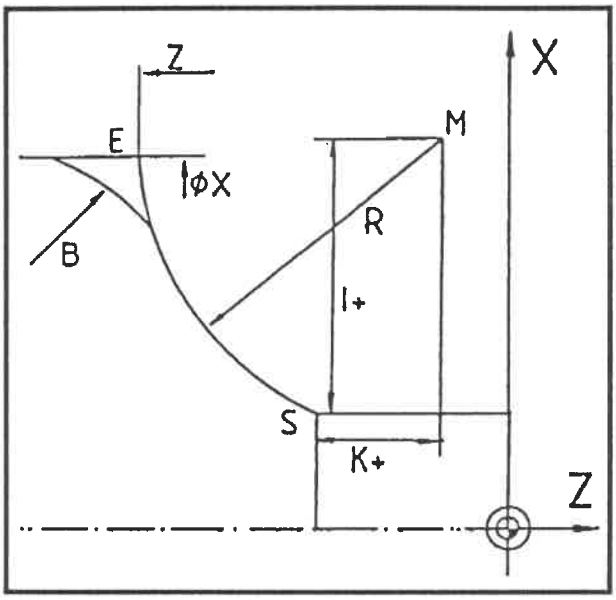
N 2 Incremental dimension, straight line interpolation.
The tool runs 81 mm against the Z-axis direction.

N 3 Straight line interpolation G1. The tool moves, at the same time, 25 mm in X direction (radius dim.) and 90 mm against Z-direction (**absolute position: X150, Z-170**).

N 4 Absolute dimension, rapid traverse away from workpiece, program end.

Circular movement G2

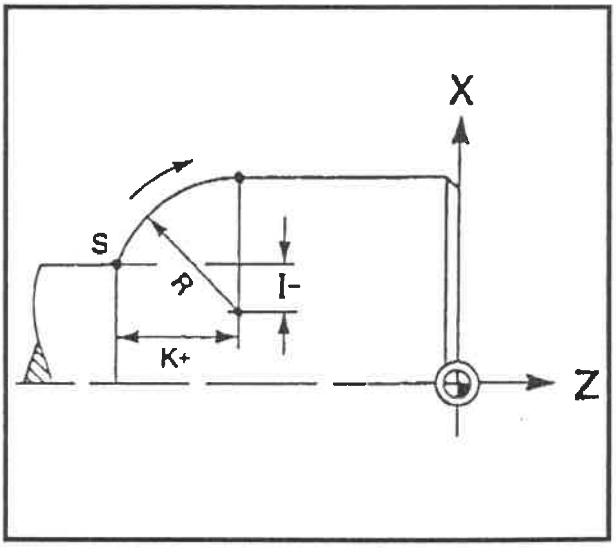
Circular arc clockwise with indication of centre in incremental dimension, G2
The tool moves **clockwise** at the given feed rate in a circular arc.



Required addresses
After selecting path condition G2, the control system requests the following inputs:

FINAL POSITION	X:
FINAL POSITION	Z:
RADIUS	R:
INTERSECT. SELECTION	Q:
CENTRE OF CIRCLE (X)	I:
CENTRE OF CIRCLE (Z)	K:
CURVE = B+ / N.TANG. =	B0:
SPECIAL FEED	E:

When the first and last point of the circular arc are entered, the centre need not be programmed.



Meaning of I and K
Depending on the location of the centre of the circle, I and K are either positive or negative.
Viewed from **starting point S** parallel to the coordinate axes towards the centre of the circle:

In direction of X-axis:	I+
against direction of X-axis:	I-
in direction of Z-axis:	K+
against direction of Z-axis:	K-

(Coordinate I is entered as a **radius value**).
Note: If the centre point coordinates I and K have not been programmed, the control calculates the centre point that produces the shorter circular arc.

Meaning of B
In addition, it is possible to program the circular arc with a subsequent curve.

B+: Radius of the curve

If there is no tangential transition between the circular arc and the **following contour**, B0 must be programmed (see also Chapter 3.3).

G2

Meaning of E
A special feed can be defined under the address E for the execution of the curve (address B). Depending on the type of feed previously defined in the routine, the value entered under E is executed in mm/min (feed in minutes) or in mm/rev (feed in revolutions).

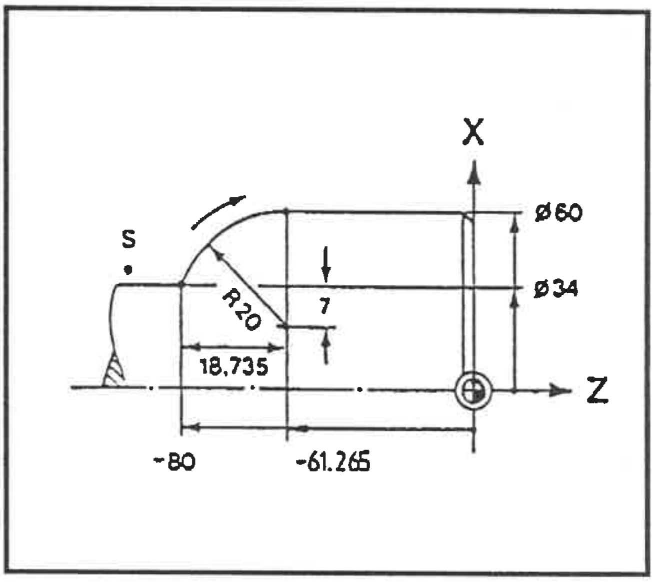
Meaning of Q
If the calculation of the end point lead to two possible solutions, the first (nearest) point of intersection is chosen by programming Q = 0 or by striking the confirmation key. The second (distant) point of intersection can be chosen by programming Q = 1.

Example:
Machining a circular arc, G2

```
N 0  G90  G96  F0.5  S180  T3  M4  M7
N 1  G0  X80  Z-85
N 2  G0  X35
N 3  G1  X34
N 4  G1  Z-80
N 5  G2  X60  Z-61.265  I-7  K18.735
N 6  G1  Z2
N 7  G0  X100  Z50  M30
```

or simply:

```
N 5  G2  X60  Z-61.265  R20
```



Explanation:

N 0 Start conditions.

N 1 When traversing to the starting point, always ensure that the path between the tool change point and starting point is clear. In this example an intermediate point is approached before the starting point in order to avoid collision.

N 3 Straight line interpolation. Approaching work surface.

N 4 The tool traverses, parallel to axis, to Z-80.

N 5 Circle interpolation G2. The target point is programmed under X and Z. Viewed from the circle starting point, the center of the circle is situated 7mm against the direction of the X-axis and 18.735 mm in direction of the Z-axis.

N 6 Straight line interpolation, longitudinal turning.

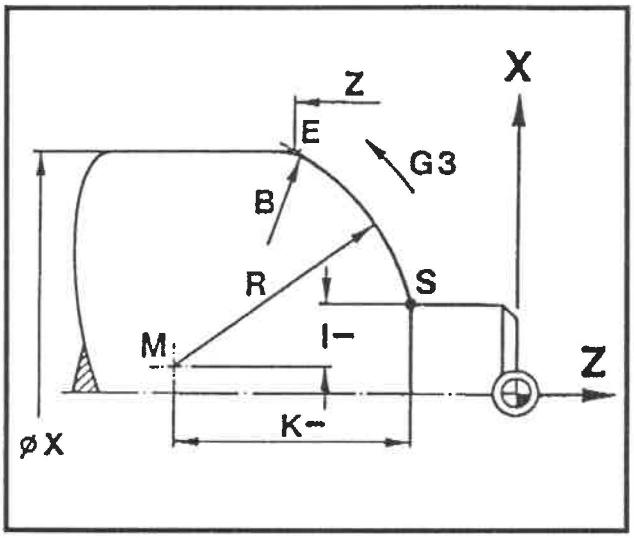
N 7 Rapid traverse away from work, program end.

Circular movement G3

Circular arc counter-clockwise, with incremental indication of centre, G3
The tool moves at the given feed speed **counter-clockwise** in a circular arc.

Required addresses
After selecting path condition G3, the control system requests the following inputs:

FINAL POSITION	X:
FINAL POSITION	Z:
RADIUS	R:
INTERSECT. SELECTION	Q:
CENTRE OF CIRCLE (X)	I:
CENTRE OF CIRCLE (Z)	K:
CURVE = B+ / N.TANG. =	B0:
SPECIAL FEED	E:



When the first and last point of the circular arc are entered, the centre need not be programmed.

Meaning of I and K
Depending on the location of the centre of the circle, I and K are either positive or negative.
Viewed from **starting point** S parallel to the coordinate axes towards the centre of the circle:

In direction of X-axis:	I+
against direction of X-axis:	I-
in direction of Z-axis:	K+
against direction of Z-axis:	K-

(Coordinate I is entered as a radius value).
Note: If the centre point coordinates I and K have not been programmed, the control calculates the centre point that produces the shorter circular arc.

Meaning of B
In addition, it is possible to program the circular arc with a subsequent curve.

B+: Radius of the curve

If there is no tangential transition between the circular arc and the **following contour**, B0 must be programmed (see also Chapter 3.3).

G3

Meaning of E
A special feed can be defined under the address E for the execution of the curve (address B). Depending on the type of feed previously defined in the routine, the value entered under E is executed in mm/min (feed in minutes) or in mm/rev (feed in revolutions).

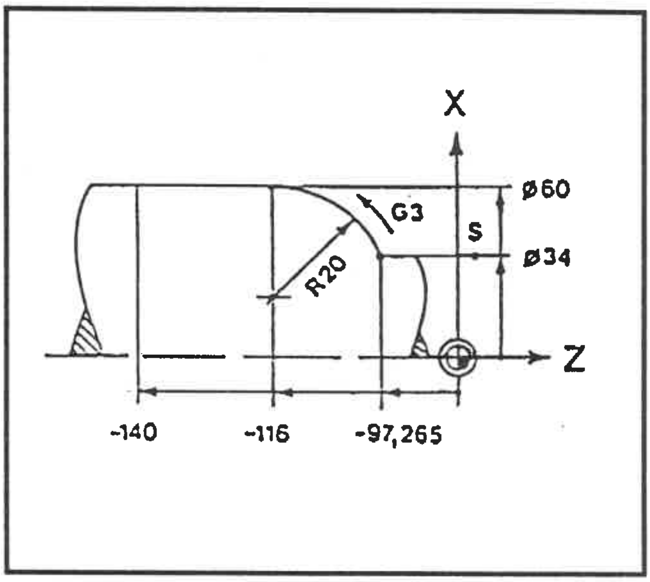
Meaning of Q
If the calculation of the end point leads to two possible solutions, the first (nearest) point of intersection is chosen by programming Q = 0 or by striking the confirmation key.
The second (distant) point of intersection can be chosen by programming Q = 1.

Example:
Machining a circular arc, G3

N 1 G0 X34 Z2
N 2 G1 Z-97.265
N 3 G3 X60 Z-116 I-7 K-18.735
N 4 G1 Z-140
N 5 G0 X100 Z50

or simply:

N 3 G3 X60 Z-116 R20



Explanation:
N 0 Start conditions.

N 1 Rapid traverse to starting point

N 2 Straight line interpolation, longitudinal turning up to start of circle.

N 3 Circle interpolation G3, destination point is programmed under X and Z. Viewed from the circle starting point the center of the circle is situated at 7 mm against the direction of the X-axis and at 18.735 mm against the direction of the Z-axis.

N 4 Longitudinal turning

N 5 Rapid traverse away from work, program end.

Period of dwell
G4

Period of dwell, G4
A period of dwell and subsequently the following program block are executed.

Required address
After selecting path condition G4, the control system requests the following inputs:

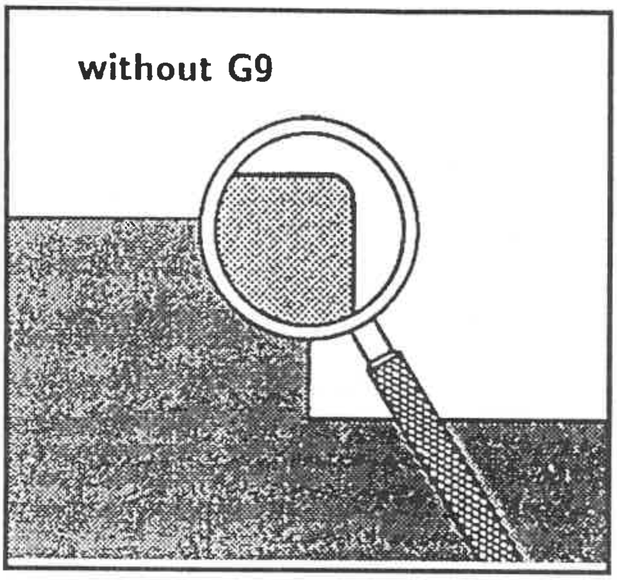
TIME F:

The period of dwell is programmed in seconds under address F. Maximum time input 99.9 s. If G4 and a traverse movement are programmed together in one block, then the period of dwell is activated after the traverse movement is finished.

Programming
N... G4 F1.5

Precision stop
G9

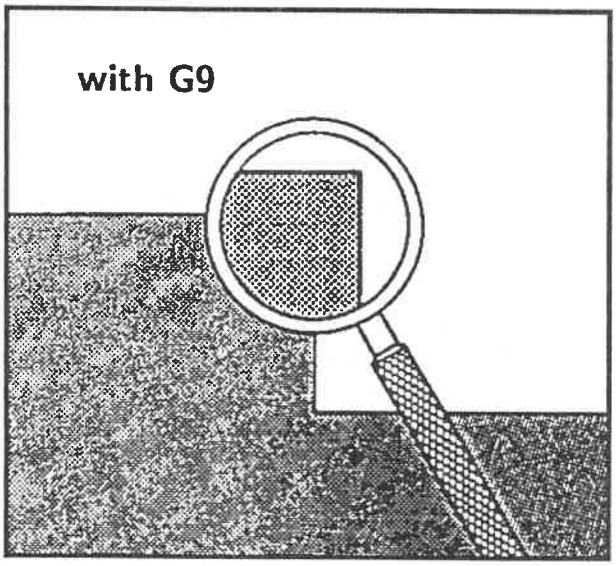
Precision Stop, G9
Usually all program blocks are executed continuously without stop. At large feed rates this leads to the rounding of edges. The higher the path speed, the larger the roundance. If sharp edges are to be machined the block concerned has to be programmed with G9. Thus, feed at the edge is reduced to zero before the next movement is executed. This function works with traverse movements with G1, G2, G12, G3 and G13.



Required address
No address is requested by the control system.

Programming without G9:
N... G1 Z200
N... G1 X100

If G9 is not written in addition in a block with G1, the result is a roundance of edges when the target position is approached.



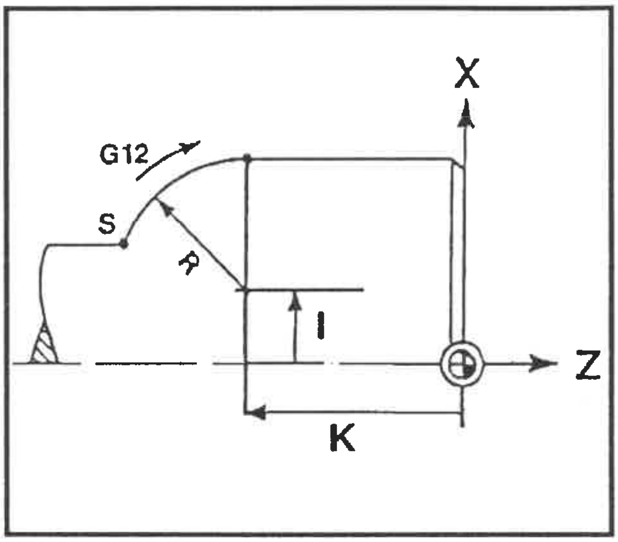
Programming with G9:
N... G1 G9 Z200
N... G1 X100

The tool tip stops precisely at the programmed position (Z200) and then executes the movement in transverse direction.

G9 in conjunction with milling movements on the C-axis
Using the function G9 it is also possible to avoid unwanted rounding of corners during frontface and circumferential milling (C-axis machining). In this case the function G9 is written into an NC block together with the programmed traverse movement of the C-axis (the function affects the traverse movements with G101, G102, G103, G111, G112 and G113). While the NC block is being executed, the actual position of the C-axis is monitored to ensure that it remains within the tolerances defined under parameter N129. The NC block is not deemed executed until the actual value for C-axis position is in the specified tolerance range. Only then is the next NC block is executed.

Circular movement G12

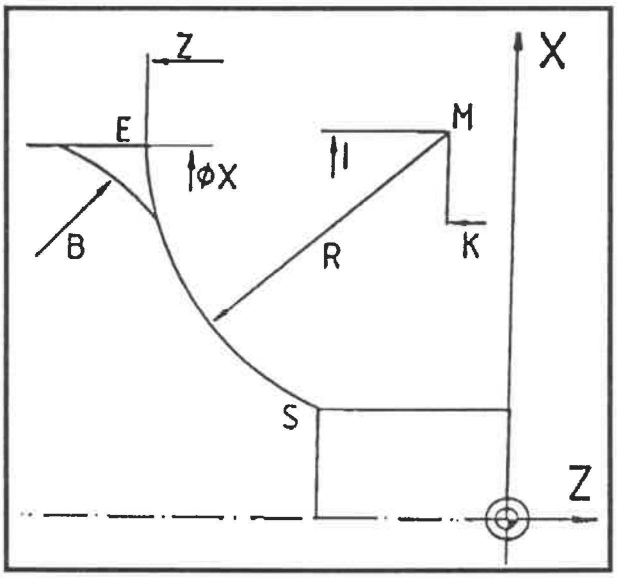
Circular arc clockwise, with indication of centre in absolute dimension, G12
The tool moves at the given feed rate **clockwise** in a circular arc.



Required addresses
After selecting path condition G12, the control system requests the following inputs:

FINAL POSITION	X:
FINAL POSITION	Z:
CENTRE OF CIRCLE (X)	I:
CENTRE OF CIRCLE (Z)	K:
RADIUS	R:
CURVE = B+ / N.TANG. =	B0:
INTERSECT. SELECTION	Q:
SPECIAL FEED	E:

When the first and last point of the circular arc are entered, the centre need not be programmed.



Meaning of I and K
Depending on the location of the centre of the circle, I and K are either positive or negative.
Viewed from the NC **zero point** parallel to the coordinate axes towards the centre of the circle:

In direction of X-axis:	I+
against direction of X-axis:	I-
in direction of Z-axis:	K+
against direction of Z-axis:	K-

(Coordinate I is entered as a radius value).
Note: If the centre point coordinates **I** and **K** have not been programmed, the control calculates the centre point that produces the shorter circular arc.

Meaning of B
In addition, it is possible to program the circular arc with a subsequent curve.

B+: Radius of the curve

If there is no tangential transition between the circular arc and the **following contour**, B0 must be programmed.

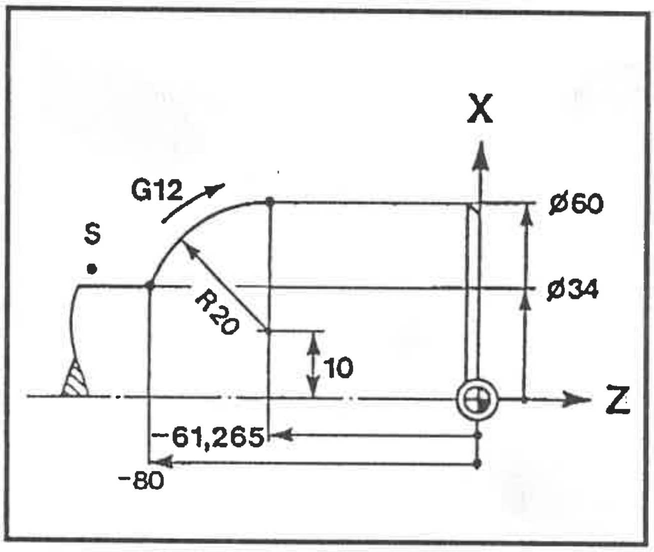
G12

Meaning of E
A special feed can be defined under the address E for the execution of the curve (address B). Depending on the type of feed previously defined in the routine, the value entered under E is executed in mm/min (feed in minutes) or in mm/rev (feed in revolutions).

Meaning of Q
If the calculation of the end point leads to two possible solutions, the first (nearest) point of intersection is chosen by programming Q = 0 or by striking the confirmation key. The second (distant) point of intersection can be chosen by programming Q = 1.

Example:
Machining a circular arc, G12

```
N 0 G96  F0.5  S180  T3  M4  M7
N 1 G0  X80  Z-85
N 2 G0  X35
N 3 G1  X34
N 4 G1  Z-80
N 5 G12  X60  Z?  I10  K-61.265  R20
N 6 G1  Z2
N 7 G0  X100  Z50  M30
```



Explanation:
N 0 Start conditions.

N 1 When traversing to the starting point, always ensure that
N 2 the path between the tool change point and starting point is clear. In this example an intermediate point is approached before the starting point in order to avoid a collision.

N 3 Straight line interpolation. Approaching work surface.

N 4 The tool traverses, parallel to axis, to Z-80.

N 5 Circle interpolation. The destination point is programmed under X and Z?, the centre of the circle in absolute coordinates I und K. The value for I is entered as a radius value (see simplified geometry programming).

N 6 Straight line interpolation, longitudinal turning.

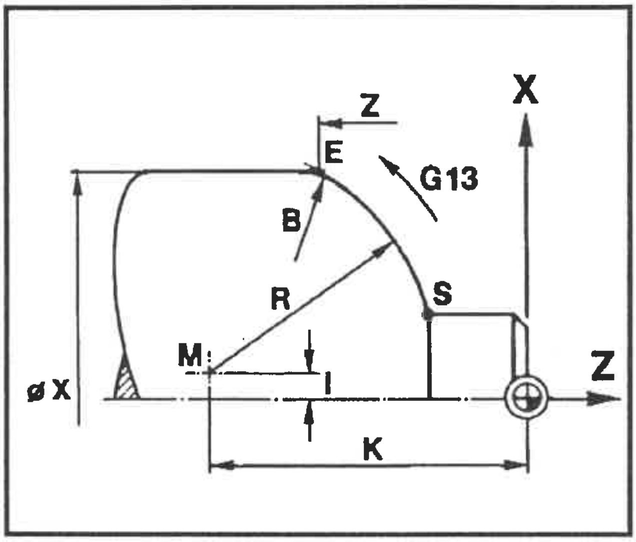
N 7 Rapid traverse away from work, program end.

Circular movement G13

Circular arc counter-clockwise, with indication of centre in absolute dimensions, G13
The tool moves at the given feed rate **counter-clockwise** in a circular arc.

Required addresses
After selecting path condition G13, the control system requests the following inputs:

FINAL POSITION	X:
FINAL POSITION	Z:
CENTRE OF CIRCLE (X)	I:
CENTRE OF CIRCLE (Z)	K:
RADIUS	R:
CURVE = B+ / N.TANG. =	B0:
INTERSECT. SELECTION	Q:
SPECIAL FEED	E:



When the first and last point of the circular arc are entered, the centre need not be programmed.

Meaning of I and K
Depending on the location of the centre of the circle, I and K are either positive or negative.
Viewed from the NC **zero point** parallel to the coordinate axes towards the centre of the circle:

In direction of X-axis:	I+
against direction of X-axis:	I-
in direction of Z-axis:	K+
against direction of Z-axis:	K-

(Coordinate I is entered as a radius value).
Note:If the centre point coordinates I and K have not been programmed, the control calculates the centre point that produces the shorter circular arc.

Meaning of B
In addition, it is possible to program the circular arc with a subsequent curve.

B+: Radius of the curve

If there is no tangential transition between the circular arc and the **following contour,** B0 must be programmed.

G13

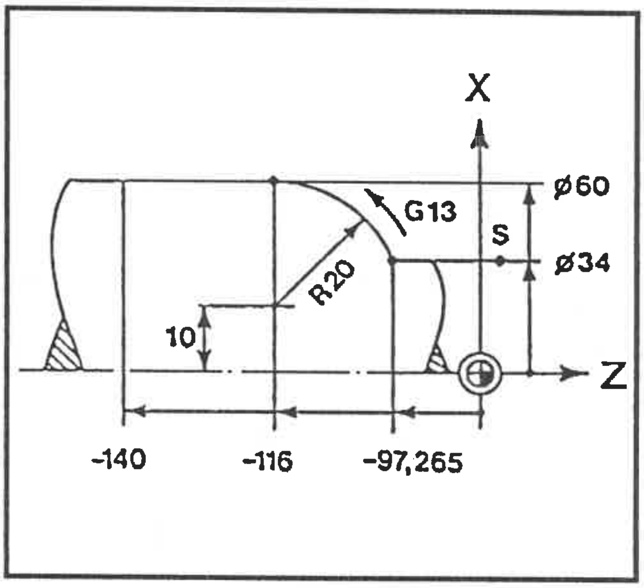
Meaning of E
A special feed can be defined under the address E for the execution of the curve (address B). Depending on the type of feed previously defined in the routine, the value entered under E is executed in mm/min (feed in minutes) or in mm/rev (feed in revolutions).

Meaning of Q
If the calculation of the end point leads to two possible solutions, the first (nearest) point of intersection is chosen by programming Q = 0 or by striking the confirmation key. The second (distant) point of intersection can be chosen by programming Q = 1.

Example:
Machining a circular arc, G13

N 0
N 1 G0 X34 Z2
N 2 G1 Z-97.265
N 3 G3 X60 Z-116 I10 K-116 R20
N 4 G1 Z-140
N 5 G0 X100 Z50 M30

other possibility:
N 3 G13 X60 Z-116 R20



Explanation:

N 1 Rapid traverse to starting point

N 2 Straight line interpolation, longitudinal turning up to start of circle.

N 3 Circle interpolation G13. The destination point is programmed under X and Z, the centre of the circle is programmed in absolute coordinates. The value for I is entered as a radius value.

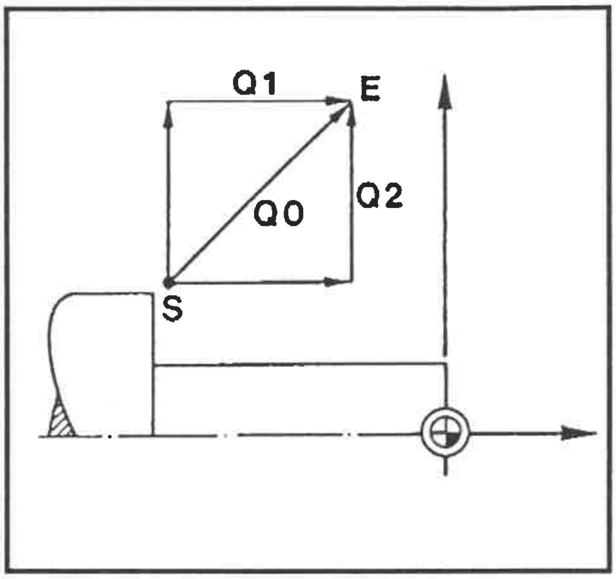
Further possibility:
It is sufficient to enter R instead of I and K, since the starting point and end point of the circle are programmed.

N 4 Longitudinal turning

N 5 Rapid traverse away from workpiece, program end.

Tool change position G14

Engaging tool change point, G14
Using function G14, the tool change point established during setup and stored under **parameter** N202 can be engaged by three different methods.



Required addresses
After selecting G14, the control system requests the following inputs:

SEQUENCE OF MOVEMENT Q:

This means:

Q0: Diagonal traverse path (or program Q only).

Q1: Traverse path first in X direction, then in Z direction.

Q2: Traverse path first in Z direction, then in X direction.

If no Q is entered, movement is as for Q0.

Caution
In addition to its own function, G14 also contains the function G90 (switchover to absolute dimensions). If the function G14 is utilized to approach the tool change point while executing a program which is programmed in incremental dimensioning, the function G90 is active after G14 has been executed. If the dimensions which follow in the program under the respective address parameters are designed for incremental dimensioning, the function G91 in the block after G14 must definitely be reprogrammed.

Speed limitation
G26

Speed limitation of the main spindle, G26
With this function the speed of the main spindle can be limited during the course of the main program.

Required address
After selecting function G26, the control system requests the following input.

SPEED S:

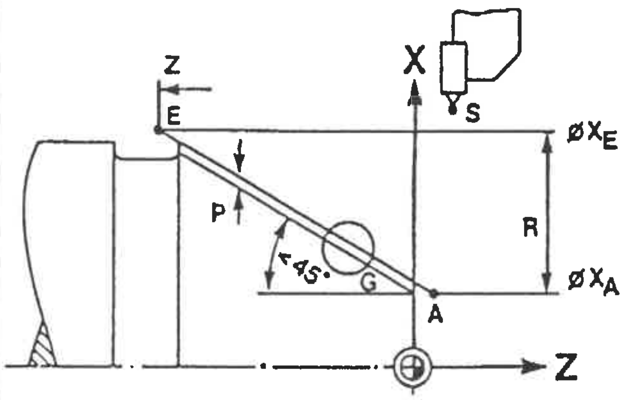
Meaning
With this function, the speed of the main spindle can be limited to a speed value in **[1/min]** entered under address S. This speed programmed under G26 takes effect until the end of the program or until replaced by renewed programming of G26.

Note
If the speed programmed with G26 is **higher** than the maximum speed determined under **Parameter N501 (address MD)**, then the speed limitation from **Parameter N501** is treated with priority.

Thread cycles
G31

The following threads can also be machined using cycles:
Longitudinal threads,
tapered threads,
and special threads
All movements, such as approach, cutting, retracting and returning are generated automatically.
All threads, with the exception of special threads, can be programmed in one block.

Tapered thread, LONGITUDINAL THREADING cycle, G31
Cycle G31 can be used to program longitudinal and tapered threads **(up to max. 45° to the Z-axis)** with constant pitch.



detail G see below diagram

Required addresses:
After selecting G31, the control system requests the following inputs:

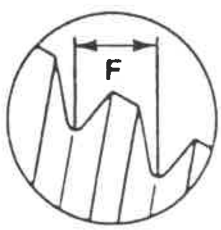
DIAMETER X:
Outside diameter of the thread at end point E.

LENGTH Z:
End point of the thread (longitudinal).

APPROACH (X) I:
Approach in cross direction

OFFSET 2ND CUT K:
Longitudinal offset in Z direction with every 2nd cut: for alternating machining of the left and righthand thread face.

detail G



THREAD DEPTH P:
Thread depth

RADIUS DIFFERENCE R:

For tapered threads, difference between radius at end point and radius at starting point:
R = (øXE - øXA)/2
For a falling contour, the value is **negative** (see upper diagram).

G31

THREAD LEAD F:
Thread lead in longitudinal direction

NO. OF LOST MOTION RUNS Q:
Additional no motion runs can be programmed under this address after the last machining cut in order to reduce cutting pressure.

SWITCHING OFF
REMAINING CUTS B:
B = 1 switches off
B = 0 does not switch off
(striking the confirmation key has the same effect as B = 0).

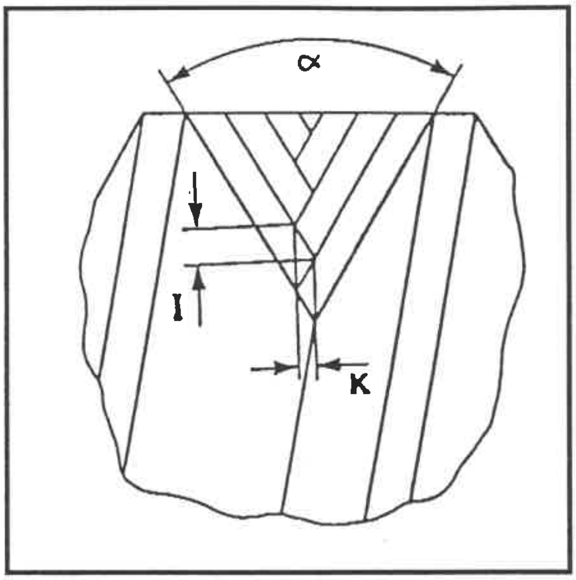
No speed alteration may be made in cycle G31. **Never use V constant** (G96 is active) or **override.** Futhermore it has to be made sure that the programmed speed is valid for the slide(s) carrying out the threading cut, otherwise a new speed must be programmed.

Starting position
At least double the thread depth above maximum thread diameter and approx. .5 mm before the start of the thread.

Cycle procedure
The control system calculates all necessary cuts. The following applies: As many cuts are carried out as are required to reach, starting at the starting point (**X-value**), the point programmed under X and Z. If the thread depth P is not a multiple of the cutting depth programmed under I, the first cut will have a smaller cutting depth. The last four cuts have 1/2, 1/4, 1/8 and 1/8th of the programmed cutting depth if B = 1 is not programmed. After each cut, the tool runs in rapid traverse to the Z coordinate of the starting position. At the end of the cycle the tool is once more positioned at the starting point.

G31

Alternating machining of flanks
By programming a value for address K, the flanks can be machined alternately.



Calculating K:
K is calculated from thread angle and cutting depth.

$$K = \tan \frac{a}{2} \cdot I$$

Example
Metric ISO threads have a thread angle *a* of 60°. The cutting depth I is to be 0.3 mm.

$$K = \tan \frac{60}{2} \cdot 0.3\ \text{mm}$$

$$= 0.577 \cdot 0.3\ \text{mm}$$

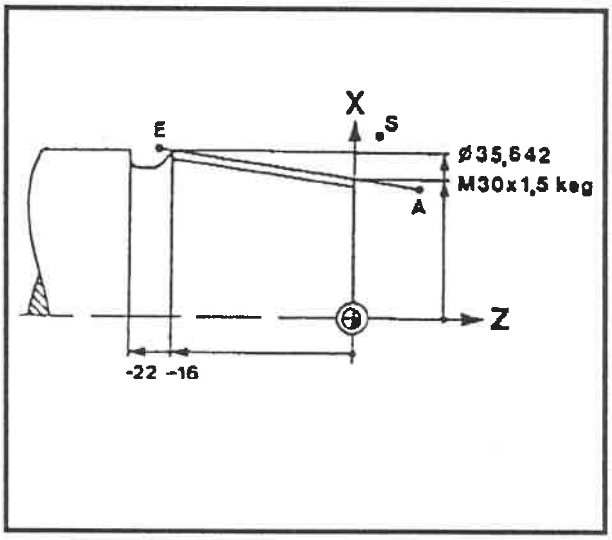
$$= 0.172\ \text{mm}$$

G31

Example

Programming
N 0 G97 F0.5 S1000 T1 M4 M7
N 1 G0 X30 Z2
N 2 G1 Z0
N 3 G1 X35.642 Z-16
N 4 G85 Z-22 I1.5 K6
N 5 G0 X200 Z100
N 6 G0 G97 X45 Z5 S150 T5
N 7 G31 X37.053 Z-20 I0.3 K0.166 P0.92 R4.408 F1.5
N 8 G0 X200 Z100 M30

Note
Starting point is at X45, Z5.
End point is at X37.053, Z-20.
Thread start A is at X = 28.237 before workpiece.



Explanation

N 0 Start conditions.

N 1 Rapid traverse to starting point.

N 2 Feed traverse to workpiece.

N 3 Turn taper.

N 4 Thread clearance cut cycle.

N 5 Rapid traverse to tool change point, tool change.

N 6 Rapid traverse to starting position (X45, Z5) for thread. Speed 150 rev./min.

N 7 Threading cycle using following information: end point, cutting and thread depth, radius difference and thread lead.

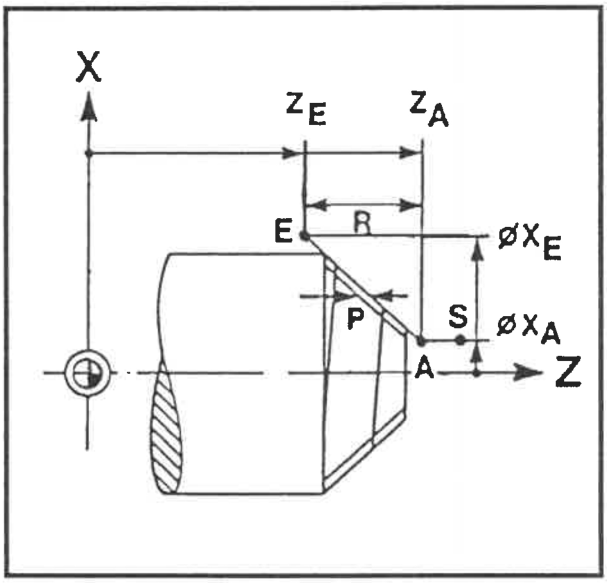
N 8 Rapid traverse to tool change point, end of program.

Thread cycles G32

Transversal thread , tapered thread
THREAD TRANSVERSAL cycle, G32
For machining cross and tapered threads (up to max. 45° to the Z-axis) with constant lead.

S = starting point
A = start point of thread
E = end point of thread

Required addresses:
After selecting G32, the control system requests the following inputs:



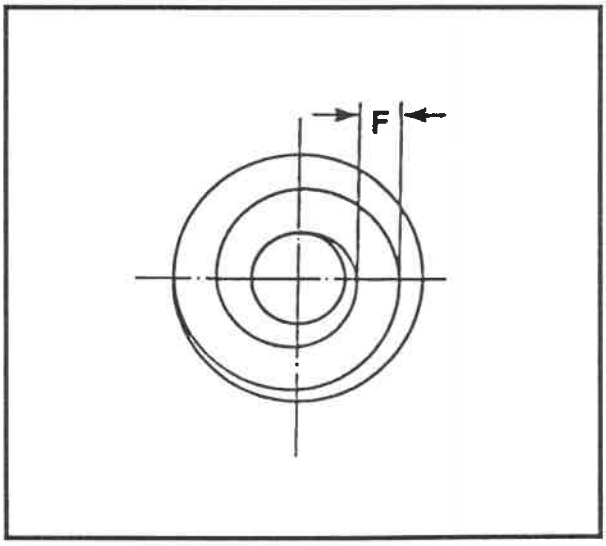
DIAMETER X:
End diameter of the thread

LENGTH Z:
Z position at end of thread

OFFSET 2ND CUT I:
Offset in cross direction for every 2nd cut. For alternating machining of left and right flank.

APPROACH (Z) K:
Approach in Z-direction

THREAD DEPTH P:



THREAD LENGTH R:
For tapered threads, the difference between start and end length. For a falling contour, the value is negative.
R = ZA - ZE

THREAD LEAD F:

NO. OF NO MOTION RUNS Q:
Additional no motion runs can be programmed under this address after the last machining cut in order to reduce cutting pressure.

SWITCHING OFF REMAINING CUTS B:
B = 1 switching off
B = 0 no switching off
(striking the confirmation key has the same effect as B = 0).

G32

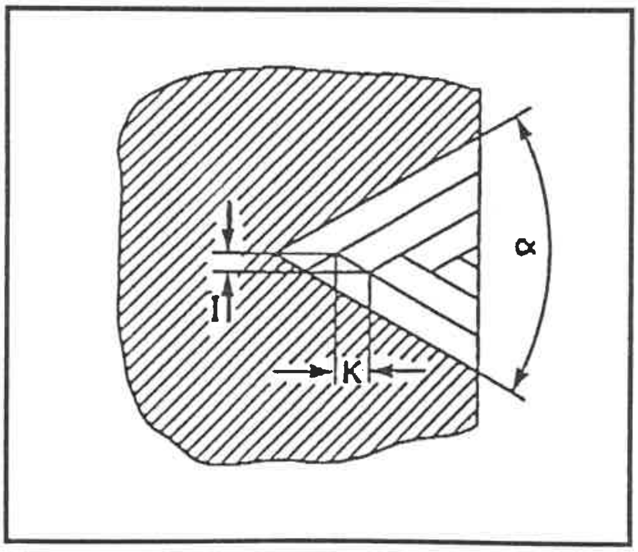
Programming
No speed alteration may be made in cycle G32. Futhermore it has to be made sure that the programmed speed is valid for the slide(s) carrying out the threading cut, otherwise a new speed must be programmed.

Please note the following for the **starting position in X:**
Depending on the path velocity in X, the forward motion of the turning spindle with the feed spindle should be **2 to 3 times the lead** of the thread to be machined.

Cycle procedure
The control system calculates all necessary cuts. The following applies:
As many cuts are carried out as are required to reach, starting at the starting point **(Z value)**, the point programmed under X and Z.
If the thread depth P is not a multiple of the cutting depth programmed under I, the first cut will have a smaller cutting depth.
The last 4 cuts have 1/2, 1/4, 1/8 and 1/8th of the programmed cutting depth.
After each cut, the tool runs in rapid traverse to the X coordinate of the starting position.
At the end of the cycle, the tool is once more positioned at the starting point.

Alternating machining of flanks:
As for G31, but with I and K swapped.

$$\boxed{I = \tan \frac{\alpha}{2} \cdot K}$$



Example for programming
see example 4 multiple-turning cycles and
example thread cycles

Thread cycles
G33

SPECIAL THREAD cycle, G33
With G33, synchronization is only established between main and feed spindles.
All traverse paths must be programmed individually.

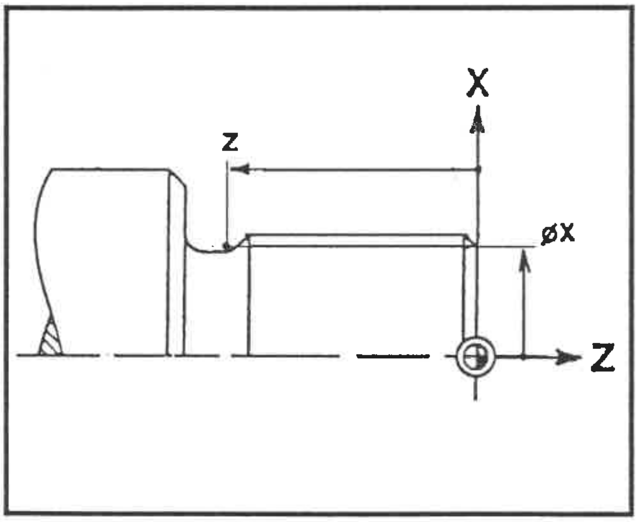
G33 performs **one** threading cut.
The following are possible:
Longitudinal thread,
cross thread,
tapered thread and
Special threads, single and
multiple start with **constant** and
variable lead.

Required addresses:
After selecting G33, the control system requests the following inputs:

DIAMETER X:
End point of the threading cut

LENGTH Z:
End point of the threading cut

THREAD LEAD F:



Programming
Each threading cut must be programmed individually.
Approach, retract and return movements must each be programmed in separate blocks. Any number of thread cuts can be programmed successively with G33.

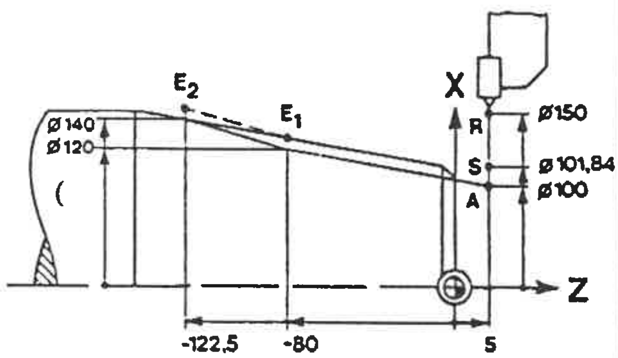
Special threads
G33 can also be programmed linked to G83. This enables the machining of special threads (without clearance cut, with slanted lead-in and lead-out).
For this link of program, too, speed may not be altered.

G33

Example
Special thread consisting of two tapered threads of equal lead merging into each other (F = 1.5 mm). First thread section tapered at 1:4.25, second thread section tapered at 1:2.125.

Programming

```
...
N10 G0 X101.84 Z5
N11 G83 X100 Z5 I0.15
N12 G33 X120 Z-80 F1.5
N13 G33 X140 Z-122.5 F1.5
N14 G0 X150 Z5
N15 G80
...
```



starting point S
link A
1st thread section E1)
2nd thread section E2)
Return R)
link established)

Explanation
Traverse to starting point S.

Program for multiple cycle.

Start point of last machining step is X100, Z5.

"Contour description" for cycle G83 consists of 2 threading cuts with G33. Approach by 0.15 mm for each new cut.

The last cut is carried out once point X100 Z5 has been reached.

At the end of machining, the tool is positioned at X150, Z5.

Thread cycles G35

METR. ISO THREADING cycle, G35
Metric ISO threads (row 1) are very simple to program using G35, as the control system autonomously calculates all values which it requires to produce a standard thread.

Required addresses:
After selecting G35, the control system requests the following inputs:

DIAMETER X:
Nominal thread diameter

LENGTH Z:
End point of the thread (longitudinal)

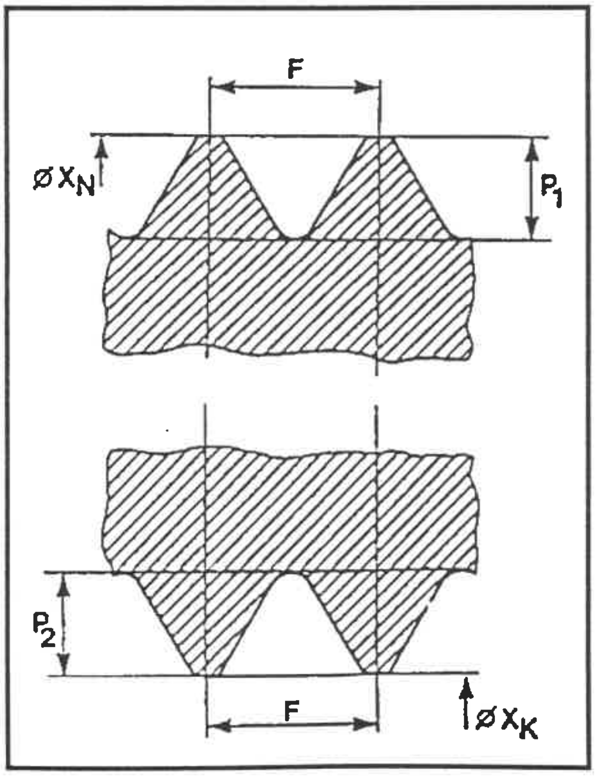
THREAD LEAD F:
Can be confirmed; the value is then obtained from DIN 13, row 1.

APPROACH (X) I:
Approach in cross direction; can be confirmed, in which case the value is automatically calculated by the control system.

NO. OF NO MOTION RUNS Q:
Additional no motion runs can be programmed under this address after the last machining cut in order to reduce cutting pressure.

SWITCHING OFF REMAINING CUTS B:
B = 1 switching off
B = 0 no switching off
(striking the confirmation key has the same effect as B = 0).

By programming the addresses accordingly (see table), G31 can also be used to produce metric ISO threads.
Futhermore it has to be made sure that the programmed speed is valid for the slide(s) carrying out the threading cut, otherwise a new speed must be programmed.



Starting position
At least double the thread depth above maximum thread diameter and approx. 5 mm before the start of the thread.

G35

DIN 13 Row I

Nominal diam. øXN	Lead F	Minor diameter øXK	Thread depth Screw P1	Thread depth Nut P2
1	0,25	0,729	0,153	0,135
1,2	0,25	0,929	0,153	0,135
1,6	0,35	1,221	0,215	0,189
2	0,4	1,567	0,245	0,217
2,5	0,45	2,013	0,276	0,244
3	0,5	2,456	0,307	0,271
4	0,7	3,242	0,429	0,379
5	0,8	4,134	0,491	0,433
6	1	4,917	0,613	0,541
8	1,25	6,647	0,767	0,677
10	1,5	8,376	0,920	0,812
12	1,75	10,106	1,076	0,947
16	2	13,835	1,227	1,083
20	2,5	17,294	1,534	1,353
24	3	20,752	1,840	1,624
30	3,5	26,211	2,147	1,894
36	4	31,670	2,454	2,165
42	4,5	37,129	2,760	2,436
48	5	42,587	3,067	2,706
56	5,5	50,046	3,374	2,977
64	6	57,505	3,681	3,248

Note: Feed rate for I and K will be calculated automatically by the control system using the values for feed and cutting speed.

Cancel graphics
G38

Cancel graphics, G38

The function G38 is used if the screen layout of the graphic simulation is to be erased at a certain point in the program and the graphic simulation is to be started anew with the NC blocks provided after this point in the program.

Required addresses

After G38 is selected the control system does not request any further entries.

Note

The function G38 must be programmed in a separate NC block.

Tool radius compensation
G40, G41, G42

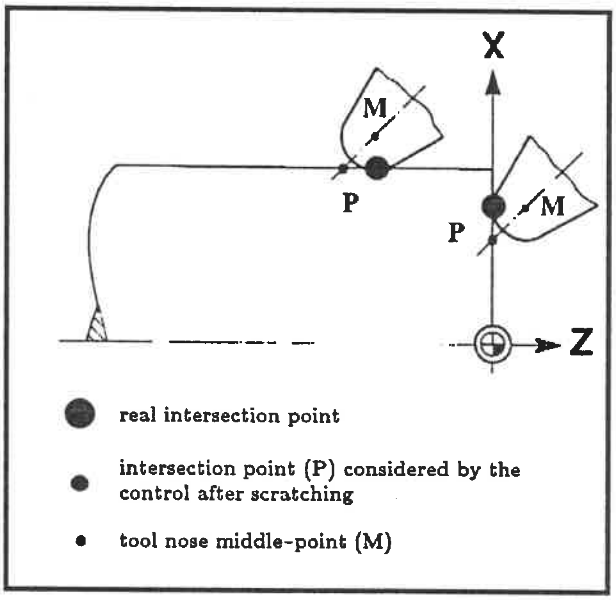
General
G40, G41 and G42 have two different functions depending on which work operation is selected:

SRK Correction of the real cutting point of the tool nose during turning

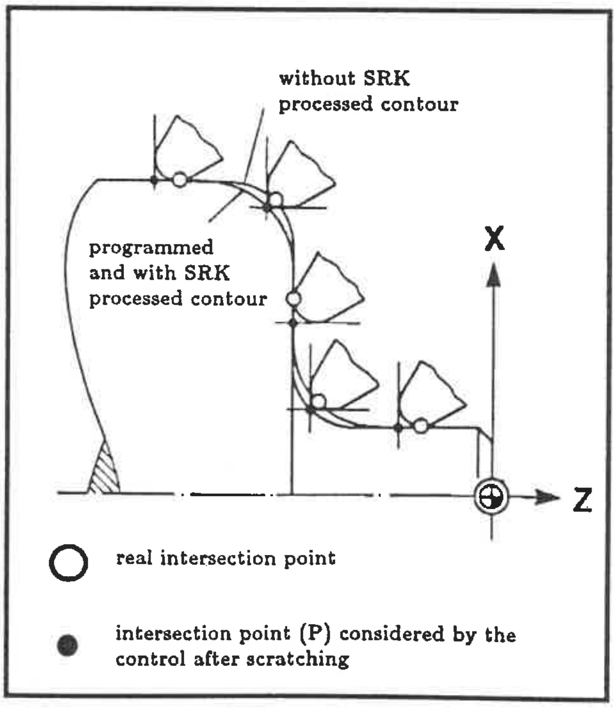
FRK Correction of the milling tool radius in terms of the contour programming referred to the middle of the milling tool during milling operations on the frontface and the circumference in conjunction with the C-axis

Purpose of tool nose radius compensation (SRK)
Th position of the intersection point on the throw-away insert changes during machining.

If complicated contours are to be turned with great precision, the position of the intersection point on the throw-away insert in relation to the contour must be taken into consideration. The control system handles this function when tool nose radius compensation is programmed.



The effect of tool nose radius compensation
Without tool nose radius compensation, point P is the reference point for the control system. This point is always on the programmed contour while the tool is traversing.
This causes slight inaccuracies during movements which do not run parallel to the X- or Z-axes (max. approx. 4/10 of the tool nose radius).
The surface emphasized in the illustration is not cut although it lies outside the programmed contour.



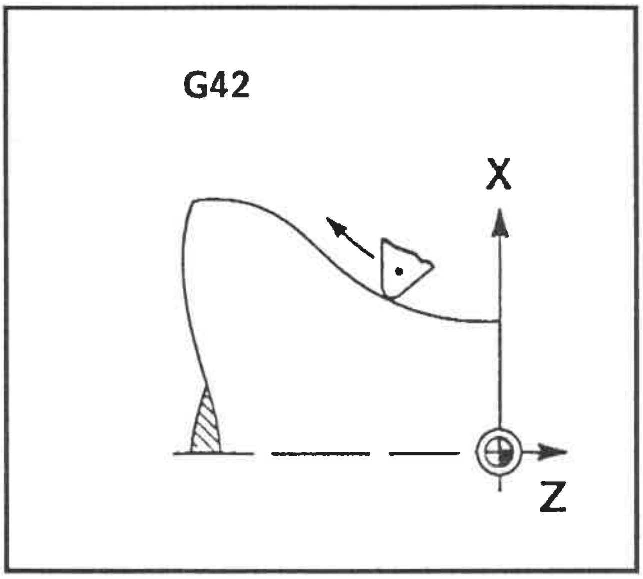
With tool nose radius compensation, the actual point of intersection always remains on the programmed contour.
The programmed and the machined contour are identical.

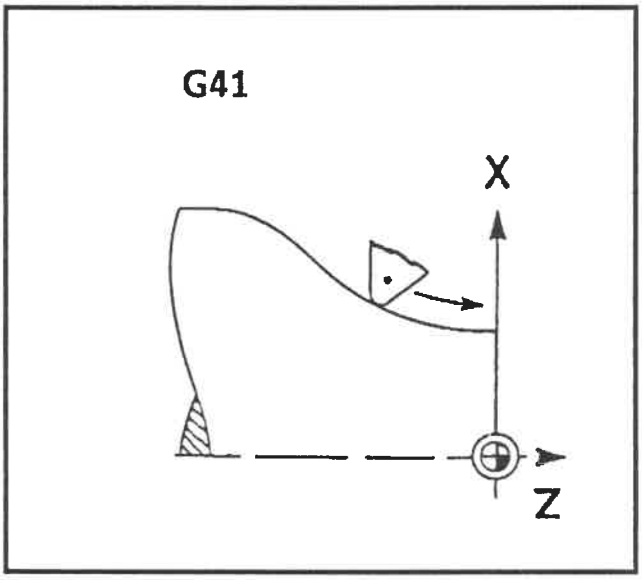
Note
Even with active SRK the current position of the theoretical tool nose tip value P is shown on the position display.

G40, G41, G42

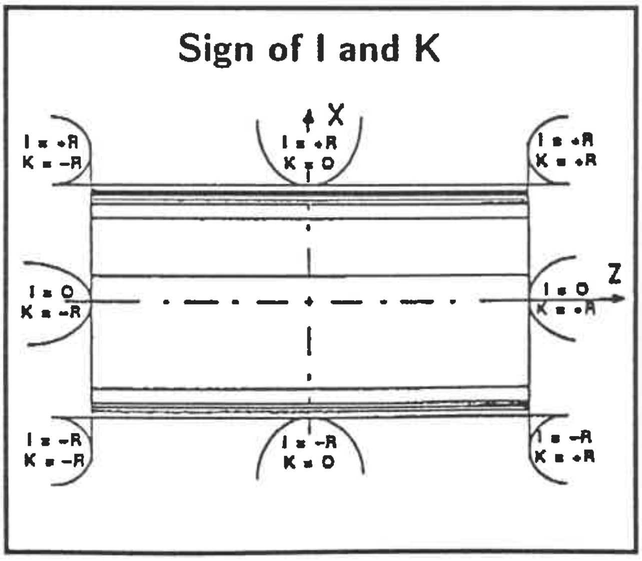
Activating tool nose radius compensation, G41, G42
Tool nose radius compensation is activated by G41 or G42.
The control system recognizes by means of G41/G42 where the tool is moving:

In traverse direction
To the left of the contour: **G41**
To the right of the contour: **G42**



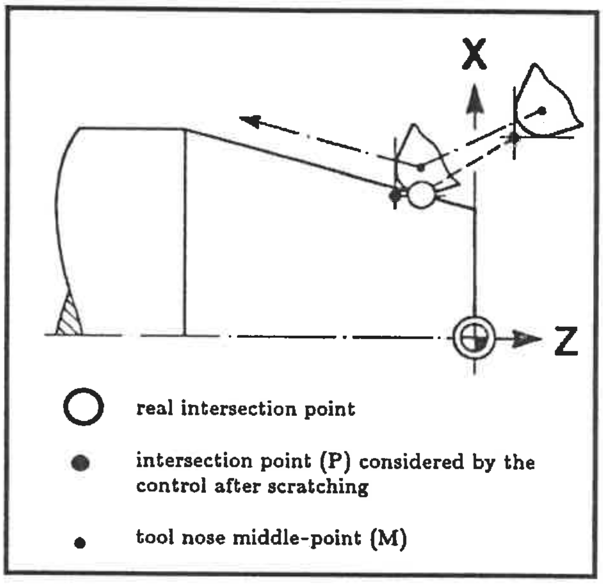


Position of the tool
The position of the tool in the tool holder is obtained from the values and the sign of I and K in the tool file (see fig.)



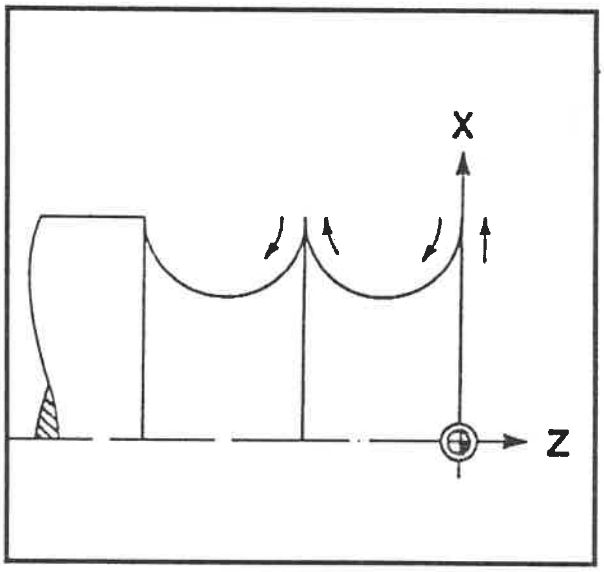
G40, G41, G42

Programming instructions for G41/G42
In the block containing G41 or G42, only one **straight line** feed movement (G00 or G01) may be programmed. The actual point of intersection is taken into consideration from the next block onward.



When tool nose radius compensation is active, **no** movements including a full **reversal of direction** may be programmed.

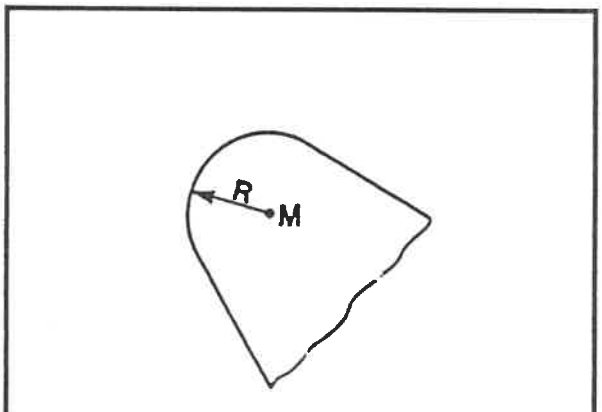
N...
N... G90 G01 X+...
N... G91 G01 Z0.01
N... G90 G01 X-...



Similarly, the functions from the "**nine hundred range**", the function G14, variable inputs V:{? ... and extended NC functions T[...], D[...] or G61 H{E... cannot be programmed as long as the SRK is active.

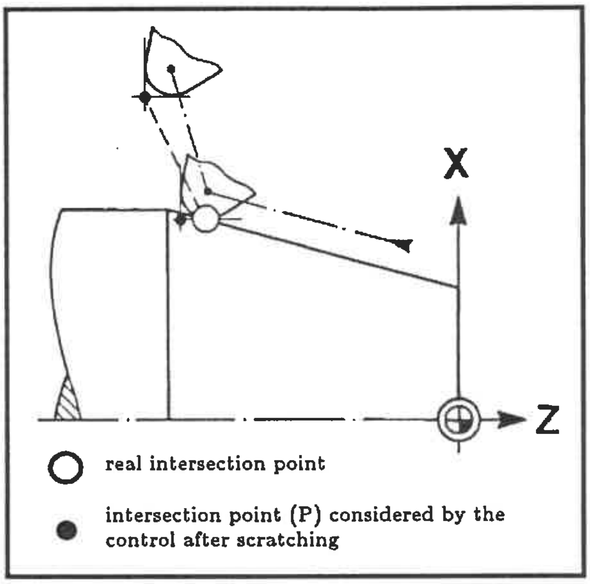
When tool nose radius compensation is active, the **tool dimensions**, including cutting edge radii, may **not be altered**.

Attention
In the case of contours containing internal arcs, make certain that the tool nose radius is at least 2 µm smaller than the radius of the arc, otherwise the tool geometry will preclude the machining of the contour and error message 5683 is displayed.



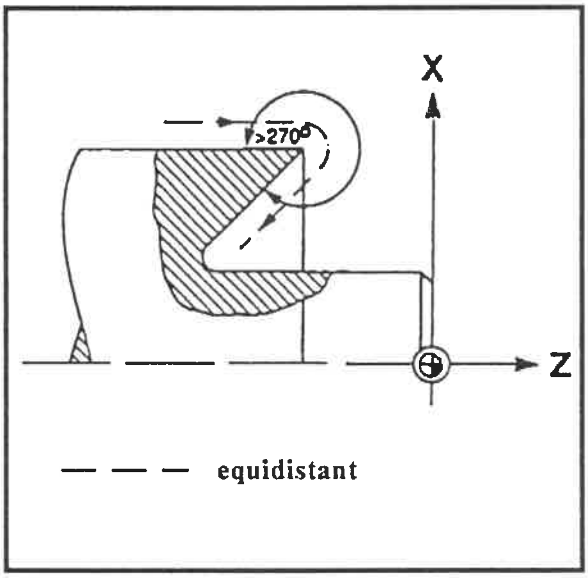
G40, G41, G42

Switching off tool nose radius compensation, G40
The control system includes tool nose radius compensation until the block before G40 is reached.
Do not program the block with G40 to contain more than one straight line feed movement with G0 and G1.
Afterwards the scratching point is considered again as intersection point by the control.

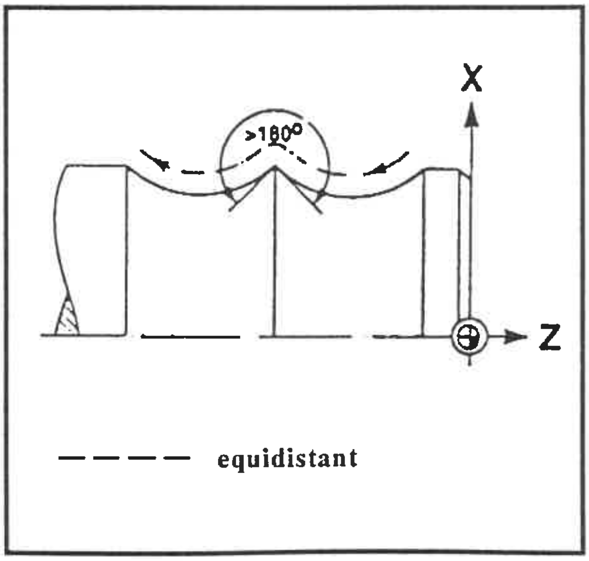


Input
The input of G40, G41 and G42 is done in the EDITOR operating mode (see chap. 4) by actuating the softkey pushbutton G FUNCTION DIRECT or G FUNCTION MENUE.

The tool path in tool nose radius compensation
The path described by the centre point of the tool nose is called an equidistant, i.e. a path with clearance from the contour. When two straight lines meet at an angle greater than 270°, the tool performs a "rolling" movement around the point of intersection, i.e. the centre point of the cutting edge moves in a circular arc around the point of intersection.



The same applies for the intersection of a circular arc with a straight line or a circular arc with a circular arc.
For angles between 180° and 270° the control system decides automatically whether the rolling movement is to be included.



G40, G41, G42

Cycles with tool nose radius compensation
G87 or G88 may not be programmed in blocks containing G40, G41 or G42.

Before threading and drilling cycles, the tool nose compensation must be switched off with G40.

In the grooving cycles (G861 to G864) select G41 or G42 so that the SRK function corresponds to the direction of the programmed contour, not the cycle sequence.

G40, G41 and G42 must always be programmed together with G00 or G01 in one block.

In cutting cycles the SRK should be switched on when approaching the contour. It should be switched off when retracting from contour.

If the end point of the following contour element is closer to the previous element than the measure forming the radius of the tool nose, contour deviations may occur.

Example:

% 2804 (main program)

```
...
...
N... G818  X...  Z...  I...
N... L4711
N... G80
...
...
```

% 4711 (subprogram)

```
...
...
N...  G42  G00  X...  Z...
...
...
...   description of contour
...
...
N...  G40  G01  X...  Z...
N...  M30
```

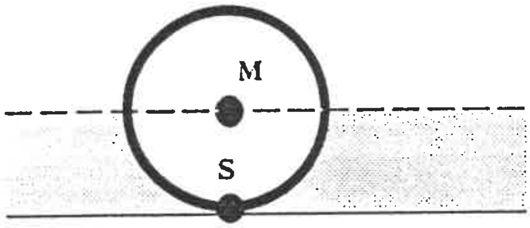
Tool nose radius compensation and interpreter stop
No interpreter stop may be programmed when tool nose radius compensation is active, i.e. none of the functions from the "nine hundred range" except G907 may be used while SRK is switched on.

Milling cutter radius compensation G40, G41, G42

Purpose of milling cutter radius compensation (FRK)
The movements of the milling cutter when processing the programmed contour refer to the centre of the milling cutter. The contour really milled on the radius of the milling cutter therefore deviates from the programmed contour, i.e. with certain machining operations the programmed contour must be corrected by the milling cutter radius.
The function is handled by the control system during programming with milling cutter radius compensation (FRK).

Effect of milling cutter radius compensation
Without milling cutter radius compensation the milling cutter middle point M is the reference point for the control system. While the milling tool is traversing, this point is always on the programmed contour.
As a result, every milling cutter movement along the milled contour exhibits a constant deviation from the programmed contour equal to the milling cutter radius.
The surface highlighted in the drawing above is machined although it is outside the programmed contour.

without FRK



M milling cutter middle point

S real cutting point of milling cutter

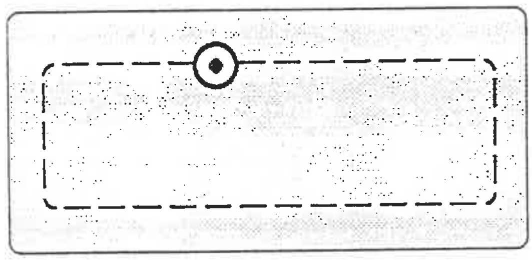
(shaded) deviation from the contour

— — programmed contour

—— milled contour

With milling cutter radius compensation, actual cutting point S always remains on the programmed contour, i.e. the middle-point path (equidistant) of the milling cutter is calculated so that the programmed contour is produced during milling.
The programmed and the produced contours are therefore identical.

with FRK



—— Actual and programmed contour of the keyway and thus the course of the cutter intersection point (S).

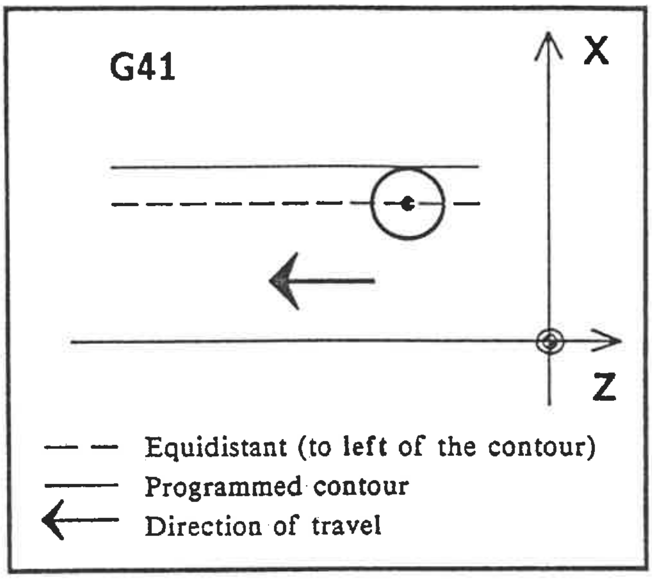
— — Cutter's middle-point path (equidistant) computed using the FRK.

Note
With FRK switched on the current position of milling cutter middle point M also appears in the position indicator; this means that these path positions are shown to which the programmed feed refers.

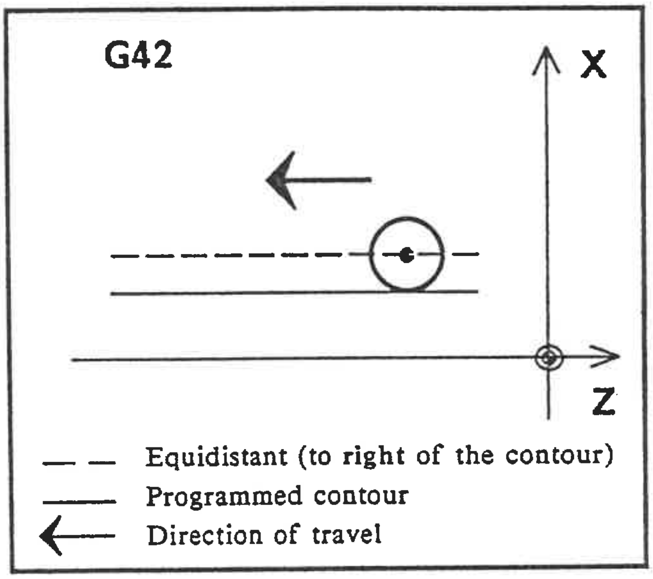
Milling cutter radius compensation G40, G41, G42

Activation of milling cutter radius compensation, G41, G42
Milling cutter radius compensation is activated with G41 or G42. From the programming of G41/G42 the control system also recognizes where the milling tool is moving.

G41 has to be programmed when, looking in the direction of travel (milling direction), the correction of the cutter radius is to be on the left of the contour, i.e. the middle-point path (equidistant) is on the left beside the actual, programmed contour.

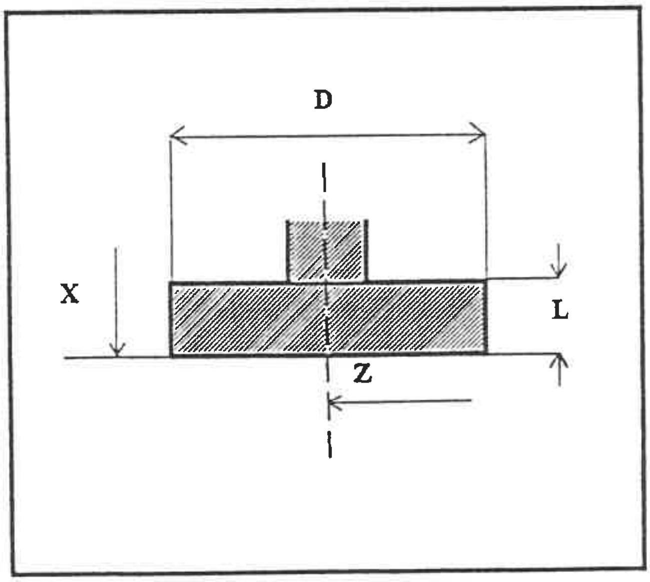


G42 has to be programmed when, looking in the direction of travel (milling direction), the correction of the cutter radius is to be on the **right** of the contour, i.e. the middle-point path (equidistant) is on the **right** beside the actual, programmed contour.



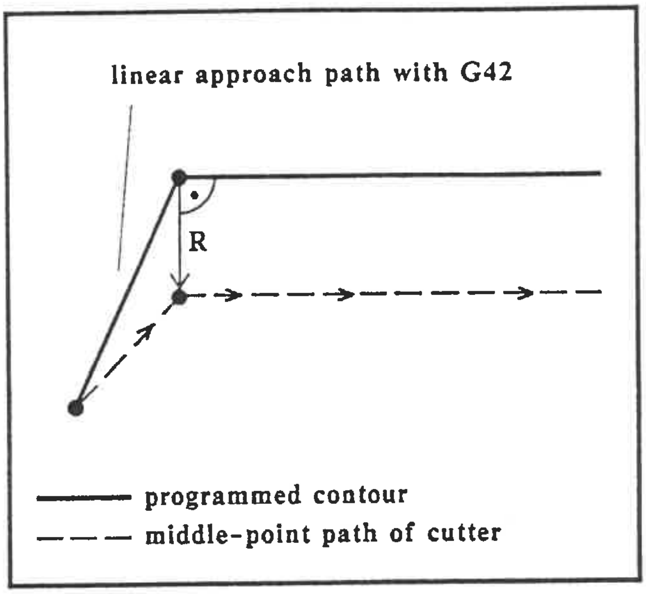
Definition of the milling tool
The milling cutter radius compensation only functions with the types of tools that are milling tools. All milling cutter types (face milling cutter, keyway milling cutter, straight-shank end mill and angle milling cutter) can be programmed with the tool types WT13, WT14, WT24 and WT0 (defined as milling tool).
The milling cutter radius is defined as a diameter dimension under address **D** in the tool file (N1001 to N1064).

The addresses **X**, **Z** and **L** are shown in the diagram to the right. It is not permissible to program any further addresses to set up a milling tool.

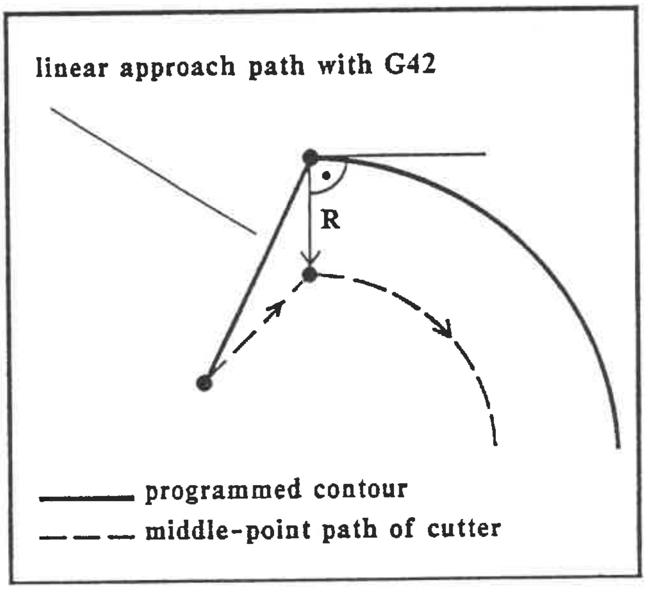


Milling cutter radius compensation G40, G41, G42

Selecting the FRK (milling cutter radius compensation) with G41/G42
In the record with G41 and/or G42 (selection record) it is only permissible to program a straight feed movement which runs parallel to the current machining level (frontface or circumference). Starting with the next NC record, the middle-point path (equidistant) of the cutter is calculated so that the programmed contour is produced during milling, i.e. the control establishes a vector the length of the milling cutter radius at the initial point of this record perpendicular to the programmed contour. While the contour is being machined, this vector is always moving perpendicular to the programmed contour and represents the middle-point path (equidistant) on which the cutter middle point moves (see illustration at top right).



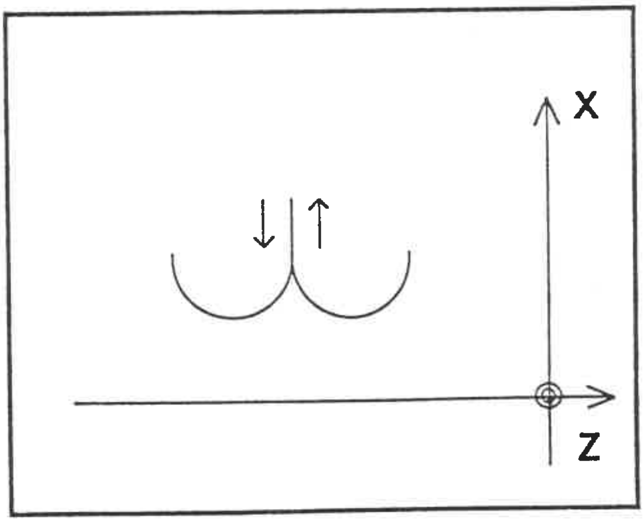
When a circular contour is involved, this vector is always perpendicular on the individual tangent to the circular contour (see centre illustration).



Attention
Under no circumstances is the FRK to be selected when the milling cutter is being fed vertically into the particular machining level (front face or circumference).
There is no allowance to change the machining plane while the FRK is active.

While the milling cutter radius compensation is active, no movements including a complete reversal of direction may be programmed.

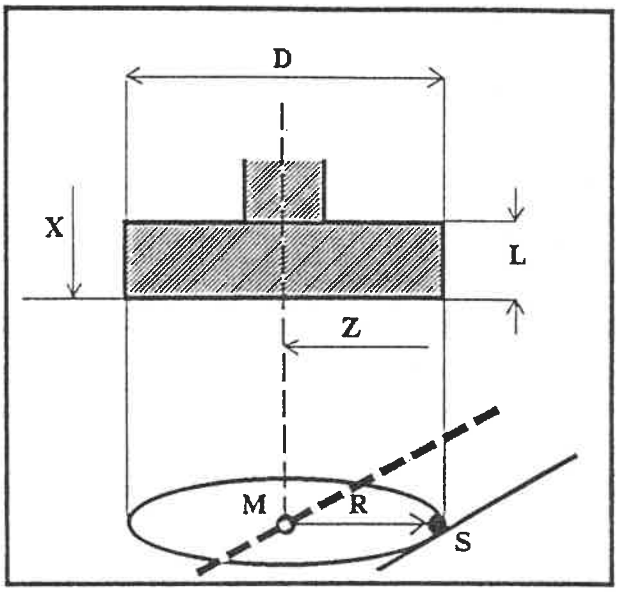
N...
N... G90 G111 Z... C-...
N... G91 G111 Z0.01 C0
N... G90 G111 Z... C+...



In addition, no zero point shifts (G152) of the C-axis are to be executed while the FRK is active.

Milling cutter radius compensation G40, G41, G42

While the miller cutter radius compensation is active, the **tool dimensions** (X, L, Z), including the milling cutter diameter (D), **may not be altered.**



Key to the diagram on the right:

M = middle point of milling cutter

R = radius of milling cutter

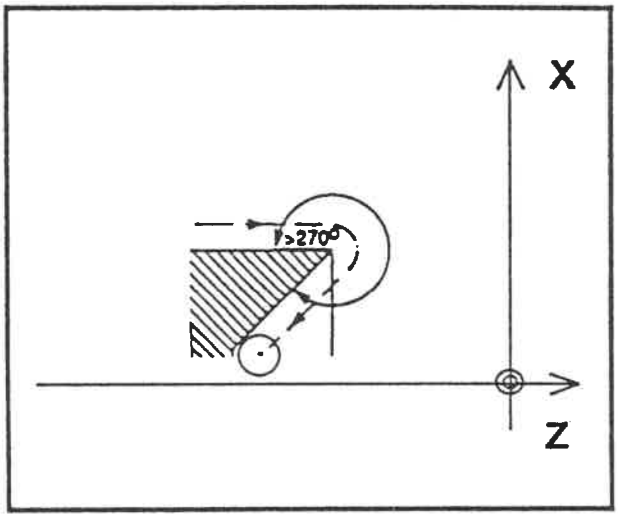
S = real cutting point

— — middle-point path of the milling cutter travelled with FRK

—— programmed contour

Tool path during milling cutter radius compensation

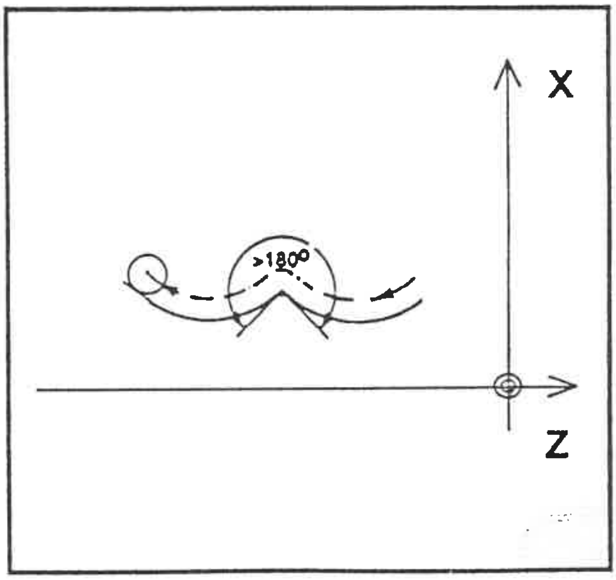
The path described by the centre point of the milling cutter is called an equidistant, i.e. a path where clearance b is the measure of the milling cutter radius from the contour. When two straight lines meet at an angle greater than 270°, the milling tool performs a "rolling" movement around the point of intersection, i.e. the centre-point path moves in an arc around the point of intersection.



The same applies for the intersection of a circular arc with a straight line or a circular arc with a circular arc. For angles between 180° and 270° the control system decides automatically whether the rolling movement is to be included.

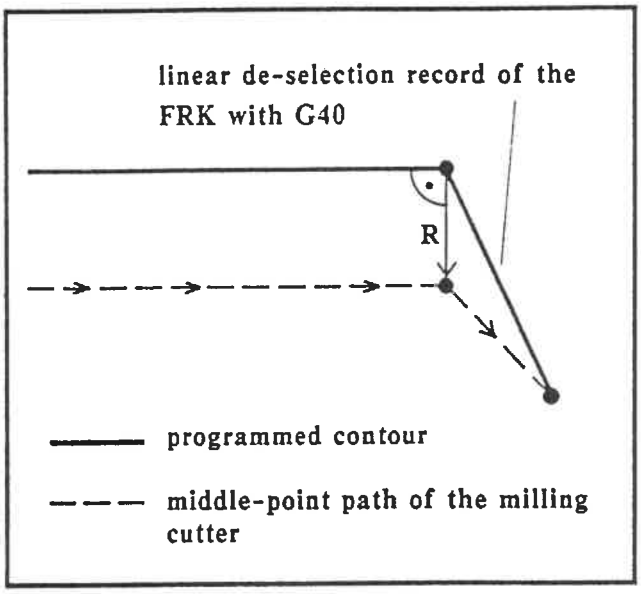
Note

If the end point of the following contour element is closer to the previous element than the measure forming the radius of the milling cutter, contour deviations may occur.



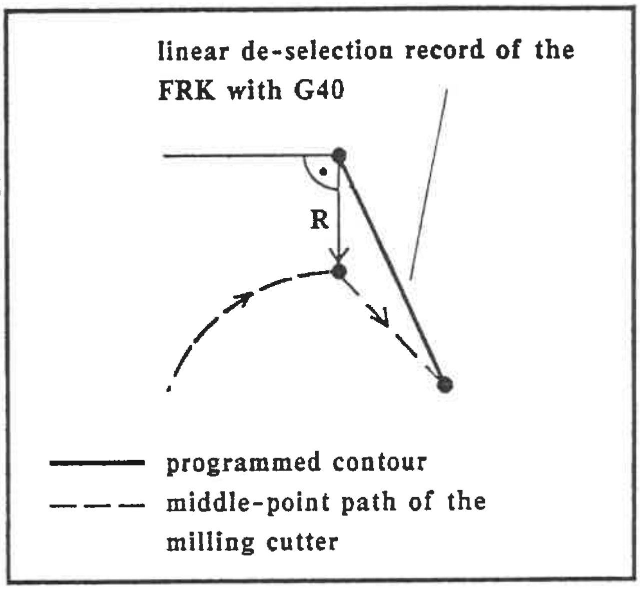
Milling cutter radius compensation G40, G41, G42

De-selection of FRK with G40
In the record with G40 (de-selection record) only a **straight line feed** movement is to be programmed; this is to run parallel to the individual machining surface (frontface or circumference).
The milling cutter radius compensation up to the record before G40 is factored in by the control. Starting with the NC record after G40, the middle-point path (equidistant) of the cutter again moves on the programmed contour so that the point of intersection of the miller deviates from the programmed contour by the radius of the cutter (see diagrams to the right).



Attention
Never deselect the FRK while the milling cutter is moving vertically back out of the particular machining level (frontface or circumference).

When changing the machining plane the FRK is to be deselected and after this once more reselected in the other machining plane.



Input
G40, G41 and G42 are entered in the EDITOR mode (see Chapter 4) with the softkey G-FUNCTION DIRECT or G-FUNCTION MENU.

Milling cutter radius compensation G40, G41, G42

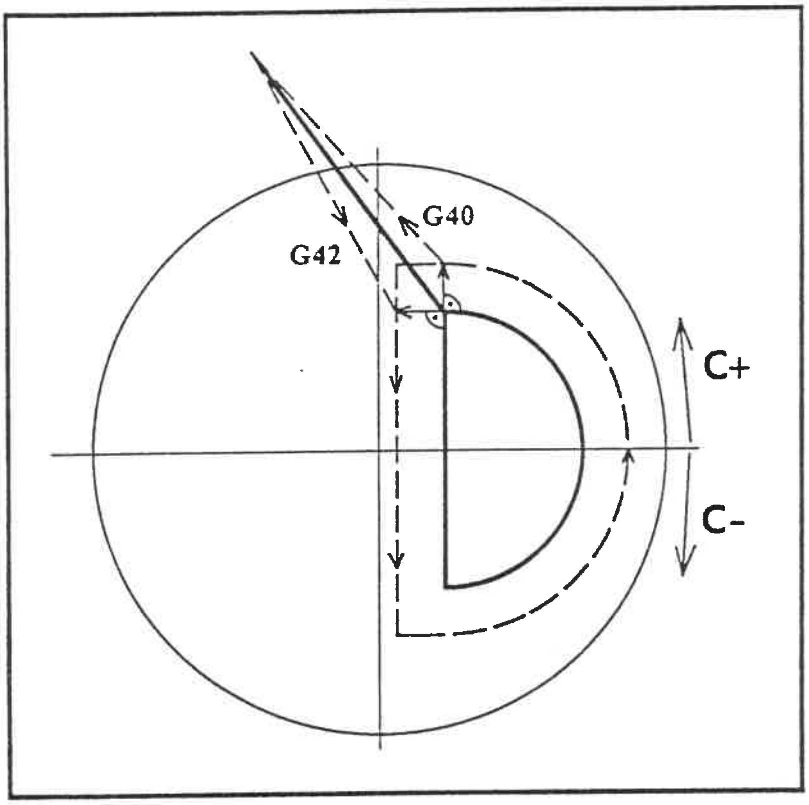
Program structure
A program part with milling on the frontface or circumference of the tool in conjunction with the C-axis and active cutter radius compensation generally has the following structure:

- Zero point shift with G59
- Tool definition WT13 (radial clamped tool for circumference surface), WT24 (axial clamped tool for front face) or possibly WT0 (special tool defined as milling tool) with milling diameter D(with function G92 under the address E)
- Pre-positioning with G0 X... Z...
- Swivelling in the C-axis with M14
- Entering technological data: speed (G197 S...), feed (G94 F... or G193 F...), cutting speed (G196 S...), coolant
- Speed limiting of auxiliary drive (G126)
- Zero point shift of the C-axis (G152)
- Pre-positioning with G100/G110
- Adjusting motion of the milling cutter to the workpiece (G1)
- Calling the FRK with G41 or G42 in conjunction with a straight line feed motion with G101 or G111
- Description of the contour with G100, G101, G102, G103 or G110, G111, G112, G113
- Liberating movement with G100/G110 and de-selection of FRK using G40
- Return motion of the milling cutter from the contour (G1)
- Swinging out C-axis (M15), switching off speed, switching off coolant
- Approaching the tool change position
- Program end or further machining

On the following pages this program structure is explained using a programming example for the front face as well as for the machining of the circumference with active milling cutter radius compensation.

Milling cutter radius compensation G40, G41, G42

Switching the FRK off and on, using a frontface machining as an example



%190955

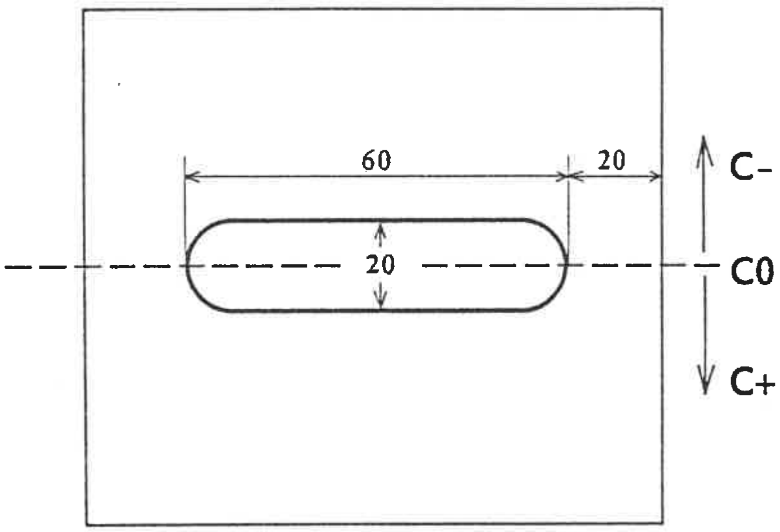
```
N 1   G59   X0   Z200
N 2   G92   X0   Z100  Q1   W24  E10  T10
N 3   G0    X140  Z5
N 4   M14
N 5   G197  G94  F100  S1000  M54  M8
N 6   G152  C0
N 7   G100  X140       C120
N 8   G1    Z-5
N 9   G101  G42  X63.246    C71.565          Selection record of the FRK
N10   G101  X63.246    C-71.565              FRK active
N11   G103  X63.246    I0    J30  C71.565    FRK active
N12   G101  G40  X140       C120             De-selection record of the FRK
N13   G1    Z5
...
...
```

Residue machining (machining off the remaining material, possibly with a different milling cutter tool, until only the convex semi-circle remains).

```
...
...
N...  M30
```

Milling cutter radius compensation G40, G41, G42

Switching the FRK off and on, using the milling of a circumference as an example



Diameter = 100 mm
Keyway depth = 5 mm
Circumference at base of keyway = 90 mm · π = 282.743 mm
1° corresponds to 0.785 mm
1 mm corresponds to 1.273°

%280455

```
N 1  G59  X0  Z200
N 2  G92  X40  Z70  Q1 W13  E8  T10
N 3  G0  X102  Z20
N 4  M14
N 5  G197  G94  F100  S1000 M54  M8
N 6  G152  C0
N 7  G110  Z-70  C0
N 8  G1  X90
N 9  G111  Z-30  C0
N10  G111  G41  Z-30  C-12.732              Selection record of the FRK
N11  G111  Z-70  C-12.732                   FRK active
N12  G113  Z-70  C12.732  J12.732  K0       FRK active
N13  G111  Z-30  C12.732                    FRK active
N14  G113  Z-30  C-12.732  J-12.732  K0     FRK active
N15  G111  G40  Z-50  C0                    De-selection record of the FRK
N16  G1  G94  X102  F1000
N17  M15  M55  M9
N18  G14
N19  M30
```

Milling cutter radius compensation G40, G41, G42

Note for frontface machining with FRK
When milling cutter radius compensation is active, no full turning may be programmed with G100. Likewise, two successive G100 commands are not permitted in which the second G100 command brings the cutter back to the starting point of the first G100 command; two such movement paths are also the equivalent of a complete turn. In such a case the movement must be performed in three partial turns:

Example
Let us assume that the point of origin of the following movement is the starting point N... G100 X10 C0

The following programming brings the cutter back to the point of origin:

N... G100 X10 C100
N... G100 X10 C200
N... G100 X10 C360

Note on the milling of a circumference with FRK
Adjustment motions of the milling cutter in the X-direction towards the circumference with active FRK result in a minimum C-axis travelling path component (C-direction) which is backed out in the next NC record.

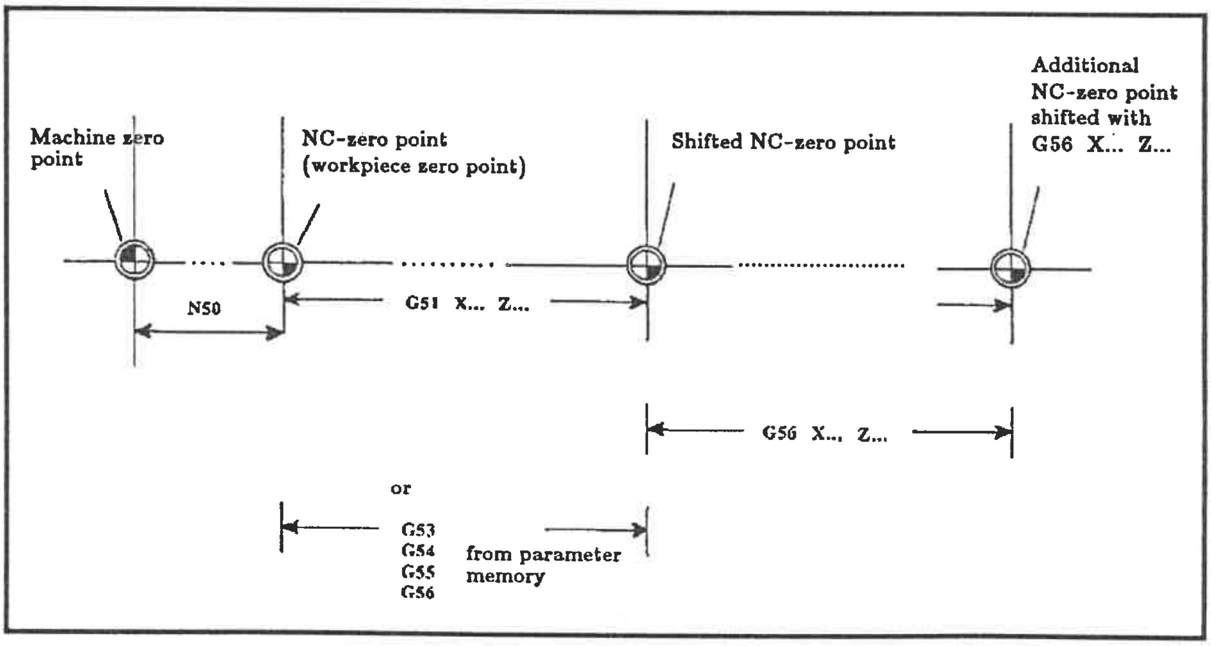
Stock allowance
G51 - G59

General

The control system provides the possibility of programming several zero point offsets. The stock allowances using G51 to G59 are described on the following pages. G51 is a programmable zero point stock allowance. Using G53, G54, G55 and G56, parameter-dependent zero point offsets can be called. G57 and G58 can be used to program a stock allowance (finishing measure) for cycles G81, G82 and G83.
Function G59 is a programmable zero point stock allowance. In addition, a **zero point offset with back-up tool** can be carried out in the setting-up mode. The procedure determines a new NC zero point.
Neither scratching or renewed dimensioning of the tools has to take place (see section 2).

Note

All programmed zero point offsets are cancelled using the functions M30 or M99 of the main program; i.e. the NC zero point programmed under parameter N50 is active.



Stock allowance
G51, G53

Programmable zero point allowance, G51
The control system adds the values stored under the addresses X and Z to the values of the NC zero point.

Required addresses
After selecting of G51, the following addresses are requested by the control system

DIAMETER	X:
LENGTH	Z:

Programming

N... G51 X10 Z20

The control system adds 10 to the value of the NC zero point in X-direction and 20 to the value in Z-direction. When the new zero point position (X10, Z20) is approached the display of actual X- and Z-values will show the value zero.
Functions G53, G54, G55 and G59 are cancelled as well as function G56, **provided that G56 was programmed <u>without</u> the addresses X and Z .**

Parameter-dependent zero point allowance, G53
The control system adds the X and Z values stored under parameter N53 to the NC zero point values.
The control system requests **no** addresses during input. Functions G51, G54, G55 and G59 are cancelled as well as function G56, **provided that G56 was programmed <u>without</u> the addresses X and Z .**

Example
N ... G53

The following was entered under parameter N53:
N53 STOCK ALLOWANCE 1 X20 Z10

The control system adds the offset values programmed under parameter N53 to the subsequent X and Z values in the program.
When the tool traverses to the new zero point position (X20, Z10), the display of actual X and Z will show the value zero.

Stock allowance
G54, G55, G56

G54:
Programming and meaning correspond to G53.
Functions G51, G53, G55 and G59 as well as function G56 are cancelled **provided that G56 was programmed** **without the addresses X and Z.**

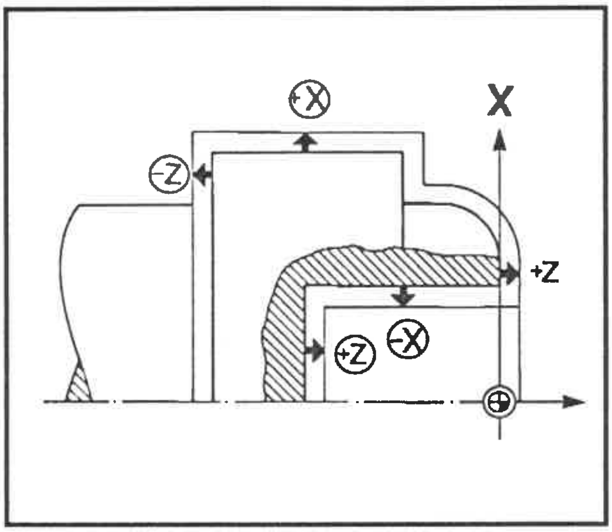
G55:
Programming and meaning correspond to G53.
Functions G51, G53, G54 and G59 are cancelled as well as function G56, **provided that G56 was programmed** **without the addresses X and Z.**

G56:
1. Programming and meaning correspond to G53.

Functions G51, G53, G54, G55 and G59 are cancelled.

2. When programmed with X or Z G56 takes over another function:
   The X and Z values are added to the currently active zero point stock allowance, i.e. G56 X... Z... acts additive as often as it is called up.
   This function can be of use during subprogram repeats when machining rods.

Stock allowance G57



Different stock allowances in X and Z, G57
Using G57 different stock allowances can be programmed in X and Z in all longitudinal, transverse and contour cycles.

Required addresses
After selecting G57 the following inputs are requested by the control system:

DIAMETER X:
Stock allowance at diameter, (diameter - input)

LENGTH Z:
Stock allowance in Z-direction

Note:
The correct sign must be used when entering the stock allowance value.

Programming
The block containing G57 must be positioned before the first cycle in which the stock allowance shall be active.
The stock allowance is active only in cycles G81, G82 and all contour turning cycles with G80.

Deleting stock allowance
If, later in the program, a cycle is to machine to finished dimensions, then G57 X0 Z0 has to be programmed.

G83 with G57
Stock allowance in cycle G83 can be programmed only for that direction for which an approach value was programmed as well.

Example:

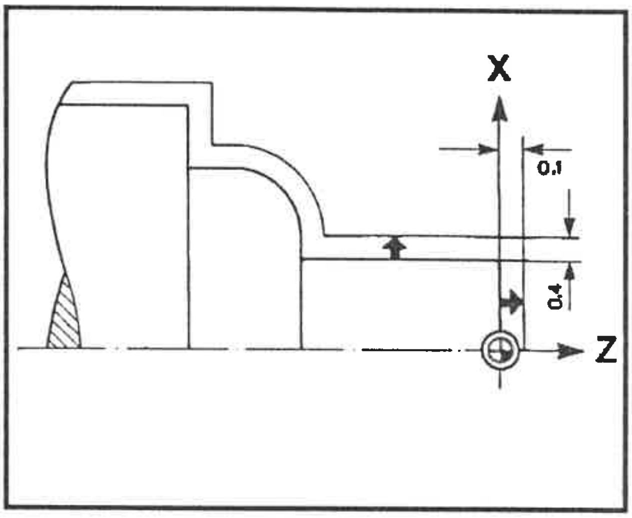
G83 I... (and no K)
Stock allowance possible only in X-direction.

G83 K... (and no I)
Stock allowance possible only in Z-direction.

G83 I... K...
Stock allowance possible in X- and in Z-direction.

Stock allowance G57

Example
Outside contour with stock allowance



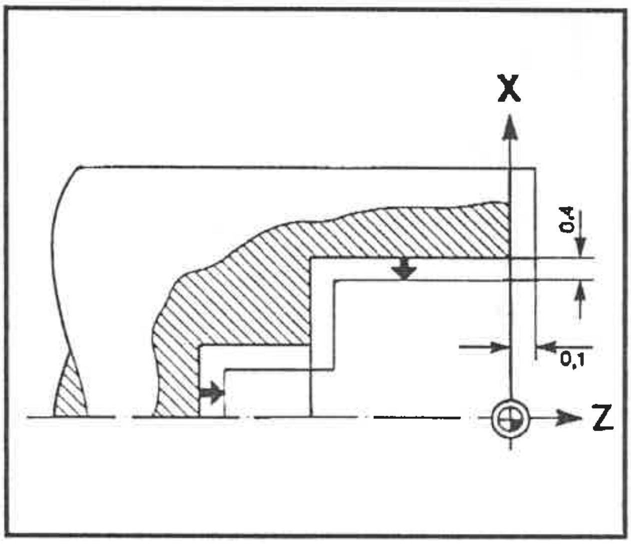
Programming

...
...
N... G57 X0.8 Z0.1
N... G83 ...
...
...

Explanation
A stock allowance of 0.4 mm is left at the radius, i.e. the diameter increases by 0.8 mm. The direction in which the stock allowance remains is X+.

The stock allowance in Z is 0.1 mm.

Example
Inside contour with stock allowance



Programming

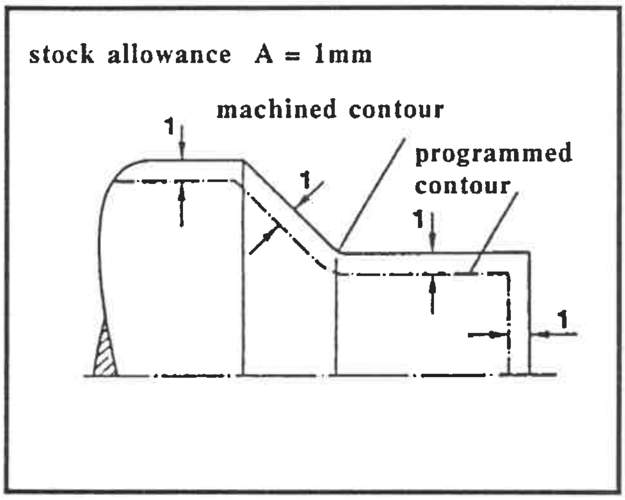
...
...
N... G57 X-0.8 Z0.1
N... G83 ...

Explanation
0.4 mm stock allowance is programmed at the radius. The diameter thus decreases by 0.8 mm. The direction in which the stock allowance remains is X-.

Stock allowance
G58

Stock allowance parallel to contour in X and Z, G58
For machining cycles G81, G82 and G83, G58 can be used to program a stock allowance parallel to the contour (equidistant) in X and Z.

Required addresses
After selecting G58, the control system requests the following inputs:



STOCK ALLOWANCE A:

Programming
When programming G58, tool nose radius compensation (see G40 - G42) must be active, as the control system will otherwise not include the stock allowance. As long as SRK is active, the stock allowance cannot be altered.

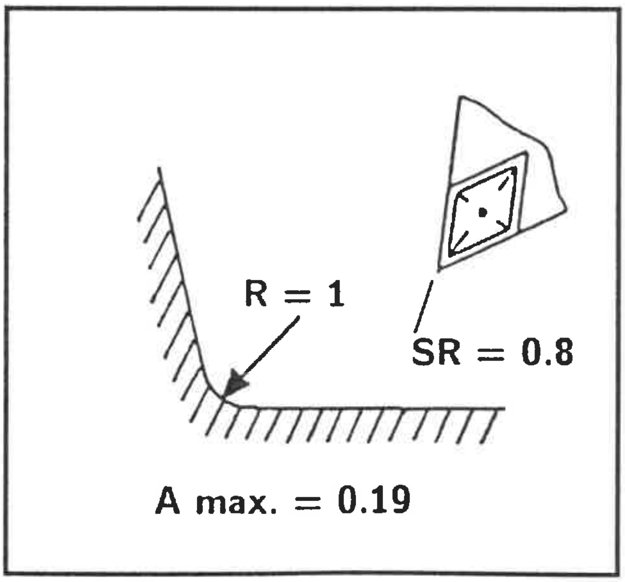
Example:
N... G41 G58 A1

The control system leaves a stock allowance of 1 mm along the contour.

Deleting stock allowance
If, later in the program, a cycle is to machine to finished dimensions, then
G58 A0 must be programmed.

Note
It is not expedient to program a stock allowance value larger than the value of the smallest inside radius which is to be machined:

Tool nose rad. + stock allow. < inside rad.



If a **negative** value is programmed under address A, then a turned part with **"negative stock allowance" (smaller part)** is being machined.

Stock allowance
G59

Programmable zero point shift, G59
The zero point shift programmed with G59 replaces the shifting value entered under NC zero point (parameter N 50). It is active until the program ends.

This cancels functions G51, G53, G54 G55 and G59 as well as function G56, **provided G56 was programmed <u>without</u> the addresses X and Z .**

Required addresses

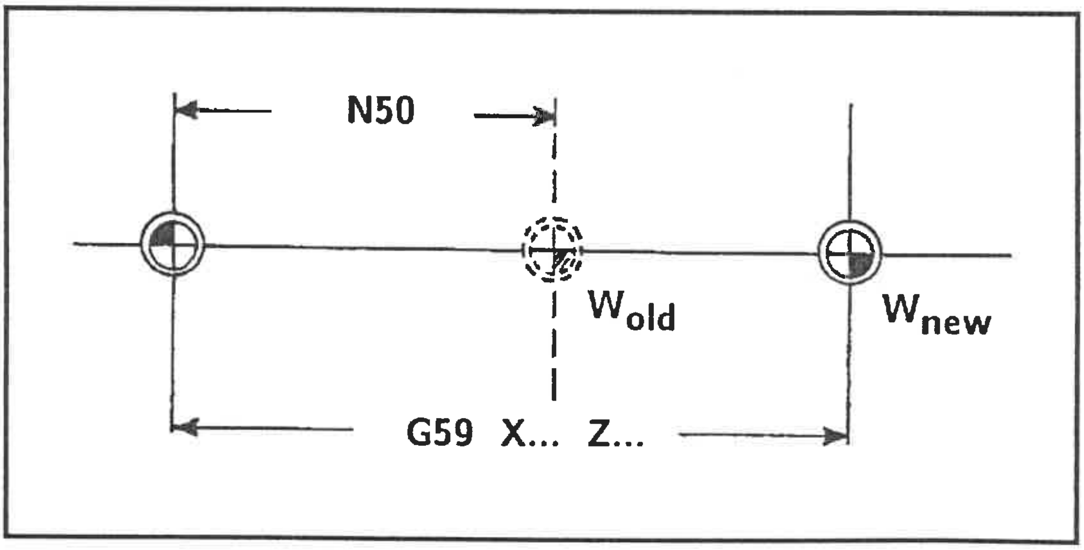
After selecting G59, the control systems requests the following inputs:

DIAMETER X:

LENGTH Z:

Programming

N20 G59 X120 Z180



Cancel protective zones G60

Protective zones function, G60
By programming function G60 the effect of protective zones programmed under **PARAMETER N201** can be **cancelled.**

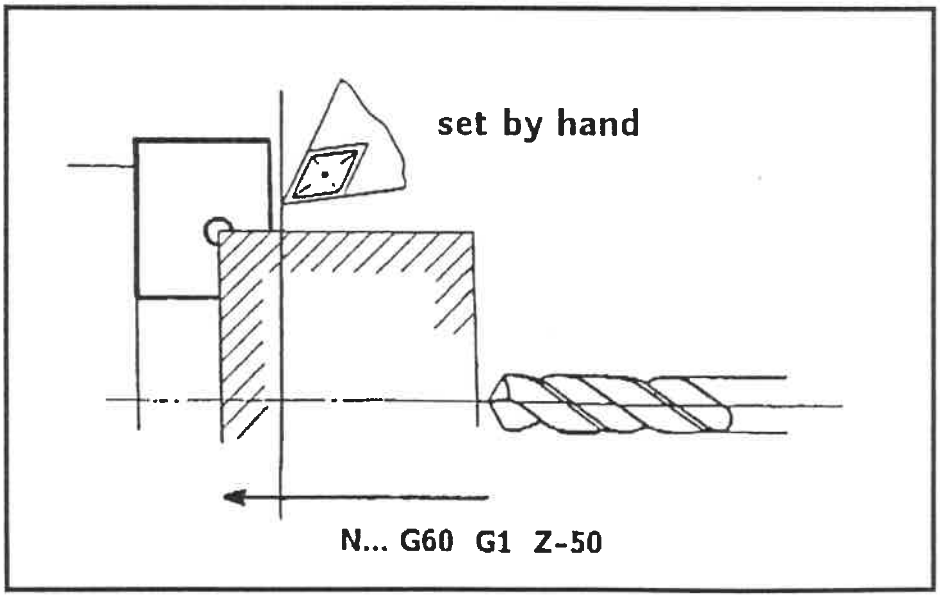
Required addresses
No addresses are requested by the control system.

Programming
Function G60 is programmed in that block in which the protective zones shall no longer be active.
Function G60 is active only in that block in which it is programmed.
If the effect of protective zones is to be cancelled for several blocks, then G60 must be programmed in each of these blocks.

Attention

When using function G60 the operator must check very carefully the programmed traverse movement path.

Danger of collision!



Unconditional branching G61

Jump function, G61
Using jump function G61, conditional and unconditional branchings can be programmed.

Required addresses
After selecting path condition G61 the control system requests the following inputs:

DISTRIBUTION OF JUMPS	H:{
BLOCK NUMBER	N:
BLOCK NUMBER	N:
BLOCK NUMBER	N:

Unconditional branching

Programming:
N16 G61 N25

When block N16 has been executed, the control system will branch to block N25 and continues execution from there. Block N17 to N24 are not executed.

Conditional branching G61

Conditional branching
As opposed to the unconditional branching, this jump can be made dependent on a condition (value of a variable or of an external event). The control system jumps to the agreed block only when the condition is fulfilled and continues the program from there.
The branch condition is entered in address H in the block containing G61, whereby the following two possibilities are provided.

1. possibility: jump depending on sign

N 12 G61 H{V11 + 17} N15 N17 N25

Explanation

G61	jump function
H{V11 + 17}	jump condition (variable) for branching
N15	block number to jump to if **H < 0**
N17	block number to jump to if **H = 0**
N25	block number to jump to if **H > 0**

2. possibility: jump with branch condition

Example 1:

N12 G61 H{V7 > 100} N20
N13...

If variable V7 is greater than 100, the control system jumps to block N20 after executing N12.
If V7 is less than or equal to 100, program execution continues at N13.

External events
G61

Jumps by means of external events
With the ELTROPILOT L - control system program execution can be influenced by means of external events which are determined using the SPS (PLC machine adaption).
The external events are requested in the parts program using the function G61.

The following structure must be used for this:

$1 N... G61 H{ E... [...] ==1} N...

number of the block N (destination of branching)

1 = set
0 = not set

bit 1 to bit 16

external event
(E1 for slide 1)

The table on the following page gives a survey of the function of the different bits in the external event E1. At this point events E2 to E5 have no meaning.

Intermittent feed
G64

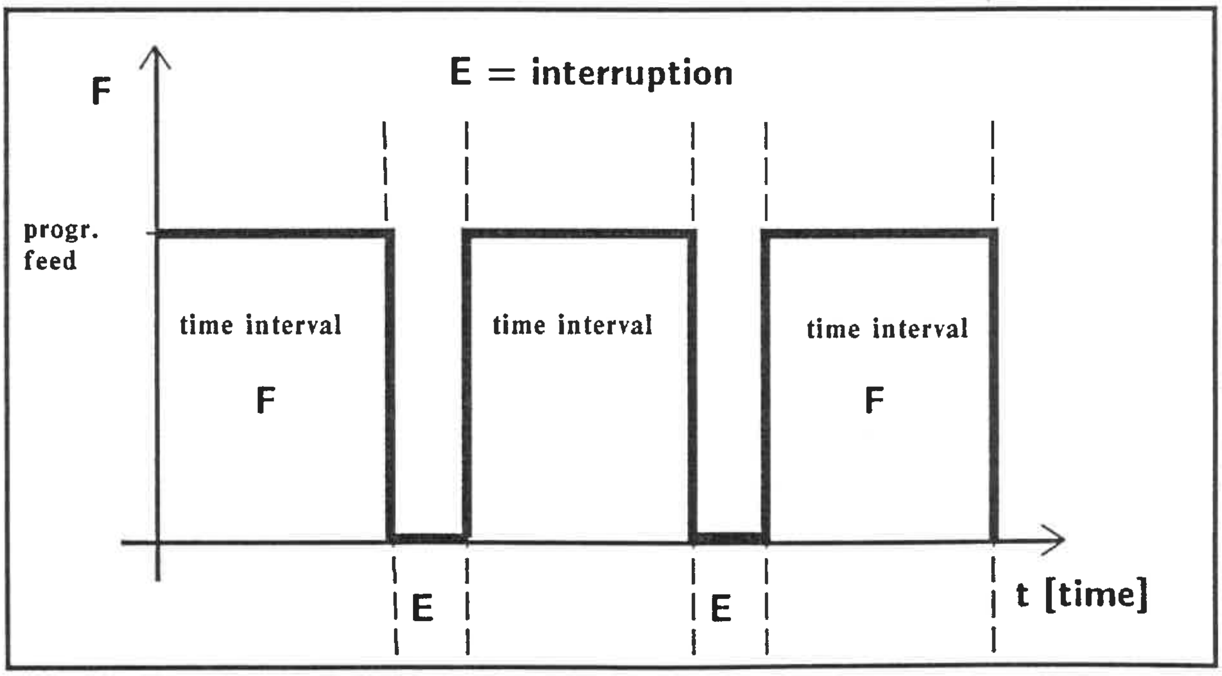
Intermittent feed, G64
By programming the function G64 the programmed tool feed movement can be interrupted momentarily (intermittent feed), i.e. feed is reduced to zero in order to ensure a continuing chip break, in particular for materials which tend to form continous chip.

The duration of the feed interruption and the length of the time intervals when programmed feed is executed can be determined by programming the two address parameters of the function G64 correspondingly.

Required addresses
After selecting function G64 the control system requests the following inputs:

DURATION OF INTERRUPTION E:
The duration of feed interruption in seconds is entered under the address E. Interruptions between 0 and 99.9 seconds may be programmed
(0s < E < 99.9s).

DURATION OF FEED F:
The duration of the time interval in seconds in which programmed feed is executed between two feed interruptions. Here, time intervals between 0 and 99.9 seconds may be programmed
(0s < F < 99.9s).



G64

Switching the intermittent feed on and off
The intermittent feed can be switched both on and off by programming the function G64.

In order to switch on the intermittent feed, the function G64 must be programmed in the parts program entering both time values under the addresses E and F. The function G64 is then stored and active even if the program branches into a subprogram.

In order to switch off the intermittent feed, the function G64 must be programmed either without entering the addresses E and F (confirmation of E and F) or by entering 0 for at least one of the two time values.

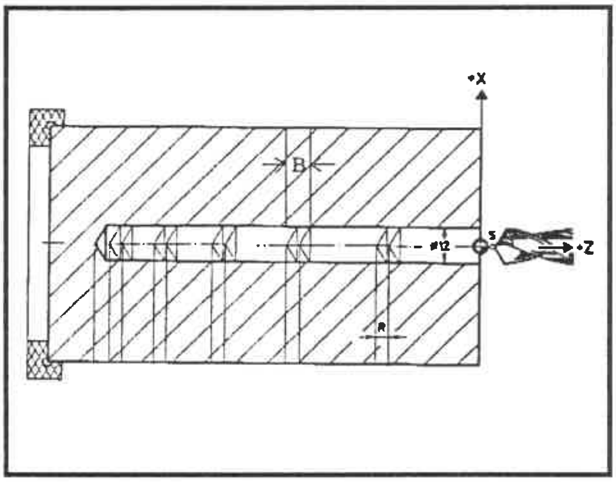
Note
If program parts in which intermittent feed is to be executed include rapid traverse movements (G0) or thread cycles (G31, G32, G33 and G35), then the programmed intermittent feed does not effect these functions.

Deep-hole drilling cycle G74

Deep-hole drilling cycle, G74
By programming function G74, it is possible to perform either boring on the axis of rotation of the workpiece using a fixed boring bar or boring at any angle using driven tools. If driven tools are used, then the following values must be programmed before activating the cycle:
- feedrate in mm/min, using G94,
- speed, using G197,
- direction of rotation of the auxiliary drive, using the corresponding M-functions.

Before that, however, the spindle must be provided with an index. This must be switched off after the boring has finished.

Required addresses
After selection of G74, the control system requests the following inputs:



END POINT X:
Coordinate of the end point of the bore (input of diameter).

END POINT Z:
Coordinate of the end point of the bore

1st DEPTH OF BORE P:
Cutting depth for the first boring cut

SAFETY DISTANCE R:
Under this address the distance is determined at which the traverse speed changes from rapid traverse to feed speed before the start of boring.

REDUCTION VALUE A:
Hereby the value is determined by which the control system automatically reduces the cutting length when moving from one cut to the next.

DISTANCE OF RETREAT B:
Under this address the distance is determined by which the borer is retracted within the bore hole after each cut in order to break chips.

MINIMUM DEPTH OF BORE W:
A minimum value for the cutting length can be entered under this address. The reduction value will not fall below this value.

G74

PERIOD OF DWELL E:
This function can be used to determine the period of dwell (in seconds) for cutting free after each separate cut.

Notes
Tool types **WT0** or **WT10** must be selected for this drilling cycle.

Depending on the programming of address B **two different versions** of this drilling cycle can be activated.

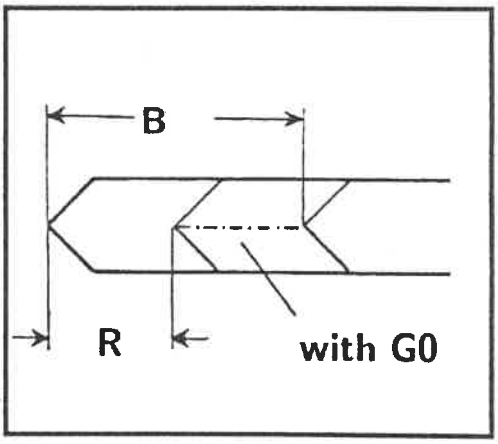
If address **B** is **not** programmed, i.e. only confirmed, then a **complete** retreat from the bore hole takes place after **each** cut in order to remove chips from the bore hole.

If, however, a value is programmed, then the borer is retracted by the programmed value of the distance of retreat B between the single cutting steps. During this retraction the borer remains **within the bore hole.**

Assumption
When calculating the programmed address the control system assumes that the borer tip is positioned at safety distance from the surface to be machined.

Cycle procedure
The borer traverses at feed speed from the starting point to the surface to be machined and performs the first cut, using the programmed cutting depth **P**.
Then the borer is retracted at rapid traverse speed either completely or by the distance value programmed for distance of retreat **B**.
In case of a complete retreat the next approach to safety distance **R** will be in rapid traverse and after that at feed speed.
The next cutting depths are reduced by the corresponding reduction values until the minimum depth of bore W is reached.

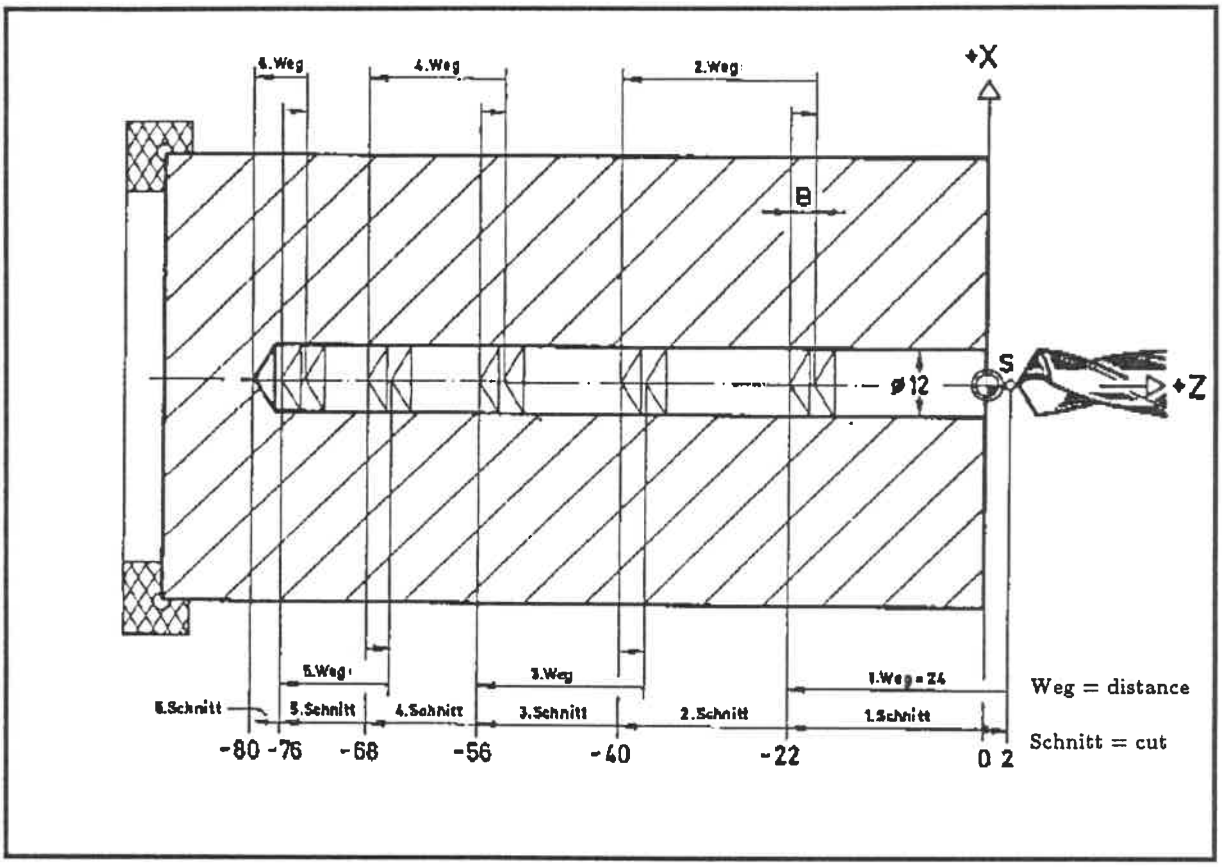
If the distance of retreat is **larger** than the safety distance, then the difference of paths between distance of retreat and safety distance will be traversed rapidly.



G74

Example 1
Boring with retreat within the bore hole
(breaking chips)

Programming with B



N1 G97 F0.1 S800 T5 M3 M7
N2 G0 X0 Z2
N3 G74 X0 Z-80 P22 R2 A4 B2 E1
N4 G14 Q2 M30

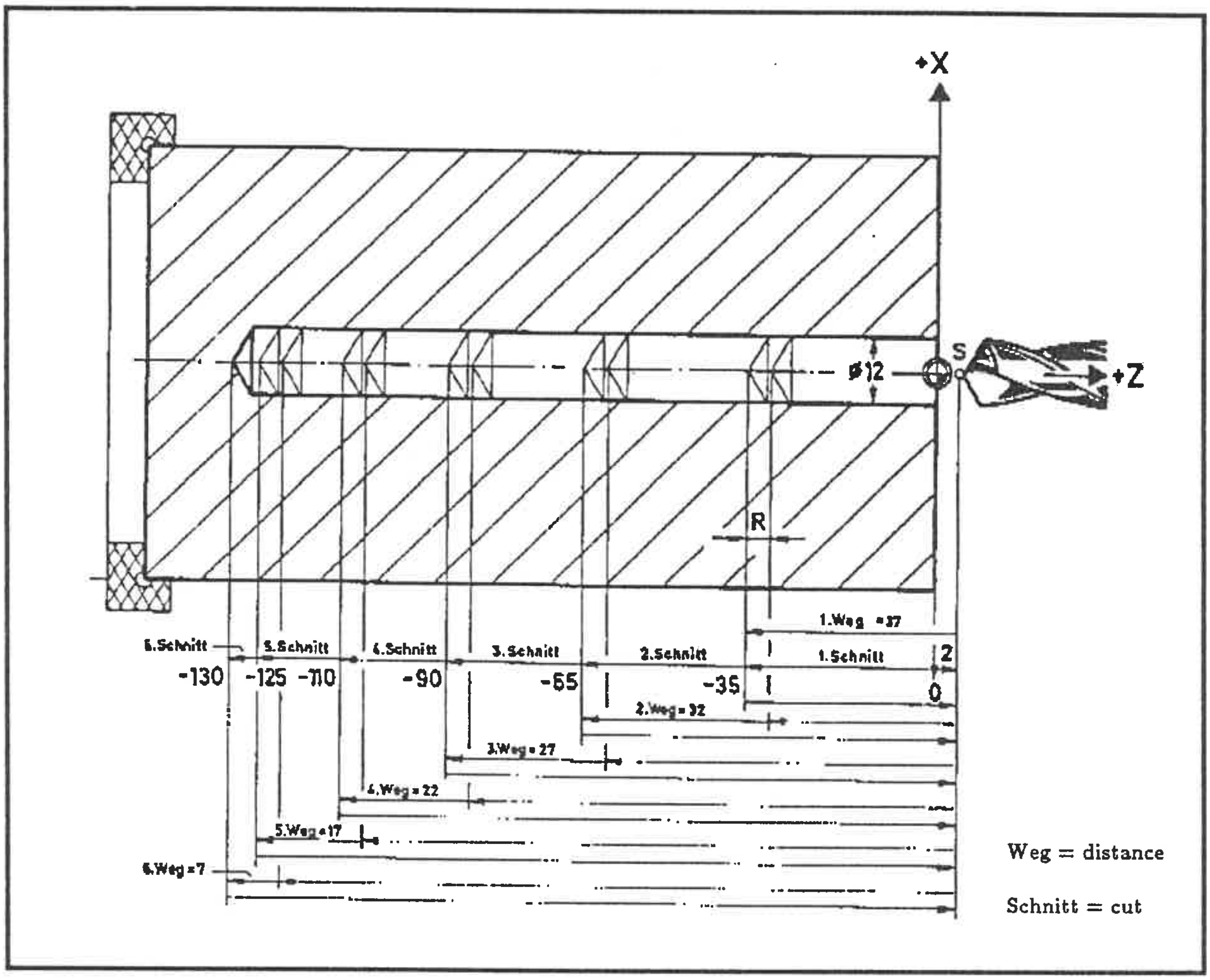
Explanation

N1	Starting conditions
N2	In rapid traverse to starting point
N3	Activating drilling cycle, first depth of bore P = 22mm, safety distance R = 2mm, reduction value for each new cut A = 4mm, distance of retreat B = 2mm and period of dwell of one second for cutting free.
N4	Traverse to tool change point, end of program.

G74

Example 2
Boring with complete retreat from the bore hole (removing of chips)

Programming without B



N1 G97 F0.1 S800 T5 M3 M7
N2 G0 X0 Z2
N3 G74 X0 Z-130 P35 R2 A5 E1
N4 G14 Q2 M30

Explanation

N1 Starting conditions

N2 Rapid traverse to starting point

N3 Activating of drilling cycle, first depth of bore P = 35mm, safety distance R = 2mm, reduction value for each new cut A = 5mm and period of dwell of one second for cutting free.

N4 Traverse to tool change point, end of program.

Front face hole circle
G77

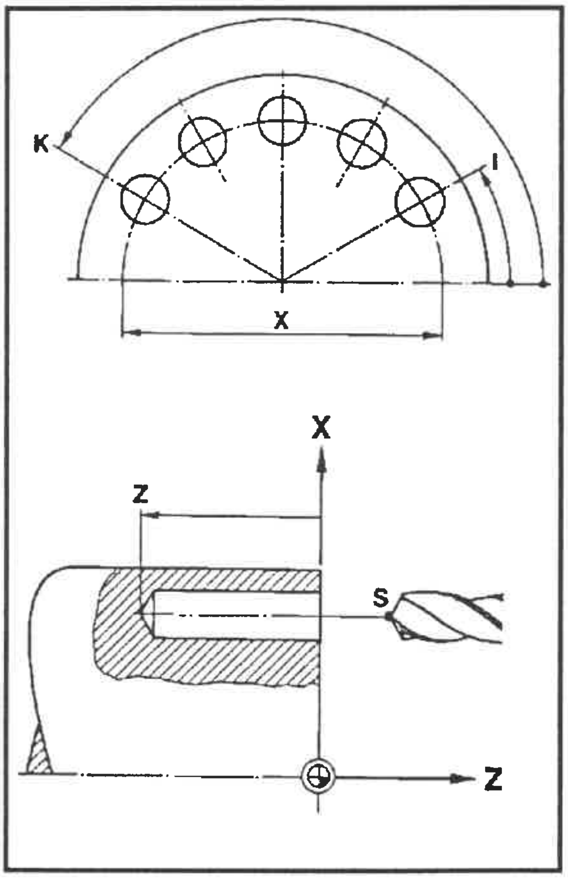
Front face hole circle, G77
By programming G77, a driven tool can beprogrammed to produce circular drilled holes on the front face of the workpiece. In accordance with the machine equipment this machining can be realized with the C-axis or with the position-controlled main spindle. During the development of the NC-program the control system perceives which sort of machining has been selected. In the case of existing C-axis, the function M14 (swivelling-in C-axis) has to be programmed before activating the cycle. After activating the cycle, the C-axis can be swivelled out with function M15.

Before activating the cycle, feed in mm/min must be programmed with G94 and the speed of the auxiliary drives with G197. The direction of rotation of the auxiliary drive is determined under the address J. Furthermore the tool must traverse to a suitable starting point S, located in X on the hole circle diameter.

Required addresses
After selecting G77, the control system requests the following inputs:

LENGTH	Z:
STARTING ANGLE	I:
END ANGLE	K:
NUMBER OF BORINGS	Q:
TOOL ROT. DIRECTION CW = 1 CCW = 2	J:

1 = clockwise 2 = counter-clockwise



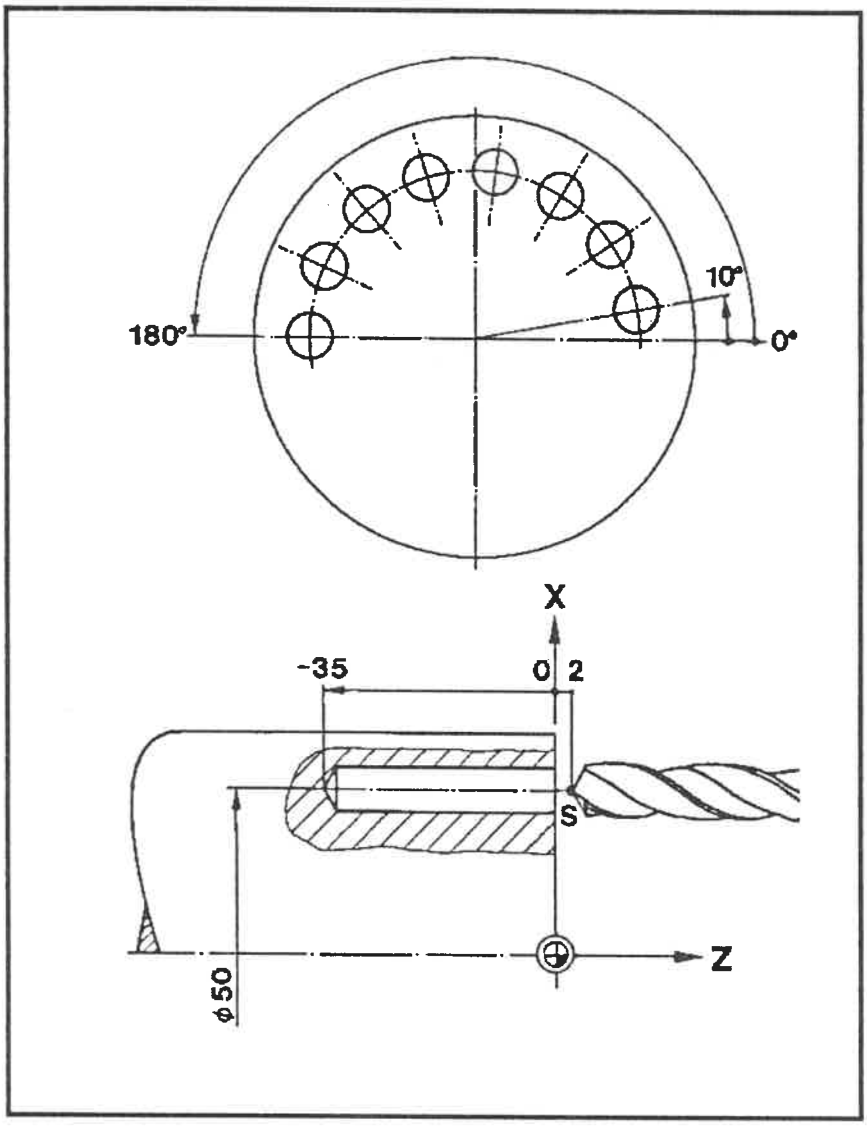
Notes
Tool type WT 10 must be selected for this drilling cycle.
If address I is merely confirmed, the control system gives I the value zero. If K is not entered, but more than one boring has been selected, the control system assumes a **full circle division.**

Cycle procedure
The spindle is positioned to an angle in accordance with the data in addresses I, K and Q. The control system procreates automatically in accordance with the machine equipment the required NC-functions (positioning, driven tool, indexing and traverse movements). At feed speed to target coordinate Z, thus producing the first bore. Rapid traverse back to starting point. This procedure is repeated the number of times given under Q.

G77

Programming

N 1 G0 G94 G197 X50 Z2 F80 S800 T1
N 2 G77 Z-35 I10 K180 Q8 J1



Explanation

N 1 Rapid traverse to starting point. X-value lies on hole circle diameter. Feed G94 [mm/min], speed for driven tool G197 (S = 800).

N 2 Activating cycle. Depth of bore 35 mm. Starting angle of the first boring is 10°. End angle of the last boring is 180°. 8 holes are drilled, borer drills in clockwise direction, (J=1).

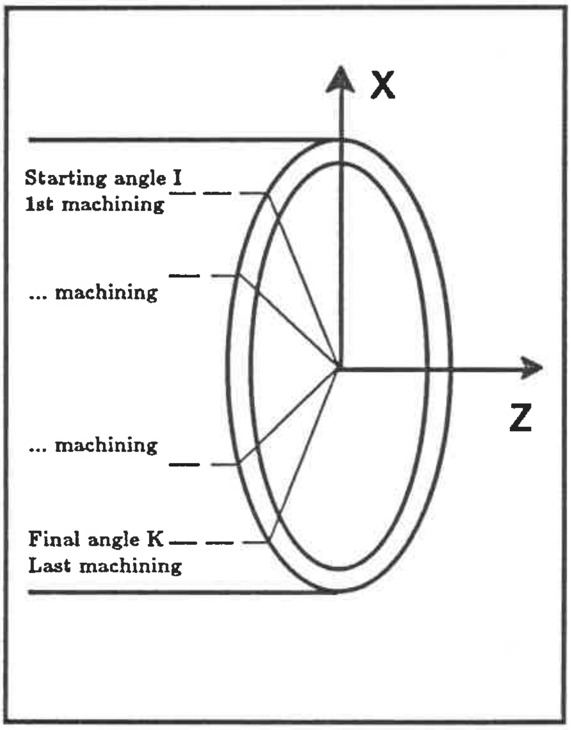
Angle circle cycle G770

Angle circle cycle, G770
Several machining procedures can be carried out simultaneously on the front surface or circumference of a workpiece by means of the driven tools. This can be done on an angle circle by programming the function G770.

Before each machining the spindle is turned to an angle between starting angle and final angle and the spindle is automatically indexed by the control system.
The desired contour can be programmed by means of a subprogram or by means of the following blocks, it has to be terminated with G80.
For this, the tool has to be moved to a suitable starting point, which is on the starting position of the contour in X direction.

Required addresses
After selecting G770, the control system requests the following inputs:

STARTING ANGLE	I:
FINAL ANGLE	K:
NUMBER OF MACHININGS	Q:



Type of machining
If no C-axis is swivelled in, the control system assumes that it is a machining process (e.g. thread boring, deep hole boring with G74) without C-axis operation, and positions the main spindle.
If, however, the C-axis is swivelled in via the parts program, positioning is achieved with greater precision and several similar milling processes can be carried out on the front face or the circumference of the workpiece.

Notes
If address I is confirmed, the control system assigns to I the value zero. If no K value is given for more than one machining process, the control system assumes a complete circle partition.

Cycle procedure
Spindle positioning at an angle according to the entered addresses I, K and Q.
Machining of the contour and repetition according to the value entered under Q.

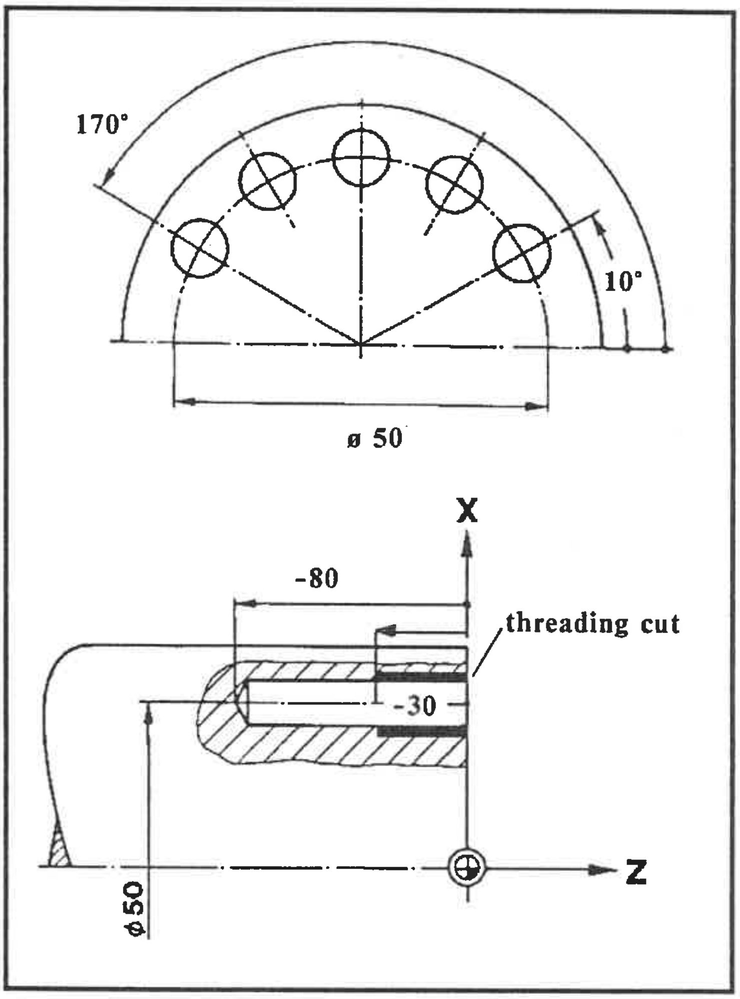
G770

Example for programming the angle circle cycle
First, five borings are machined by means of the drilling deep cycle and then a thread is cut.
The drilling deep cycle is called up by means of a sub program while the threading cut is entered directly.

Programming

%1909 (main program)

```
...
N10  T2
N11  G0  X50  Z3
N12  G770  I10  K170  Q5
N13  L11
N14  G80
N15  G0  X100  Z200  M55
N16  T6
N17  G197  G94  F40  S400  M53
N18  G0  X50  Z2
N19  G770  I10  K170  Q5
N20  G1 Z-30  M53
N21  G1  Z2  M54
N22  G80
N23  G0  X100  Z200  M55
N24  M30
```



%11 (sub program)

```
N1   G197  G94  F80  S800  M53  M08
N2   G74  X50  Z-80 P22  R2  A4  E1
N3   M30
```

G770

Explanation:

N10 Call tool for the drilling deep cycle with G74
(tool type WT10 must be used for this)

N11 Positioning with rapid traverse

N12 Call angle circle cycle (starting angle 10°, final angle 170°, five borings;
i.e. the sub program L11 is executed five times consecutively)

N13 Call sub program (%11)
- N1 Speed of driven tool, feed per minute, direction of rotation of driven tool, coolant
- N2 Call drilling deep cycle, first boring depth P = 22mm, safety distance R = 2mm, reduction value per new cut A = 4mm and period of dwell for cutting free one second
- N3 End of sub program

N14 End of angle circle cycle

N15 Rapid traverse away from workpiece, switch off driven tool

N16 Change in thread cutter

N17 Feed and speed of the driven tool (thread cutter), clockwise direction

N18 Positioning with rapid traverse

N19 Call angle circle cycle (starting angle 10°, final angle 170°, five threading cuts;
i.e. the following contour is executed five times consecutively)

N20 Threading cut to Z-30, clockwise direction of rotation of driven tool

N21 Retreat within the thread, counter-clockwise direction of rotation of driven tools

N22 End of cycle

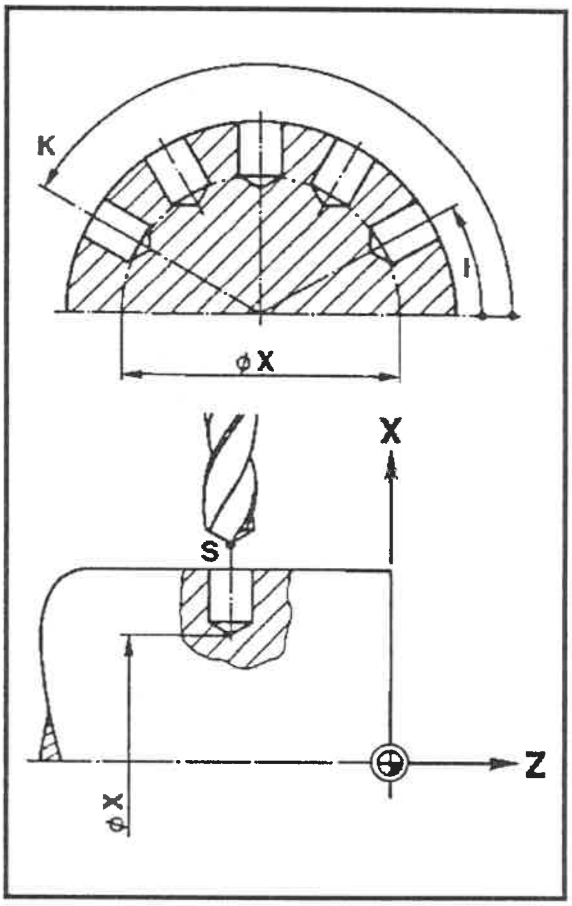
N23 Rapid traverse away from workpiece, switch off driven tool

N24 End of program

Circumference hole circle G78

Circumference hole circle, G78
By programming G78, a driven tool can be programmed to produce circular drilled holes on the circumference of the workpiece. In accordance with the machine equipment this machining can be realized with the C-axis or with the position-controlled main spindle. During the development of the NC-program the control system perceives which sort of machining has been selected. In the case of existing C-axis, the function M14 (swivelling-in C-axis) has to be programmed before activating the cycle. After activating the cycle, the C-axis can be swivelled out with function M15.

Before activating the cycle, feed in mm/min must be programmed with G94 and the speed of the auxiliary drives with G197. The direction of rotation of the auxiliary drive is programmed under address J. Furthermore, the tool must traverse to a suitable starting point S, located in Z
on the hole circle diameter.



Required addresses
After selecting G78, the control system requests the following inputs:

DIAMETER	X:
STARTING ANGLE	I:
END ANGLE	K:
NUMBER OF BORES	Q:
TOOL ROT. DIRECTION CW = 1 CCW = 2	J:

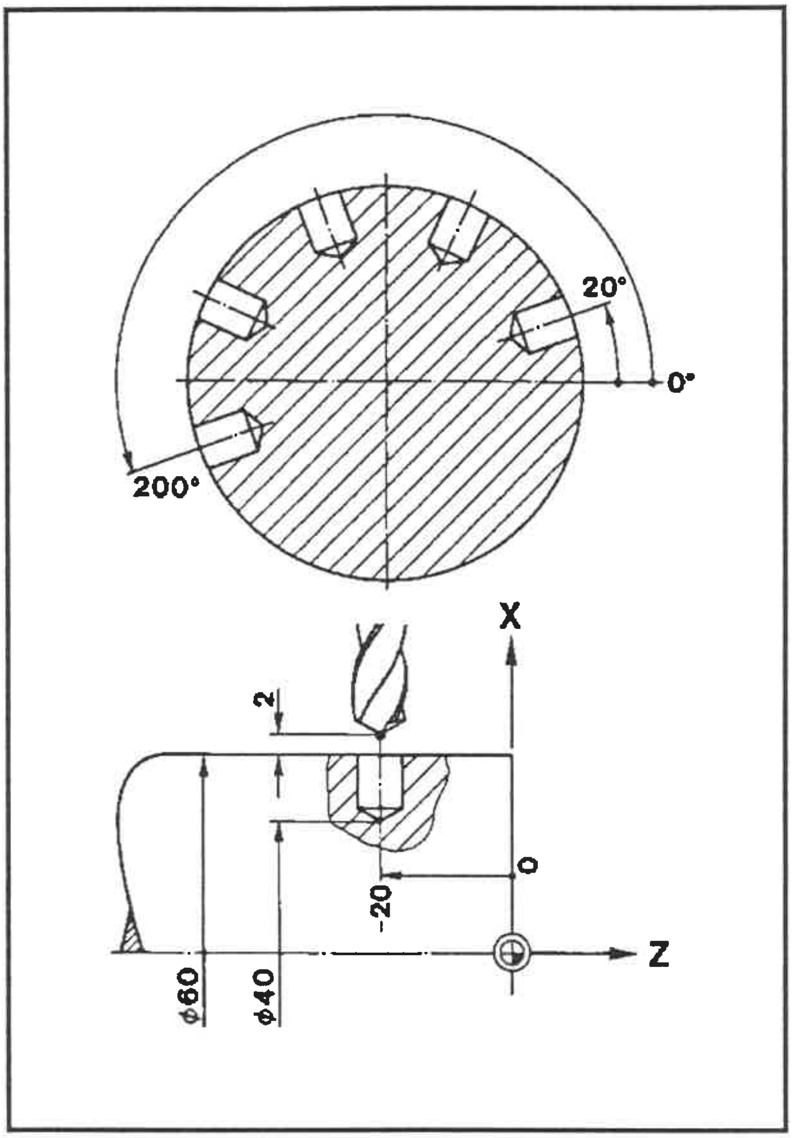
1 = clockwise   2 = counter-clockwise

Notes
Tool type WT 10 must be selected for this drilling cycle.
If address I is merely confirmed, the control system gives I the value zero. If K is not entered, but more than one boring has been selected, the control system assumes a **full circle division.**

Cycle procedure
The spindle is positioned to an angle in accordance with the data in addresses I, K and Q. The control system procreates automatically in accordance with the machine equipment the required NC-functions (positioning, driven tool, indexing and traverse movements).
Traverse at feed speed to destination coordinate X, i.e. drill first hole.
Rapid traverse back to starting point.
This procedure is repeated the number of times given under Q.

G78

Programming
N 1 G0 G94 G197 X62 Z-20 S800
N 2 G78 X40 I20 K200 Q5 J1



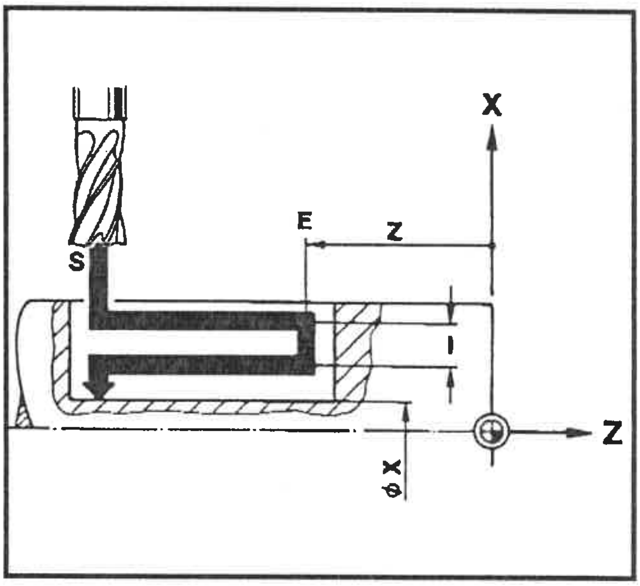
Explanation

N 1 Rapid traverse to starting point. Z-value lies on the diameter of the hole circle. Feed G94 [mm/min], speed for driven tool G197 (S = 800).

N 2 Activating cycle. Drilling depth 10 mm. Starting angle of the first boring is 20°. End angle of the last boring is 200°. Five holes are drilled, drilling in clockwise direction (J=1).

Milling keyways
G79

Milling keyways on circumference, G79
By programming of G79 a driven tool can be used for milling keyways parallel to the Z-axis on the circumference of a workpiece. The width of the keyway depends on the diameter of the cutter. In accordance with the machine equipment this machining can be realized with the C-axis or with the position-controlled main spindle (M19). During the development of the NC-program the control system perceives which sort of machining has been selected. In the case of existing C-axis, the function M14 (swivelling-in C-axis) has to be programmed before activating the cycle.

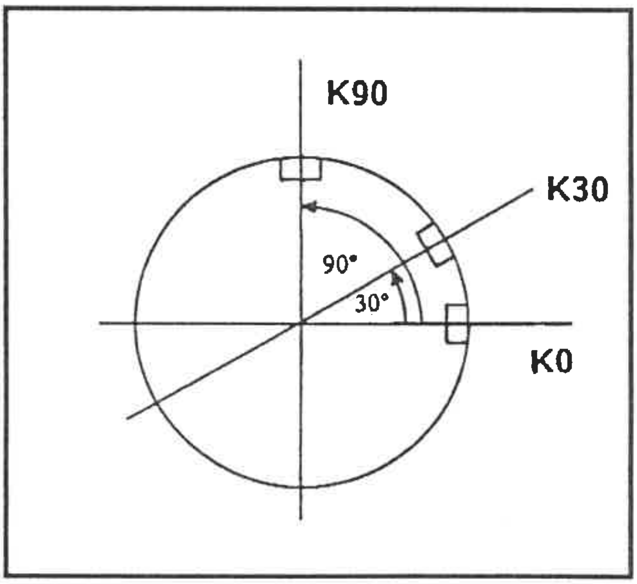


Before activating the cycle, feed in mm/min must be programmed with G94 and speed of the auxiliary drive with G197. Furthermore, the tool must traverse to a suitable starting point S which is located above **one** end point of the keyway.

Required addresses
After selecting G79 the control system requests the following inputs:

DIAMETER	X:
LENGTH	Z:
APPROACH (X)	I:
TOOL ROT. DIRECTION CW = 1 CCW = 2	J:
END ANGLE	K:

If a value is programmed under K, then a point arresting at the given value will be performed before the start of the actual cycle procedure.

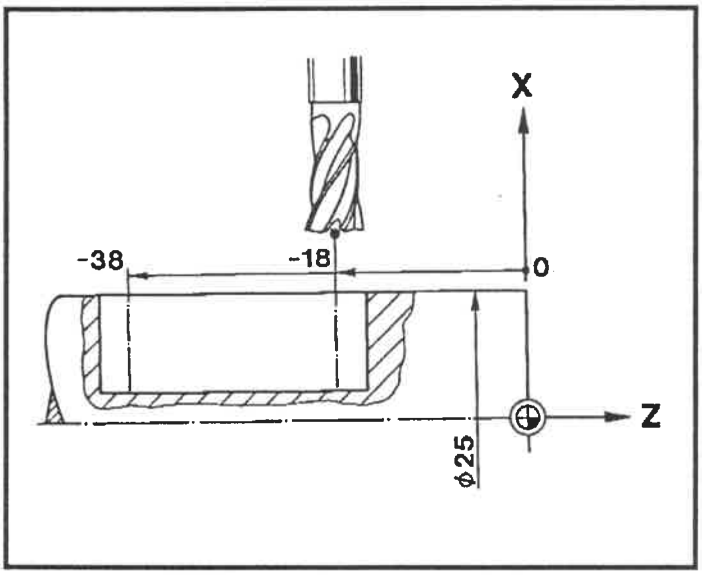


Cycle procedure
The tool traverses to starting point S which represents an end point of the keyway.
The cutter performs the approach in X programmed in I until the end point E of the keyway is reached.
The number of approach operations is given by the depth of the keyway and the programmed approach. It may be possible that the last cutting step will not be performed at the full value of approach.

G79

Programming

N 1 G0 G94 G197 X27 Z-18 F80 S200 T1
N 2 G79 X15 Z-38 I1 J1 K30
N 3 G14 Q0
N 4 M30



Explanation

N 1 Rapid traverse to starting point at one end of the groove. Speed of the driven tool S200 (**WT 13 only**). Feed 80 mm/min.

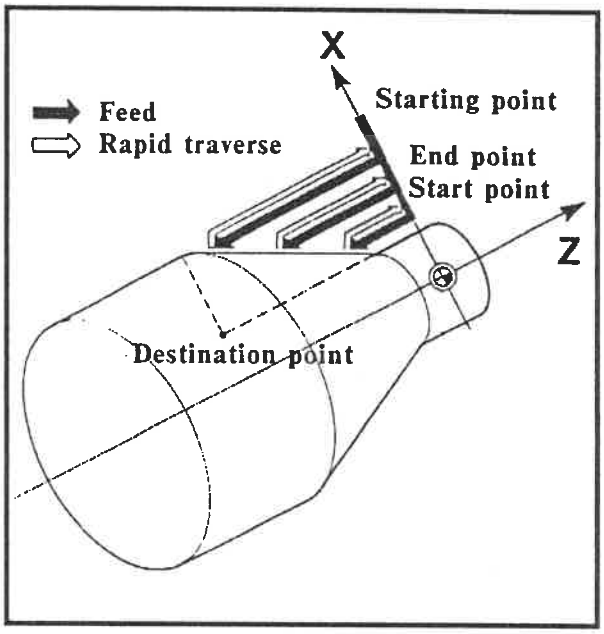
N 2 Activating cycle. Z-value represents the other end point of the keyway with reference to workpiece zero point. The X-value indicates the depth of the keyway as a diameter dimension. Approach in X is 1 mm (i.e. 6 approaches). Direction of rotation of driven tool is clockwise (J = 1). Point arresting at a spindle angle of 30° (K = 30).

N 3 Traverse to tool change point.

N 4 End of program.

Cycles
G80 - G864

Introduction
For frequently occurring work steps, the ELTROPILOT L provides so-called cycles, i.e., the control system knows the necessary procedures for such steps, and only the current dimensions need be entered. Cycles are available for the following machining tasks:
Longitudinal turning
(incl. stock allowance)
Facing (incl. stock allow.)
Multiple cycles
(incl. stock allowance)
Threading
Undercuts
Grooving
We differ between
- cycles **with** contour table
(G817, G827, G818, G828, G819, G829, G83, G836, G861 and G862)
Programming: Cycle call-up, contour table, cycle end
- and cycles **withouth** contour table
(G81 and G82)
Programming: in one block.



Cycle input
After calling the cycle, the required addresses are requested in the input block. Addresses which are not required can be omitted using the confirm key.

Calling the cycle
Cycles are programmed under address G, e.g. G81 cycle for longitudinal roughing.

Destination point ZP (applies only to G81, G82)
The destination point in the cycle is defined under addresses X and Z.
The control system requires the destination point in order to calculate the axis movements. The point need not be engaged.
Input of destination point is possible both in absolute and in incremental dimensioning.

Starting point S
Before the beginning of a cycle, the starting point must be engaged.
From the starting point, the control system calculates all axis movements up to the end of the cycle.

G80 - G864

Contour table
The contour table contains a description of the contour geometry using G-commands. A maximum of 80 geometry elements (straight lines or circular arcs) is permissible. For example, one undercut corresponds to 6 or 7 geometry elements respectively. The intermediate blocks of the SRK generated by the control system are regarded as geometry elements, too. The number of quadrants used represents the number of geometry elements of circular arcs when these are covering several quadrants. Therefore it is useful to divide cycles with contour table into several sub-contours when a large number of geometry elements is contained in the contours of these cycles. Each sub-contour can then be programmed as a subprogram after a **separate** cycle call-up.

Example:

```
N...
N...  G0
N...  G818  ...
N...  L...  (max. 80 geometry elements)
N...  G80
N...  G818  ...
N...  L...  (max. 80 geometry elements)
N...  G80
N...
```

Note: The following functions may not be programmed in **contour cycles** (or in their contour sub programs) and also in the angle circle cycle G770, since they switch off the necessary pre-interpretation of the control system

- **G900 to G999**
- **variable input**
- **tool change (T... or G92 ... T...)**
- **G14**

(disregard of this rule will lead to the display of error messsage **5343.)**

Starting point A
This is the point at which the tool starts to traverse at feed speed after the last approach.

End point E (applies to G81, G82)
At the end of the cycle the tool traverses to point E. This is located at 1mm above the starting point.
When using G818, G819, G828, G829, G861 and G862, at the end of the cycle the tool is positioned **at** the starting point.
When functions G817 and G827 are used, the tool is **not** positioned at the starting point at the end of the cycle.

Cycle end
G80

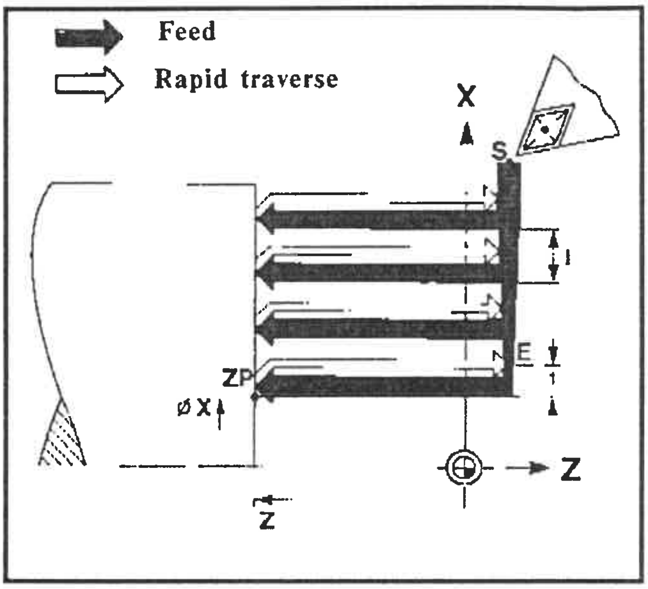
End of cycle, G80
Cycles consisting of several blocks must be ended using the command G80.
Apart from G80 **no other command** must be contained in this block.
In the following all cycles will be described in detail. Examples are provided for explanation.

Important: The axis movement after G80 must not contain a geometry calculation (X? or Z?).
Addresses X and Z must be programmed with a value in the second axis movement at the latest.

Longitudinal cycle G81

LONGITUDINAL cycle, G81

Required addresses
After calling the cycle with G81, the following inputs are requested:



DIAMETER X:
Diameter at destination point

LENGTH Z:
Longitudinal dimension of destination point

APPROACH (X) I:
Amount of feed in X (radius value)

APPROACH (Z) K:
No function, simply confirm

APPROACH PATH FUNCTION
Q = 0 approach in rapid traverse
Q = 1 approach at programmed feed

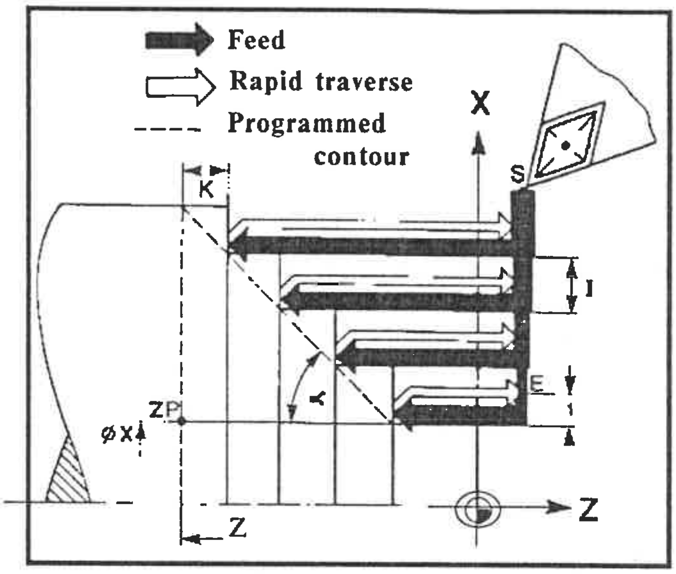
If no value for Q is entered, the approach is in rapid traverse. The control system calculates the approach direction from the start and destination point coordinates. If only one address is required, address I is omitted.

Note
It is imperative that the next traversing path after the cycle is called up with G81 represent a movement in X- and Z-direction.

Cycle procedure
Approach by the amount I from starting point (S) against the direction of the X-axis.
Turn under feed up to programmed Z-value.
Disengage under 45° until 1 mm above the turned diameter.
Rapid traverse to Z coordinate of starting point.
The control system calculates the last approach depth, which may be smaller than I.
Disengage until 1 mm above workpiece, rapid traverse to end point E, which has the Z coordinate of the starting point and the X coordinate of the destination point plus 1 mm.

G81

Longitudinal turning with straight adjoining contour, LONGITUDINAL cycle, G81
In this form of G81, steps are left on the adjoining straight line. There is **no** traversing to destination point!



In contrast to the cycle illustrated on the last page this cycle does contain a programmed contour.

Required addresses
After calling the cycle with G81, the following inputs are requested:

DIAMETER X:
Diameter at destination point

LENGTH Z:
Longitudinal dimension of destination point

APPROACH (X) I:
Amount of feed in X (radius value)

APPROACH (Z) K:
Offset from one cut to the other in Z

APPROACH PATH FUNCTION Q:
Q = 0 approach in rapid traverse
Q = 1 approach at programmed feed

If no value for Q is entered, the approach is in rapid traverse. If only one cut is required, addresses I and K are omitted!

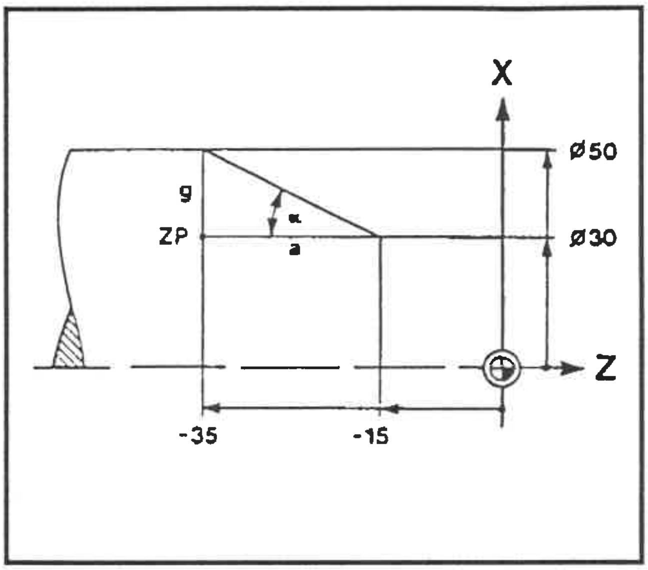
Note
It is imperative that the next traversing path after the cycle is called up with G81 represent a movement in X- **and** Z-direction.

Cycle procedure
Approach by the amount I from starting point (S) against the direction of the X-axis.
Turn under feed up to programmed Z coordinate minus stock allowance.
Disengage under 45° until 1 mm above the workpiece.
Rapid traverse to Z coordinate of starting point.
Approach again (I + 1 mm)
...
...
The control system calculates the last approach depth, which may be smaller than I.
Disengage until 1 mm above workpiece, rapid traverse to end point E, which has the Z coordinate of the starting point and the X coordinate of the destination point plus 1 mm.

G81

Calculation of K

$$K = \frac{I}{\tan a}$$



In the right-angled triangle, the following applies:

$$\tan a = \frac{\text{opposide side g}}{\text{adjacent side a}}$$

a = 35-15 = 20

$$g = \frac{50-30}{2} = 10$$

(30 and 50 are diameter values)

K can then be calculated:

At the programmed cutting depth of I = 1 mm,

K = 1 : 1/2 = 2

In this example, G817, G818 and G819 can also be used, in which case K need not be calculated.

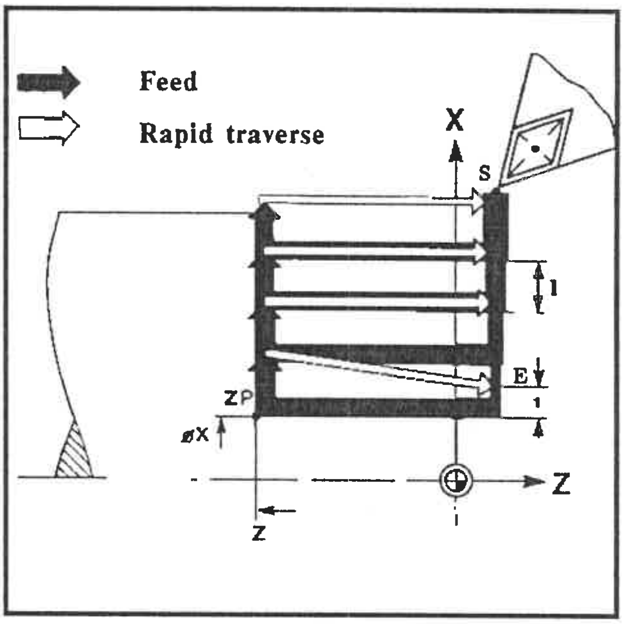
G81

Longitudinal turning with straight adjoining contour and subsequent facing, LONGITUDINAL cycle, G81

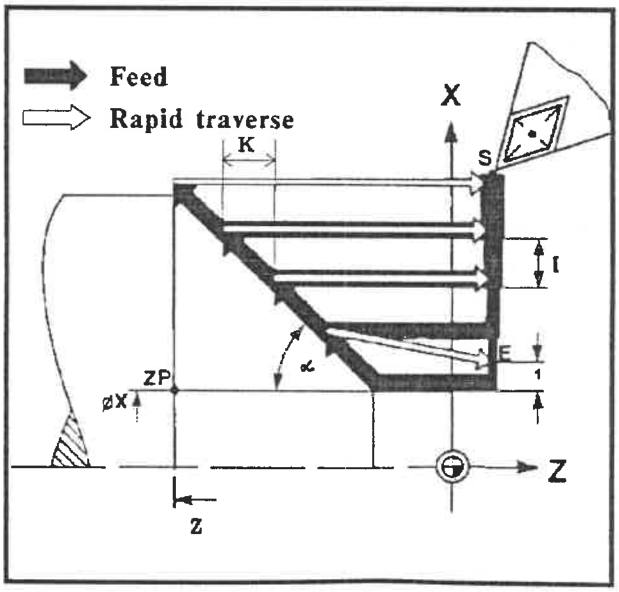
Required addresses:
In order to true the steps, **address I** must be programmed with **negative sign.** The other addresses remain as in the previous example!

Note
It is imperative that the next traversing path after the cycle is called up with G81 represent a movement in X- and Z-direction.

Cycle procedure:
Instead of disengaging at 45° and moving back to 1 mm clearance, the workpiece is turned, after the feed movement, longitudinally along the adjoining straight line up to the previous approach depth, thus removing the steps!

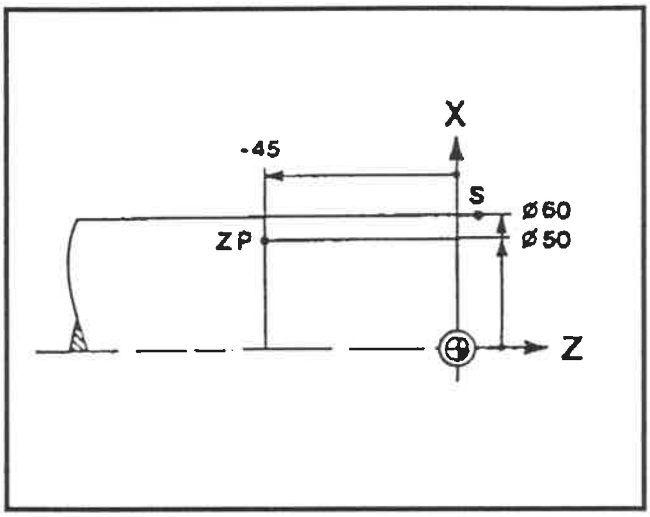


By means of I- (minus) a corresponding tool movement can be programmed even without a right-angled continuation.



G81

Example 1
Starting point: X 60.0
Z 2.0



Programming
N 0 G96 F0.1 S150 T3 M4 M7
N 1 G0 X60 Z2
N 2 G81 X50 Z-45
N 3 G0 X150 Z100 M30

Explanation
N 0 Start conditions.

N 1 Rapid traverse to starting point.

N 2 Call cycle. Destination point is programmed under X and Z. As no I and K have been entered, machining is done in one cut.

N 3 Traverse to tool change point, end of program.

If **several cuts** are to be made, block N2 is modified:

N 2 G81 X50 Z-45 I2

The tool makes three cuts. The first two at 2 mm and the last at 1 mm cutting depth.

If the **cross surface** is also to be machined, I must be programmed with negative sign:

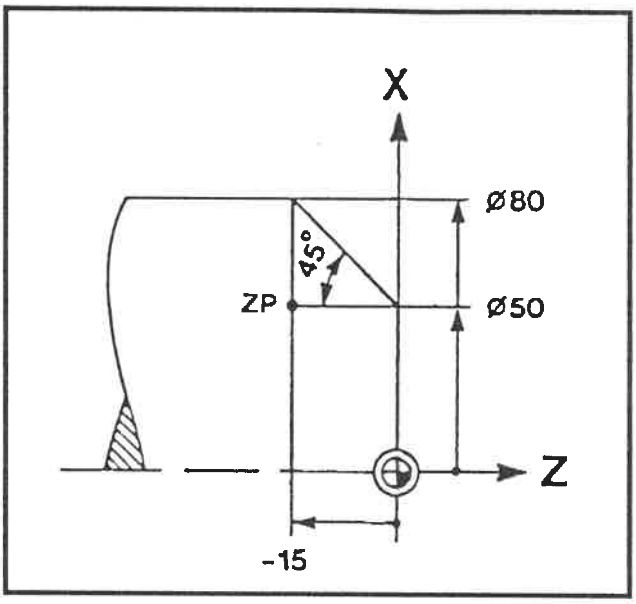
N 2 G81 X50 Z-45 I-2

The tool makes 3 cuts. The negative sign of I means that a cross cut is made in addition to the longitudinal cut.

G81

Example 2

Starting point: X 80.0
Z 2.0



Programming

N 0 G96 F0.3 S150 T3 M4 M7
N 1 G0 X80 Z2
N 2 G81 X50 Z-15 I-5 K5
N 3 G0 X150 Z100 M30

Explanation

N 0 Start conditions.

N 1 Rapid traverse to starting point.

N 2 Call cycle. Destination point is programmed under X and Z. Three cross cuts are turned. K effects the stock allowance:

tan a = g/a = 15/15 = 1

At a programmed cutting depth of I = 5 mm, K = 5

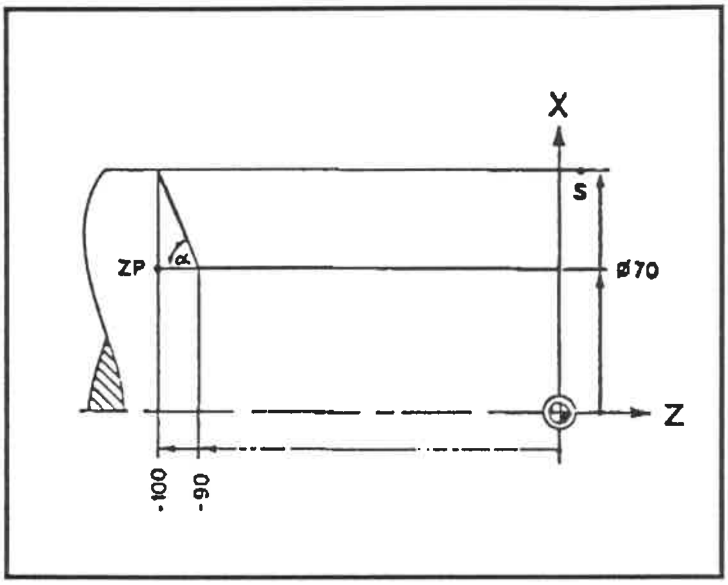
N 3 Traverse to tool change point, end of program.

G81

Example 3
Feed value I = 4 mm

$$\tan a = \frac{120-70}{2} : 10 = 2.5$$

$$K = \frac{4}{2.5} = 1.6$$



Programming
N 0 G90 G95 G96 F0.5 S180 T1 M4 M7
N 1 G0 X120 Z2
N 2 G81 X70 Z-100 I-4 K1.6
N 3 G0 X200 Z150
N 4 M30

Explanation

N 0 Start conditions (absolute dimension, feed in mm/rev., cutting speed constant, feed 0.5 mm/rev., 180 m/min, tool number 1, spindle on, coolant on, gear range 3.

N 1 Rapid traverse to starting point 2 mm before workpiece.

N 2 **"Longitudinal turn"** cycle; destination point 70 mm ; length 100 mm; feed value 4 mm; tool trues step (I-); stock allowance 1.6 mm.

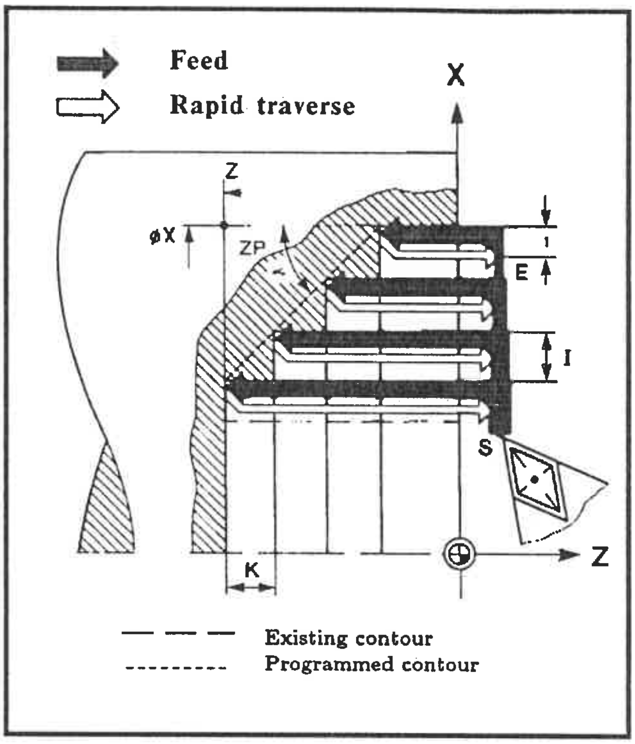
N 3 Rapid traverse away from workpiece.

N 4 End of program.

G81

Longitudinal turning on inside contour, LONGITUDINAL cycle, G81

Required addresses:
After calling the cycle with G81, the following inputs are requested:



DIAMETER X:
Diameter at destination point

LENGTH Z:
Longitudinal dimension of destination point

APPROACH (X) I:
Amount of feed in X (radius value)

APPROACH (Z) K:
Offset from one cut to the other in Z.

APPROACH PATH FUNCTION
Q = 0 approach in rapid traverse
Q = 1 approach at programmed feed

If no value is entered for Q, approach is in rapid traverse.

Programming
As for outside contour.
The control system recognizes whether machining is outside or inside by the coordinates of the starting and destination points.

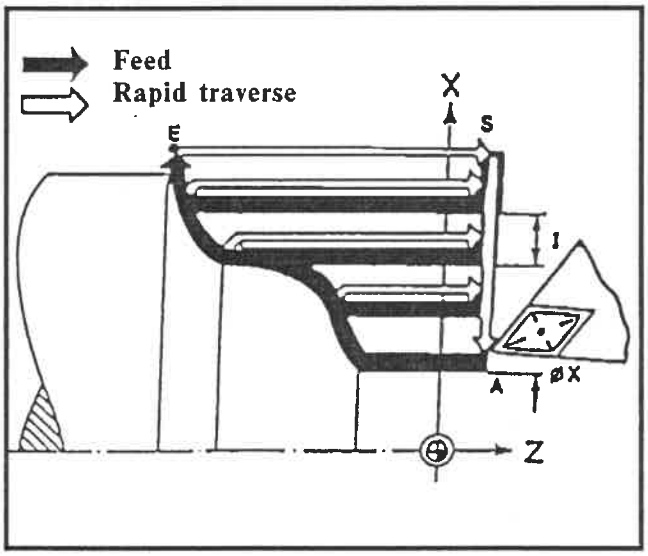
Note
It is imperative that the next traversing path after the cycle is called up with G81 represent a movement in X- and Z-direction.

Cycle longitudinal roughing G817

Longitudinal turning with any adjoining contour, without plunging into contour, with truing of steps, Cycle LONGITUDINAL ROUGHING G817

Programming
The blocks following the cycle call with G817 contain the description of the contour against which cutting is being done. The destination of the first G command must be the starting point of the contour, as the control system takes the first path element of the cycle to be the approach to the contour. Note: The starting point may not be on the contour. All following blocks are contour description consisting of max. 80 NC blocks. End of cycle must then be programmed with G80 in a separate block.

Required addresses
After selecting G817, the control system requests the following inputs:



DIAMETER X:
Parameter X represents feed movement limitation in X direction, i.e. if X is situated above the X-value of the contour start, the contour will be only partially machined.

Application: divide roughing into segments of varying approach.

APPROACH (X) I:

APPROACH PATH FUNCTION Q:
Q = 0 Adjustment during rapid traverse
Q = 1 Adjustment with programmed feed

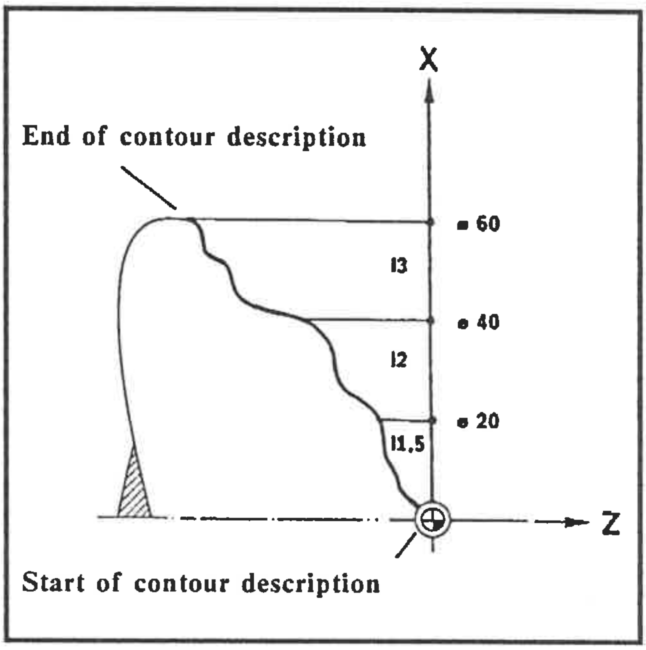
If no Q is given, the adjustment is made with rapid traverse.
If the last adjustment reaches the contour, this adjustment motion is executed with programmed feed instead of rapid traverse.

Cycle procedure
Basically as for G81, but in this case a cut is made along the programmed contour at the end of the cutting cycle.
At the end of the cycle the tool is positioned at 1mm above the deepest machined approach.

G817

Dividing roughing into segments of varying approach
At the end of the cycle the tool is positioned 1 mm above the deepest machined approach. Therefore the roughing procedure can easily divided into segments of varying approach. The contour table is written in a subprogram.

Proceed as in the following program structure:



%1909 (main program)

```
N...
N... G0 X62 Z2
N... G817 X40 I3
N... L4711
N... G80        (Cycle ends on X42)
N... G817 X20 I2
N... L4711
N... G80        (Cycle ends on X22)
N... G817 X0 I1.5
N... L4711
N... G80        (Cycle ends on X2)
N... G14
N...
```

%4711 (subprogram)

```
N... G0 X0  Z0
N...
N...
N...
N... G1 X... Z...
N... M30
```

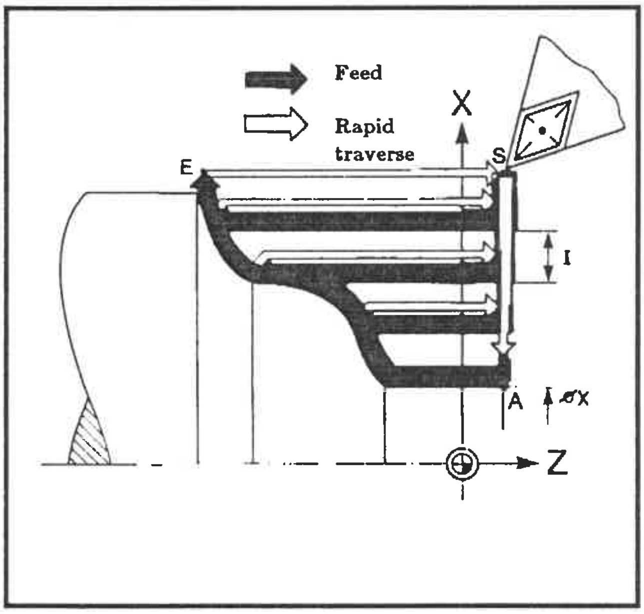
Note
Falling contour elements are **not** treated even if they are contained in the contour table. They can be treated subsequently with, for example, G819 or G862 and a tool with the corresponding cutting angle.

Cycle longitudinal roughing G818

Longitudinal turning with any adjoining contour, without plunging into contour, truing of steps, return to starting point, Cycle LONGITUDINAL ROUGHING G818

Programming
The blocks following the cycle call with G818 contain the description of the contour against which cutting is being done. The destination of the first G command must be the starting point of the contour, as the control system takes the first path element of the cycle to be the approach to the contour. **Note:** The starting point may not be on the contour. All following blocks are contour description consisting of max. 80 NC blocks. End of cycle must then be programmed with G80 in a separate block.

Required addresses
After selecting G818, the control system requests the following inputs:



DIAMETER X:
Parameter X represents feed movement limitation in X direction, i.e. if X is situated above the X-value of the contour start, the contour will be only partially machined (**Application:** dividing roughing into segments of varying approach).

APPROACH (X) I:

APPROACH PATH FUNCTION Q:
Q = 0 Adjustment during rapid traverse
Q = 1 Adjustment with programmed feed

If no Q is given, the adjustment is made with rapid traverse.
If the last adjustment reaches the contour, this adjustment motion is executed with programmed feed instead of rapid traverse.

Cycle procedure
Basically as with G81, but in this case a cut is made along the programmed contour at the end of the cutting cycle.
At the end of the cycle the tool is positioned back at the starting point.

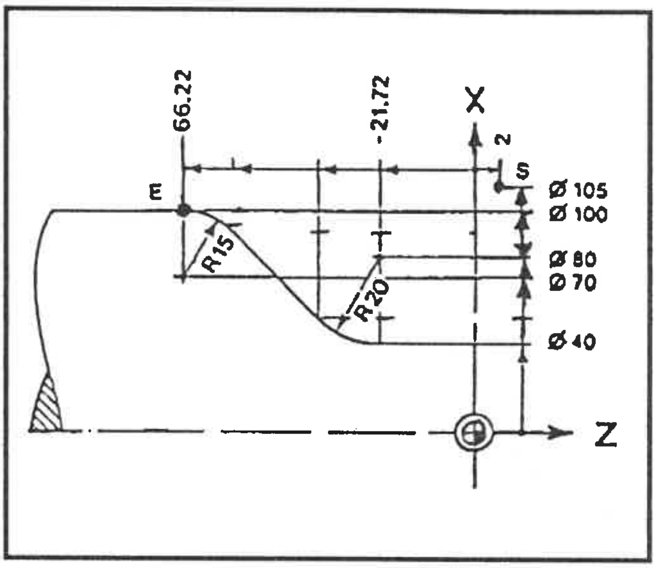
Note
Falling contour elements will **not** be treated, even when they are contained in the contour table.
They can subsequently be treated with, for example, G819 or G862 and a tool with the corresponding cutting angle.

G818

Example

Programming

```
N 0  G96  G95 F0.12  S150  T1  M4
N 1  G0  X105  Z2
N 2  G818  X40 I5
N 3  G1  X40  Z-21.72
N 4  G2  X?  Z?  I20  K0
N 5  G1  X?  Z?
N 6  G13  X100  Z-66.22  I35  K-66.22
N 7  G80
N 8  G14  M30
```



Explanation

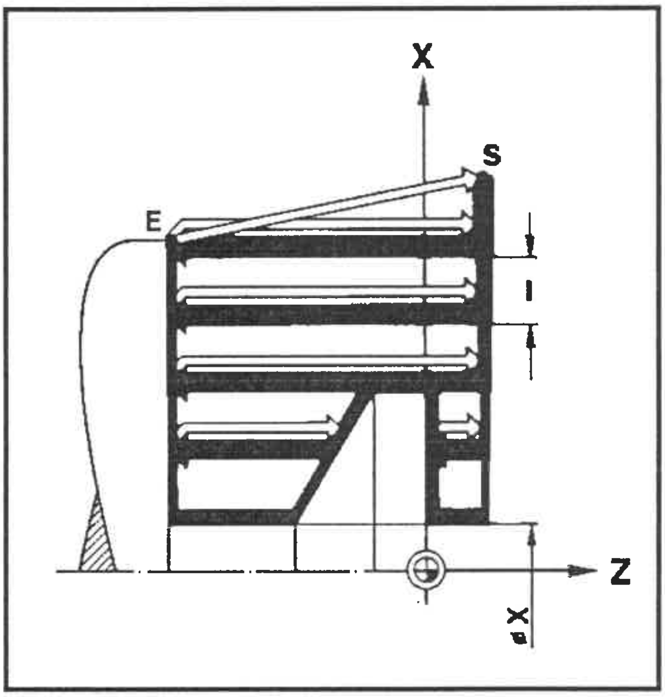
N 0	Starting conditions.
N 1	Rapid traverse to starting point.
N 2	Declaration of cycle longitudinal turning with any adjoining contour. X indicates **not the destination point**, but the **first point of the contour A.**
N 3	Straight line with destination point.
N 4	Circular arc. The destination point is calculated automatically.
N 5	Straight line. The destination point is calculated automatically.
N 6	Circle, with indication of center of the circle.
N 7	End of cycle.
N 8	Rapid traverse to tool change point, end of program.

Cycle procedure

Longitudinal cutting along the contour, tool is retracted, traverses back, approaches and performs next cut and so on. At the end the tool moves along the contour and removes the "steps". Finally the tool traverses back to the starting point.

Falling contour cycle G819

FALLING CONTOUR cycle (longitudinal), G819 with plunging into contour
If a contour is to be described which runs in longitudinal direction and contains falling contour elements, this can be machined more simply using contour cycle G819.
The destination of the first G command must be the starting point of the contour, as the control system takes the first path element of the cycle to be the approach to the contour. **Note:** The starting point may not be on the contour. The individual contour elements are programmed in the subsequent blocks.
The contour description may not exceed 80 NC-blocks.



Required addresses
After selecting G819, the control system requests the following inputs:

DIAMETER X:
Cutting depth in the diameter dimension for the last approach, usually the start point of the contour.
If, for example, X lies above the X-value of the contour start, the first part of the contour (up to the point where the cut first meets the contour) will be only partially machined. The limitation by X is, however, not effective in contour valleys.

(**Application:** divide machining into segments of different approach)

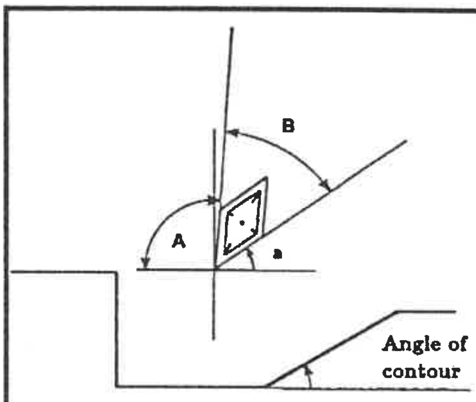
APPROACH (X) I:
Approach depth of the individual cuts

SPECIAL FEED E:
for plunging

G819

Cycle procedure
Error message **"Residual material not treated because of tool geometry"** may appear due to a modification of the contour by the SRK.
After the last approach has been completed, the tool traverses along the contour to the end point and then back to the starting point.
If the last adjustment reaches the contour, this adjustment motion is executed with programmed feed instead of rapid traverse.

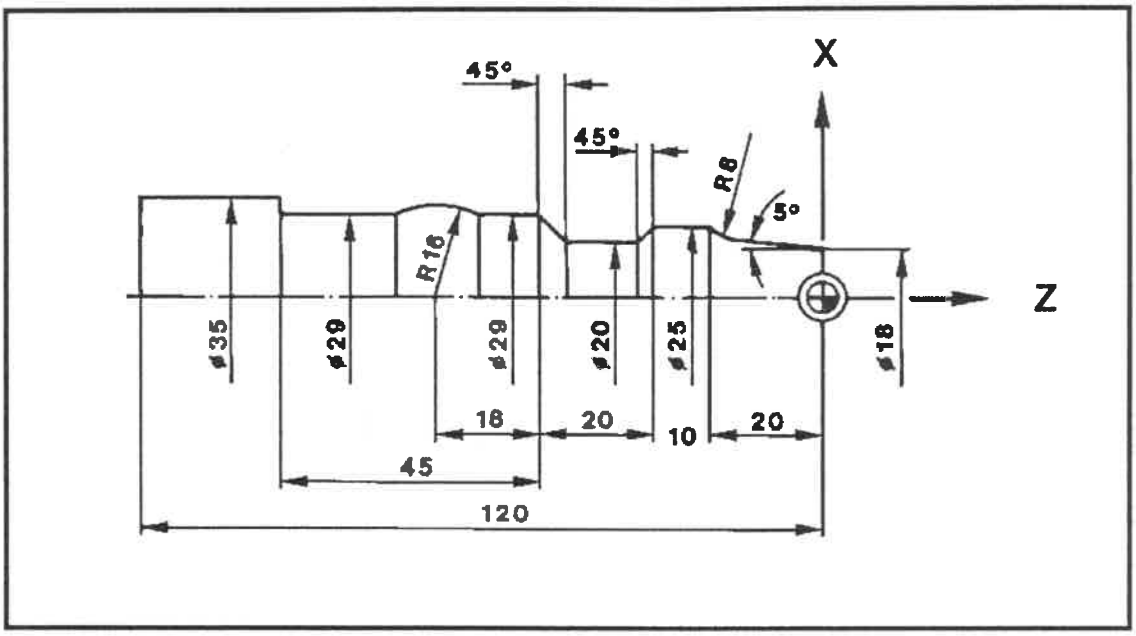
Tool
The tool used can be a roughing tool, a finishing tool or a button tool. When roughing or finishing tools are used, the angles at the cutting edge must be entered in the tool file under A and B.
The tool plunges at a maximum angle of 180°- A - B.
If a clearance angle is to remain during plunging, a larger cutting angle B can be entered.



e.g.: A = 93°; B = 55°; a = 32°
Angle of contour = 30°

G819

Example
Falling contour
Simplified geometry-programming
for transition straight line - circle



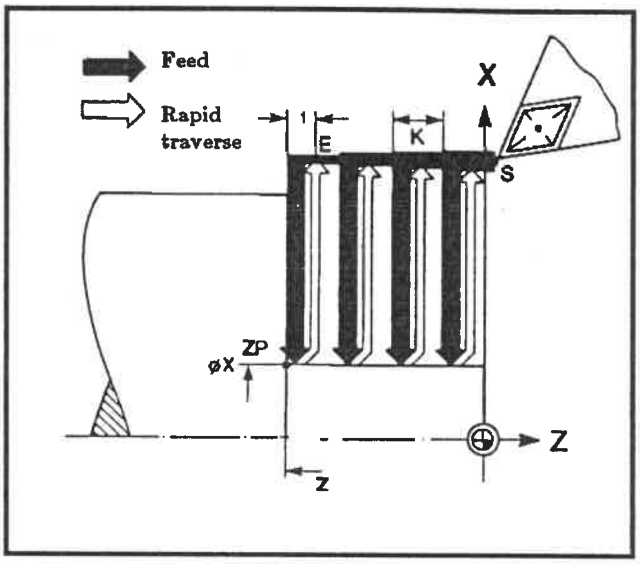
N 1 G96 G95 F0.3 S200 T1 M4
N 2 G0 X40 Z2
N 3 G819 X18 I2 E0.15
N 4 G0 X18 Z0 G42
N 5 G1 X? Z? A5
N 6 G2 X25 Z-20 R8 B0
N 7 G1 Z-30
N 8 G1 X20 Z? A-45
N 9 G1 Z? A0
N10 G1 X29 Z-50 A45
N11 G1 Z? A0 B0
N12 G13 Z? R16 I0 K-68 B0
N13 G1 Z-95
N14 G1 X40 G40
N15 G80
N16 M30

Explanation

N 1	Starting conditions.
N 3	Cycle call, feedrate I = 2 mm.
N 4 to N14	Description of the (mainly) falling contour. Simplified Geometry-programming for conical shaft thickness increase, decrease at 45°slope and round bulges.
N15	End of cycle.

Cycle transversal G82

Facing with right-angled adjoining contour, CROSS cycle, G82



Required addresses
After calling the cycle with G82, the control system requests the following inputs:

DIAMETER X:
Diameter at destination point

LENGTH Z:
Destination position in longitudinal direction

APPROACH (X) I:
No function, simply confirm

APPROACH (Z) K:
Amount of feed in Z-direction

APPROACH PATH FUNCTION Q:

Q = 0 Approach in rapid traverse
Q = 1 Approach at programmed feed speed

If no value for Q is entered, approach is in rapid traverse.
The control system recognizes whether machining is outside or inside by the coordinates of the starting and destination points and by the sign of I and K of the tool in the tool file.

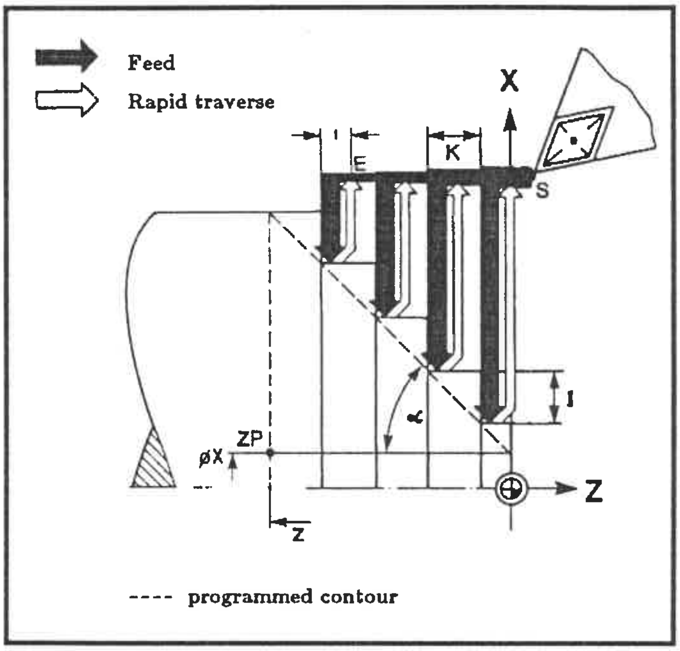
Note
It is imperative that the next traversing path after the cycle is called up with G82 represent a movement in X- and Z-direction.

Cycle procedure
- Approach by the amount K from starting point in direction Z- (minus).
- Turn under feed up to programmed diameter.
- Disengage under 45° until 1 mm above the workpiece.
- Rapid traverse to starting diameter.
- Approach once more (K+1 mm). The control system calculates the last feed rate which could be smaller than K.
- Disengage, rapid traverse to end point.

G82

Facing with straight adjoining contour, CROSS cycle, G82

Required addresses
After calling the cycle with G82, the control system requests the following inputs:



DIAMETER X:
Diameter at destination point

LENGTH Z:
Destination position in longitudinal direction

APPROACH (X) I:
Offset (X-direction)

APPROACH (Z) K:
Amount of feed (Z-direction)

APPROACH PATH FUNCTION Q:
If no value is programmed for Q, then approach is in rapid traverse.
The control system recognizes whether machining is outside or inside by the coordinates of the starting and destination points.

Calculating I

$$\boxed{I = K \cdot \tan a}$$

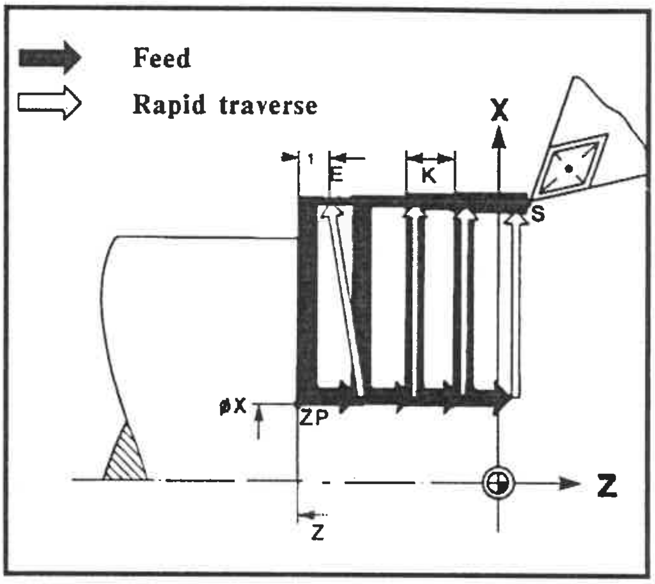
Steps are left on the adjoining straight line if K is programmed with positive sign!

Cycle procedure
- Approach from starting point in direction Z (minus) by K.
- Turn in rapid traverse to programmed diameter minus offset I.
- Disengage under 45° to 1 mm above workpiece.
- Rapid traverse to starting diameter.
- Approach once more (K+1 mm). The control system calculates the last amount of feed, which may be smaller than K!
- Disengage, rapid traverse to end point.

G82

Facing with straight adjoining contour and subsequent longitudinal turning, TRANSVERSAL cycle G82

Right-angled continuation
By entering K- (minus), a corresponding tool movement in longitudinal direction can be programmed.



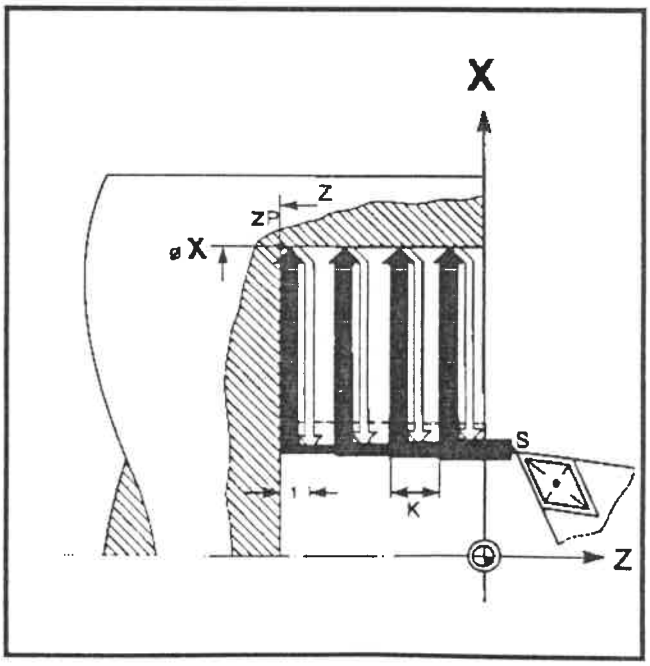
Required addresses:
In order to true the steps, **address K must be programmed with negative sign.**

The other addresses remain as in the previous example!

Note
It is imperative that the next traversing path after the cycle is called up with G82 represent a movement in X- and Z-direction.

Cycle procedure
After the feed movement, the workpiece is turned in transversal direction along the adjoining straight line up to the previous feed depth (instead of disengaging at 45° and moving back to 1mm clearance as in the previous example). The steps are thus removed.

Facing on inside contour

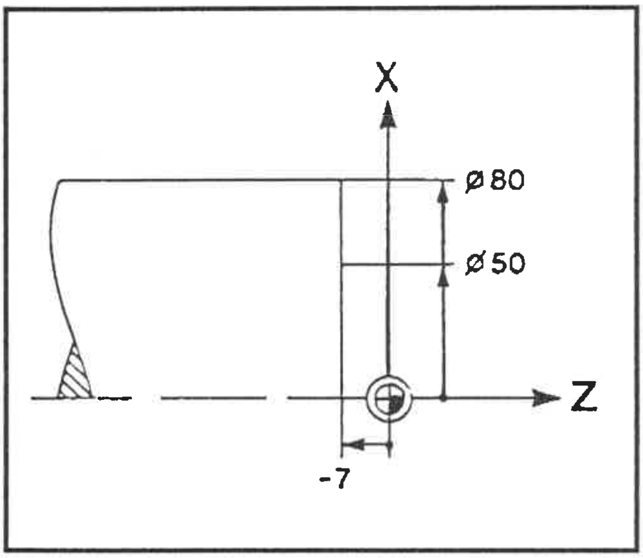


Programming
As for outside contour. The control system recognizes whether machining is inside or outside by the coordinates of the starting and destination points and by the sign of I and K of the tool in the tool file.

G82

Example 1

Programming
N 0 G90 G95 G96 F0.1 S150 T4 M4 M7
N 1 G0 X82 Z0
N 2 G82 X50 Z-7
N 3 G0 X150 Z100 M30



Explanation
N 0 Start conditions.

N 1 Rapid traverse to starting point.

N 2 Call cycle. Destination point is programmed under X and Z.
As no I and K have been entered, machining is done in one cut.

N 3 Traverse to tool change point, end of program.

If **several cuts** are to be made,
block N2 is modified thus:

N 2 G82 X50 Z-7 K3

The tool makes three cuts. The first two at 3 mm and the last at 1 mm cutting depth.

If the **front face** is also to be machined,
K- must be programmed:

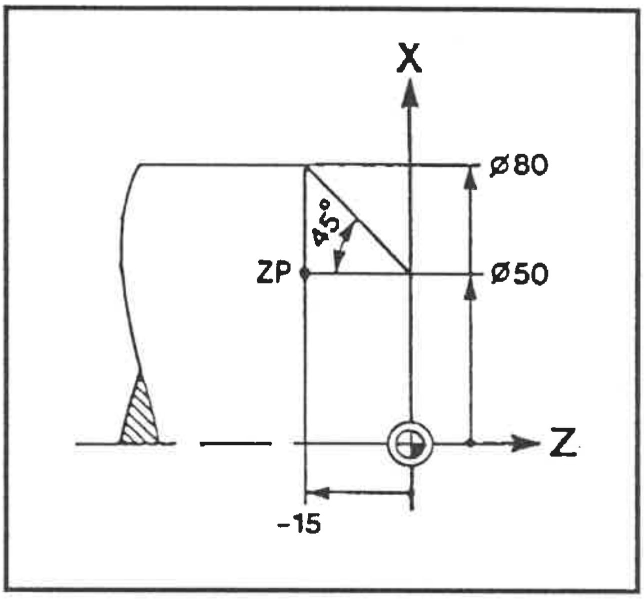
N 2 G82 X50 Z-7 K-3

The tool makes three cuts. The negative sign of K- means that a longitudinal cut is made additionally.

G82

Example 2

Programming
N 1 G0 G96 G95 X82 Z0 F0.1 S150 T1 M4
N 2 G82 X50 Z-15 I5 K-5
N 3 G0 X250 Z20 M30



Explanation

N 1 Rapid traverse to starting point.

N 2 Call longitudinal turning cycle. Destination point is determined, I5 determines the offset in X-direction, K-5 determines cutting depth and machining of the contour.

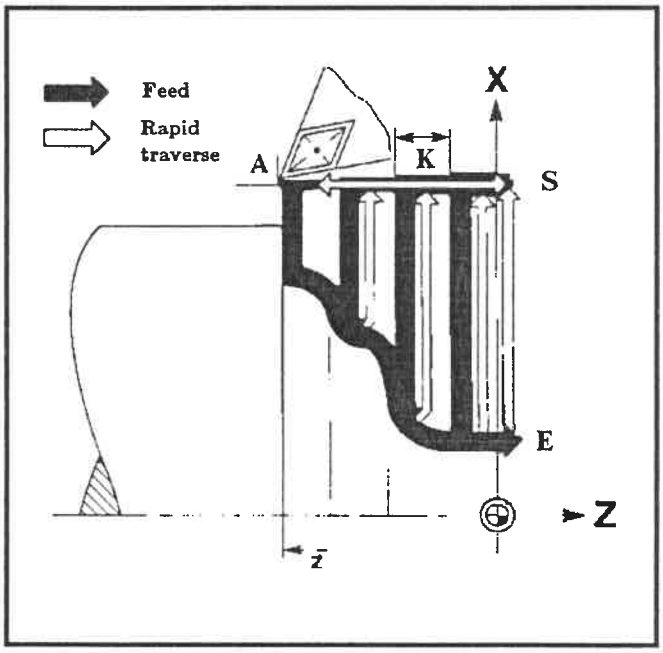
N 3 Traverse to tool change point, end of program.

Transverse roughing G827

Longitudinal turning with any adjoining contour, truing of steps, without plunging into contour, TRANSVERSE CUT cycle, G827

Programming
The blocks following the cycle call with G827 contain the description of the contour against which cutting is being done. The destination of the first G command must be the starting point of the contour, as the control system takes the first path element of the cycle to be the approach to the contour. Note: The starting point may not be on the contour. All following blocks are contour description consisting of max. 80 NC blocks. End of cycle must then be programmed with G80 in a separate block.

Required addresses
After selecting G827, the control system requests the following inputs:



LENGTH Z:
Parameter Z represents feed movement in Z-direction; if e.g. Z is situated above the Z-value of the contour start, the contour will be only partially machined. (Application: divide roughing into sections of varying approach).

APPROACH (Z) K:

APPROACH PATH FUNCTION Q:
If no value is entered for Q or it is programmed with zero, then approach is in rapid traverse. If approach is to be with G01, then Q1 has to be programmed.
If the last adjustment reaches the contour, this adjustment motion is executed with programmed feed instead of rapid traverse.

Cycle procedure
Basically as for G82, but in this case a cut is made along the programmed contour at the end of the cutting cycle. Between disengaging the tool from the workpiece and the next approach a period of dwell of one second is automatically inserted by the control system (for adaptation of speed). **At the end of the cycle the tool is positioned 1 mm above the deepest machined point.**

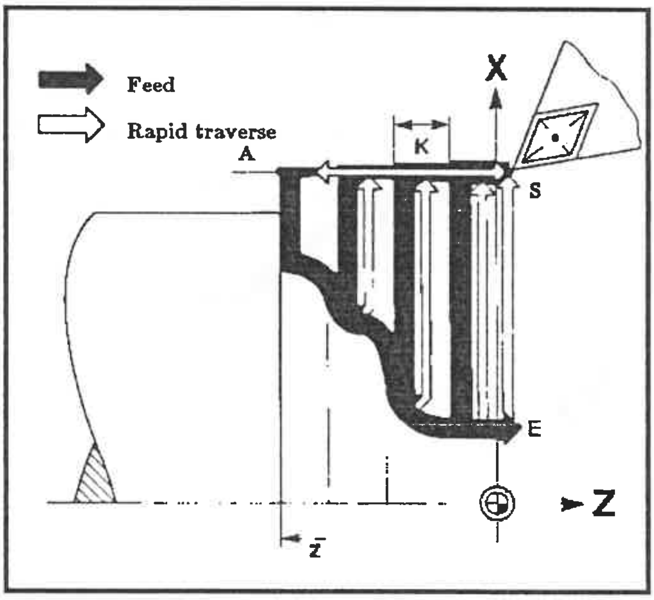
Transverse roughing G828

Longitudinal turning with any adjoining contour, truing of steps, without plunging into contour, TRANSVERSAL CONTOUR cycle, G828

Programming
The blocks following the cycle call with G828 contain the description of the contour against which cutting is being done. The destination of the first G-command must be the starting point of the contour, as the control system takes the first path element of the cycle to be the approach to the contour. Note: The starting point may not be on the contour. All following blocks are contour description consisting of max. 80 NC blocks. End of cycle must then be programmed in a separate block.

Required addresses
After selecting G828, the control system requests the following inputs:

LENGTH Z:
Parameter Z represents feed movement in Z-direction; if e.g. Z is situated above the Z-value of the contour start, the contour will be only partially machined (**Application:** divide machining into sections of varying approach).



APPROACH (Z) K:

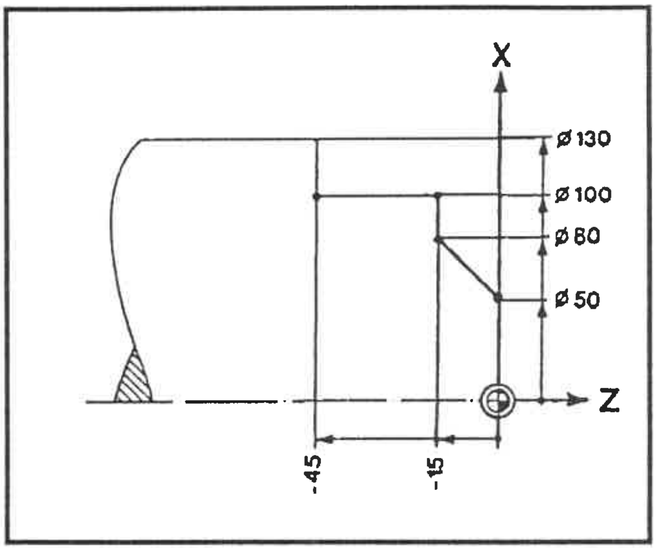
APPROACH PATH FUNCTION Q:
If no value is entered for Q or it is programmed with zero, then approach is in rapid traverse. If approach is to be with G01, then Q1 has to be programmed.

Cycle procedure
Basically as for G82, but in this case a cut is made along the programmed contour at the end of the cutting cycle. Between disengaging the tool from the workpiece and the next approach a period of dwell of one second is automatically inserted by the control system (for adaptation of speed). **At the end of the cycle the tool is positioned at the starting point.**
If the last adjustment reaches the contour, this adjustment motion is executed with programmed feed instead of rapid traverse.

G828

Programming

N1 G95 G97 F0.3 M4 M7 S650 T1
N2 G0 X132 Z2
N3 G828 Z-45 K5
N4 G1 X100 G41
N5 G1 Z-15
N6 G1 X80
N7 G1 X50 Z0 G40
N8 G80
N9 G14 M30

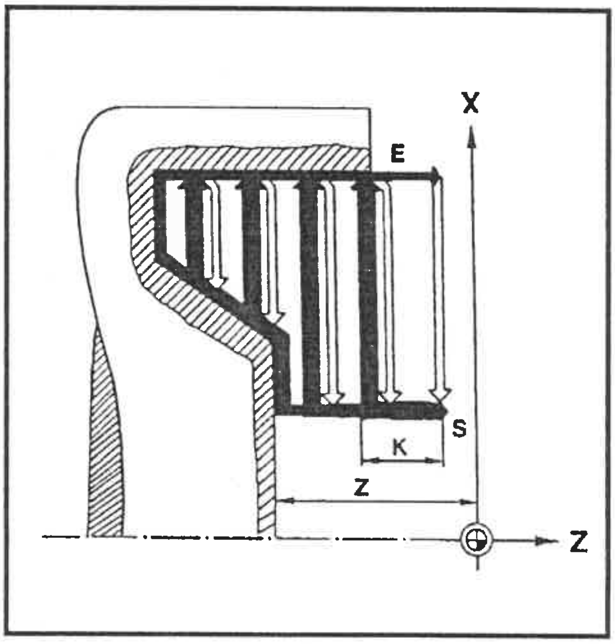


Explanation

N1	Starting conditions
N2	Rapid traverse to a point in front of workpiece
N3	Cycle call, approach in Z-direction (K = 5)
N4	Straight line (traverse to contour); activating SRK
N5	Straight line
N6	Straight line
N7	Straight line, cancelling SRK
N8	End of cycle
N9	Traverse to tool change point; end of program

Transverse roughing G829

Cycle FALLING CONTOUR (transversal), G829 with plunging into contour
If a contour is to be described which runs in transversal direction and contains mainly falling contour elements in X-direction, this can be machined more simply using contour cycle G829. The destination of the first G-command must be the starting point of the contour, as the control system takes the first path element of the cycle to be the approach to the contour. **Note: The starting point may not be on the contour.** The individual contour elements are programmed in the subsequent blocks.
The contour description may not exceed 80 NC blocks.



Required addresses
After selecting G829 the control system requests the following inputs:

LENGTH Z:
Cutting depth for the last approach; usually the start point of the contour in transversal direction.
If e.g. Z lies in front of the Z-value of the contour start, the first part of the contour (up to the point where the cut first meets the contour) will be only partially machined. The limitation by Z is, however, not effective in contour valleys.
(**Application:** Divide machining into segments of different approach)

APPROACH (Z) K:
Approach depth of the individual cuts in Z-direction.

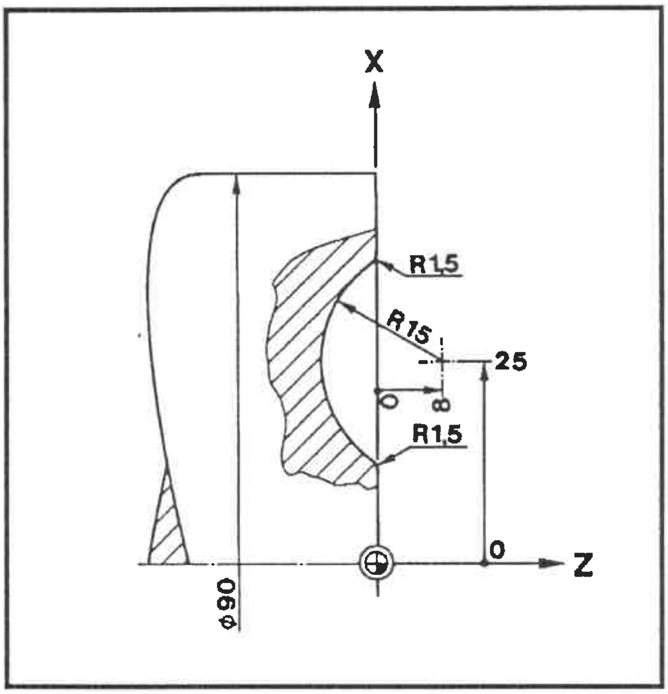
SPECIAL FEED E:
for plunging

Cycle procedure
The error message **"Residual material not treated because of tool geometry"** may appear due to a modification of the contour by the SRK.
After the last approach has been completed, the tool traverses along the contour to the end point and then back to the starting point.
The control system automatically inserts a period of dwell of one second between each retracting of the tool from the workpiece and the following approach.
If the last adjustment reaches the contour, this adjustment motion is executed with programmed feed instead of rapid traverse.

G829

Tool
The tool used can be a roughing tool, a finishing tool or a button tool. When roughing or finishing tools are used, the angles at the cutting edge must be entered in the tool file under A and B. The tool plunges at a maximum angle of 180°- A - B. If a clearance angle is to remain during plunging, a larger cutting angle B can be entered.

Example



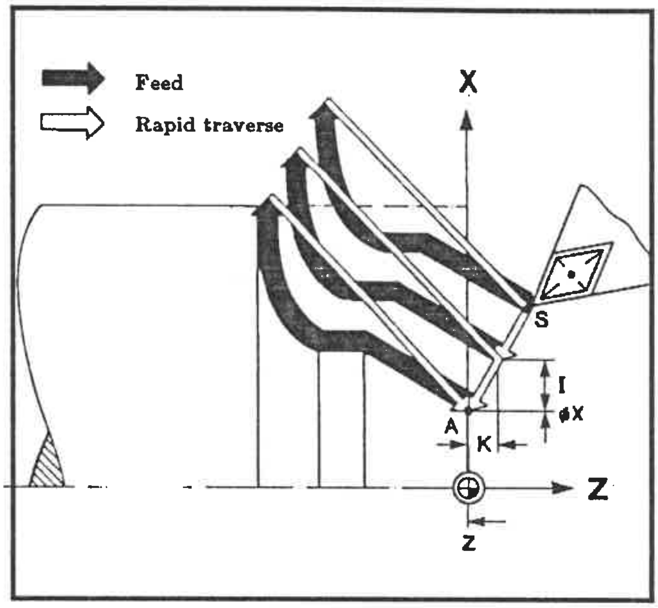

```
...
...
N 8  G0  X90  Z3  T2
N 9  G829  Z0  K2
N10  G1  X90  Z0
N11  G1  X?  A-90  B1.5
N12  G13  X?  R15  I25  K8  B1.5
N13  G1  X0
N14  G80
...
...
```

Explanation

N 8	Rapid traverse to contour.
N 9	Cycle call, feedrate K = 2 mm.
N10 to N13	Contour description. Simplified geometry programming for transition straight line - roundance.
N14	End of cycle.

Cycle contour G83

Cycle CONTOUR, G83
(roughing parallel to contour)
This cycle is for machining where G81 and G82 are not expedient, e.g. with pre-formed blanks.



Required addresses
After selecting G83 the control system requests the following inputs:

DIAMETER	X:
LENGTH	Z:
APPROACH (X)	I:
APPROACH (Z)	K:

Programming
In programming, the following cases must be differentiated:

- Machining **formed parts**
- Machining **solid material**
- Use as a **multiple cycle**, i.e. programmed movement steps can be carried out repeatedly in various locations using cycle G83.

Note
Feed can not be changed during the cycle G83. In case feed changes are programmed within the contour description of G83, they will be ignored by the control system.

G83

Programming formed parts
Cycle CONTOUR, G83
(roughing parallel to contour)

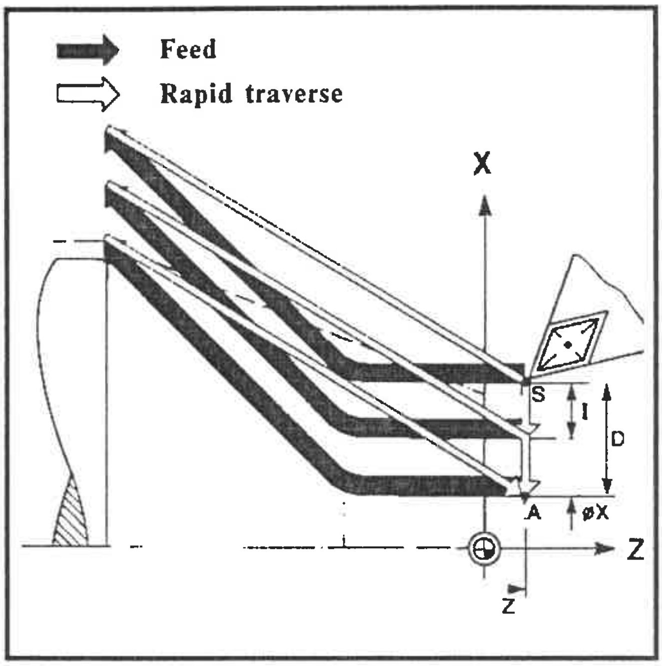
Example:
N 10 G83 X... Z... I... K...

The end contour is programmed in the following blocks. The first point is programmed in the block containing the cycle call.

It is expedient to program constant cutting speed G96.

G80, end of cycle, must be alone in a separate block.

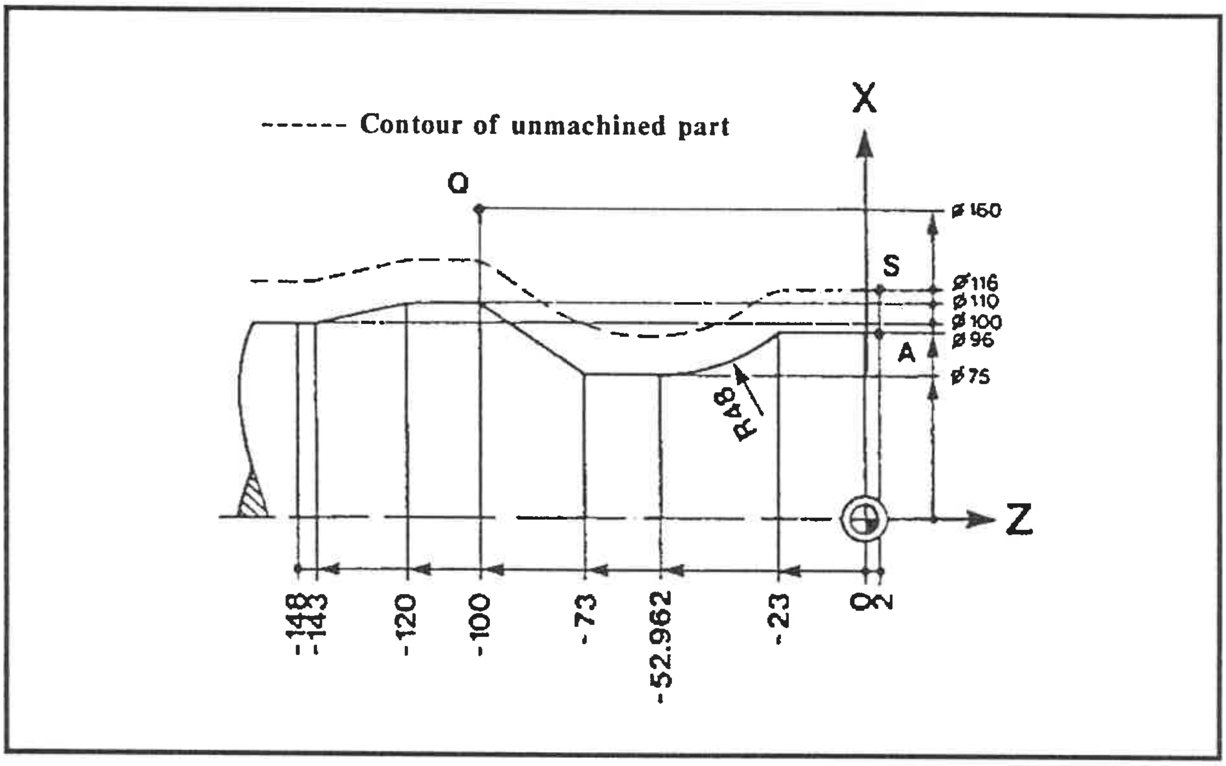
After G80, the last programmed feed or last spindle speed is active.



Cycle procedure
The tool is positioned at machining starting point S. The tool must traverse to this point in the block **before** the cycle call.
The following is a description of the approach procedure in X-direction. Approach in Z-direction is done accordingly.
The value of I and K determines the **length** of the individual approach path.
The **direction** of approach is always from starting point S to start point A.

G83

Example



Programming

N 0 G90 G95 G96 F0.5 S180 T1 M4 M8
N 1 G0 X116 Z2
N 2 G83 X96 I6
N 3 G1 Z-23
N 4 G2 X75 Z-52.962 I37.5 K-29.962
N 5 G1 Z-73
N 6 G1 X110 Z-100
N 7 G1 Z-120
N 8 G1 X100 Z-143
N 9 G1 Z-148
N10 G0 X160 Z-100
N11 G80
N12 X150 Z300 M30

G83

Explanation

N 0	Starting conditions, tool 1.
N 1	Traverse to point S.
N 2	Begin cycle G83. The first contour point is programmed. Z already has the correct value and need therefore no longer be entered. Approach in X-direction, I = 6.
N 3 to N 9	Description of end contour. The first contour section is automatically extended by 2 mm by the determined Z coordinate of the starting point.
N10	Rapid traverse to point Q due to risk of collision.
N11	End of cycle.
N12	Tool change point, end of program.

G83

Cycle procedure
Traverse to starting point.
The control system recognizes that approach is in X only (address I) and adds the difference D to all programmed X-values.

$$D = \frac{øXS - øXA}{2} = \frac{116 - 96}{2} = 10$$

The first machining step is carried out using this "stock allowance".

In order to avoid collision, a rapid traverse to a point Q away from the contour is programmed at the end of the contour description.

In G80 the tool moves in rapid traverse to point S.
Approach by I = 6.

The "stock allowance" is therefore 10 - 6 = 4 mm

The second maching step is carried out.
The tool moves back to the start point of the second machining step.

The control system calculates the final approach.
In this example 4 mm remain.

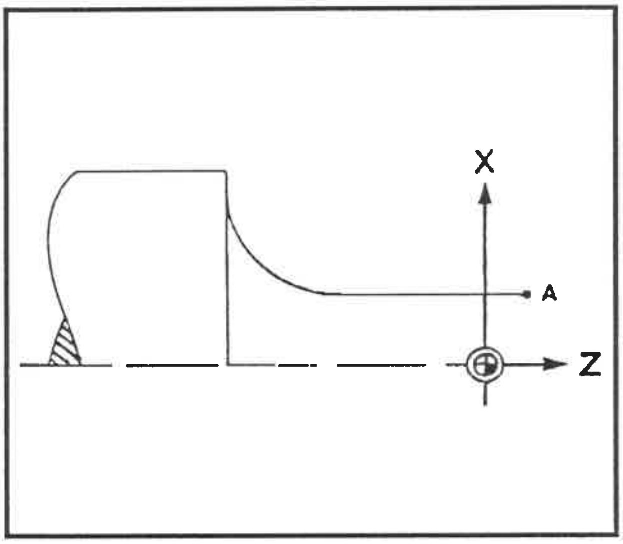
The end contour is turned.

The tool moves in rapid traverse to point A.

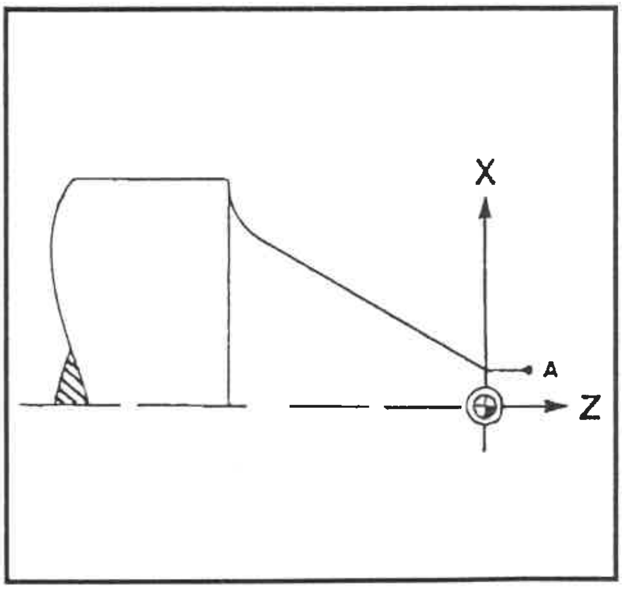
G83

Contour extension
In all G83 cycles, **approach** is in **rapid traverse**. This traverse path must lie outside the workpiece so that the approach does not coincide with the material. The contour to be programmed must be extended as follows.

Axis-parallel contour:
The first (axis-parallel) contour section is extended by 2 mm. No additional movement need be programmed.



Non-parallel contour:
An axis-parallel straight line must be programmed before the actual desired contour (length approx. 2 mm).

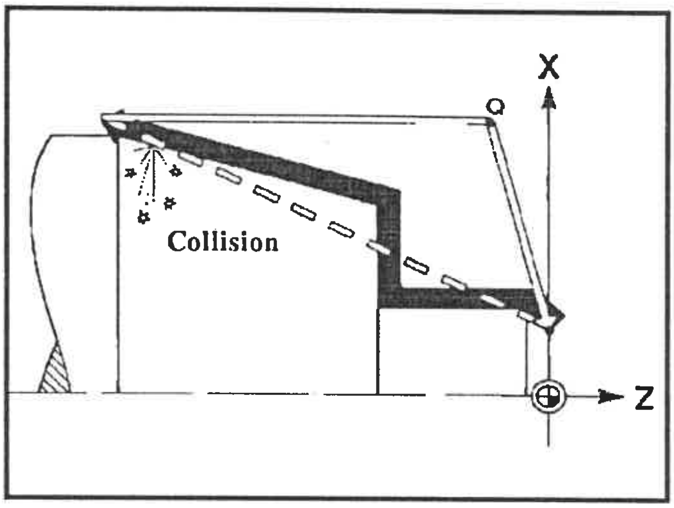


G83

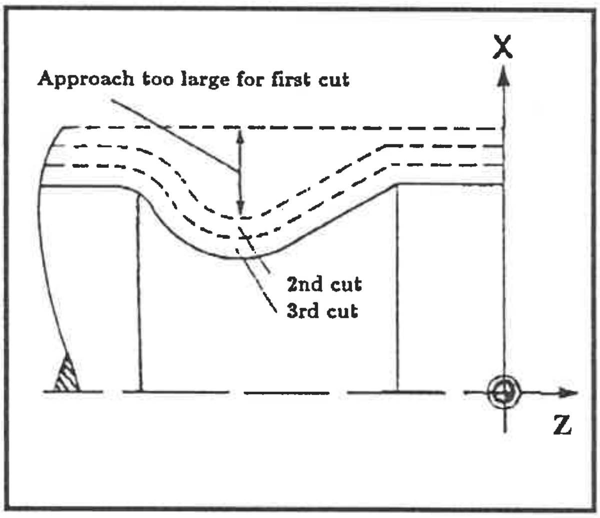
Leaving the contour
To avoid collision, the tool must in some cases follow a certain route when leaving the contour at the end of the cycle.
To do this, the following block is written after programming the contour:

N.. G0 X- and Z-coordinates of point Q.

Thus the tool moves in rapid traverse to point Q after machining the contour.

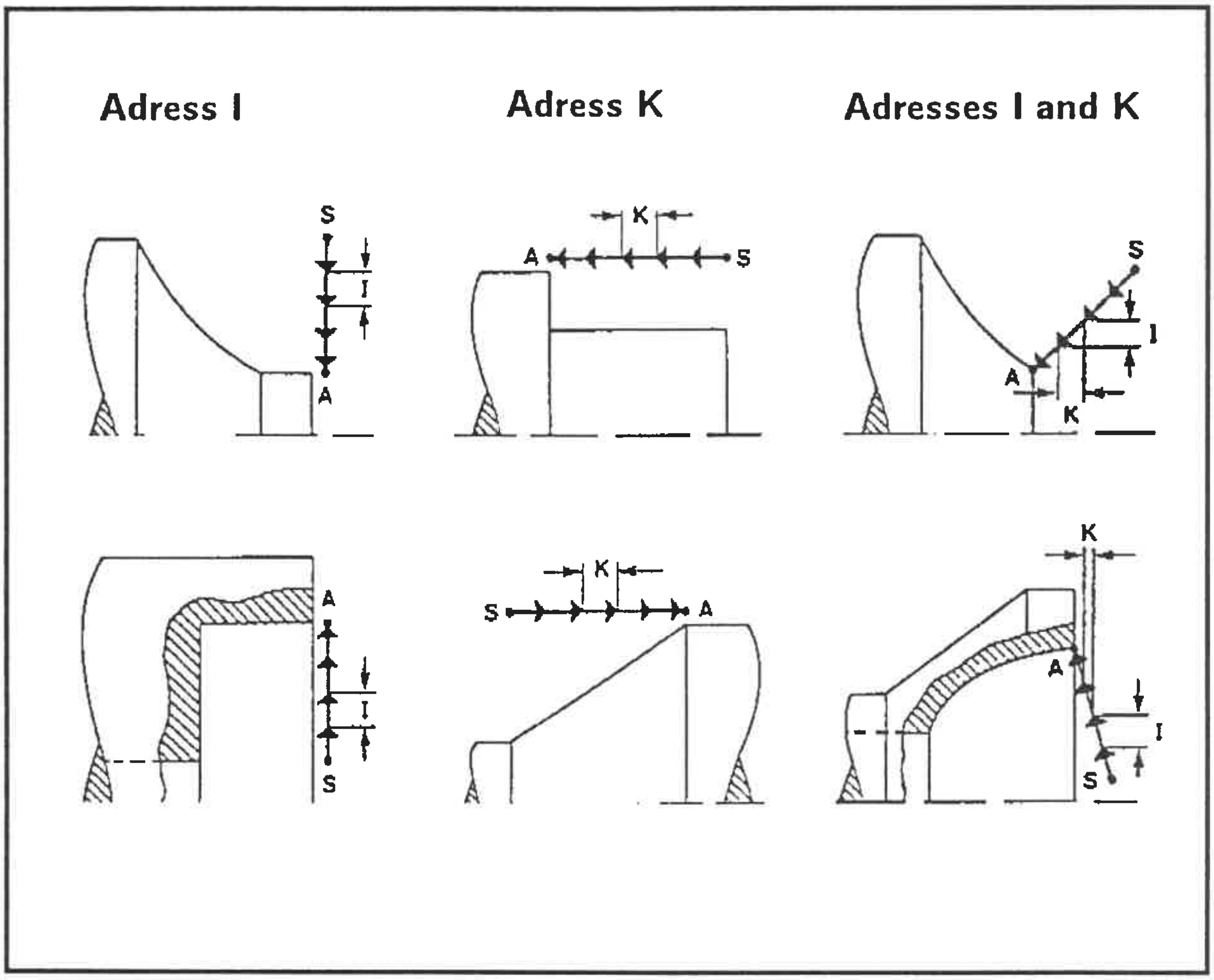


Falling contours
In falling contours, too much depth of cut may occasionally be given in some positions in the first cut.
Estimating these events can be facilitated by programming the contour in several individual sections.



G83

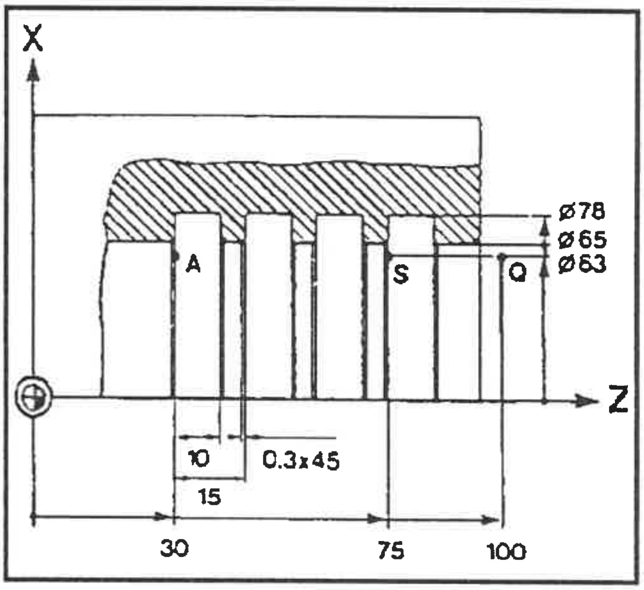
Approach movement/approach direction
The control system calculates the approach direction from starting point S and start point A of the contour. Addresses I and K are programmed without sign.



G83

G83 as multiple cycle
G83 can generally be used for multiple repetitions of contours, or cycles.
Here are a few examples to illustrate this.

Example 1
Grooves with multiple repetitions and cycle G86 in cycle G83.
Workpiece zero point has been placed at the end of the workpiece.



Programming

```
N0   G96  F0.15  S90  T6  M4  M7
N1   G0  X63  Z100
N2   G0 Z75
N3   G83  X63  Z30  K15
N4   G86  X78  Z30  I0.2  K10
N5   G80
N6   G0  Z100
N7   G0  X350  Z300  M30
```

Note
A = beginning of the contour for G86 (see here). In this case, A is the starting point for the last recess and must be programmed at G83.
S = Starting point for the first recess.

Explanation

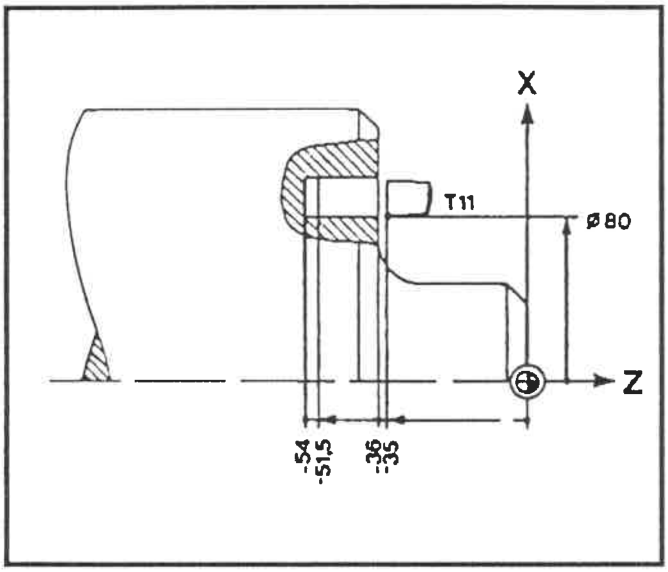
N0	Absolute dimensions, feed in mm/rev., tool data.
N1	Traverse to starting position above point Q (without Q there is **risk of collision**)
N2	
N3	Cycle data. The coordinates of the **last** machining cycle are programmed.
N4	Call recess cycle with chamfer and finishing cuts. The left inside corner of the groove must be entered. I = 0.2 determines stock allowance in X-direction, K10 is the width of the groove and F0.1 the feed. The chamfer of 0.3 is determined by the starting point in N3.
N5	End of cycle.
N6	After machining, the tool is positioned at X = 63, Z = 36. The tool runs in rapid traverse to Q.
N7	Traverse to tool change point, end of program.

G83

Example 2

Intermittent grooving for chip breaking

By programming G83, the tool runs further back along the path on which it has been cutting.



Programming

N0 G96 F0.12 S80 T11 M4 M7
N1 G0 X80 Z-35
N2 G83 Z-51.5 K2.5
N3 G1 Z-54
N4 G0 Z-51.5
N5 G80
N6 G0 Z-34
N7 G0 X200 Z300 M30

Explanation

N0 Starting conditions.

N1 Traverse to starting point, constant cutting speed.

N2 Cycle data. "First" point of contour, approach in Z-direction.

N3 Description of contour: Cutting along a straight line up to Z-54.

N4 Traverse back to "first" point.

N5 End of cycle.

N6 Rapid traverse out of groove.

N7 Traverse to tool change point, end of program.

Cycle procedure

- Traverse to starting point.

- Cut, run back in rapid traverse, approach once more by K.

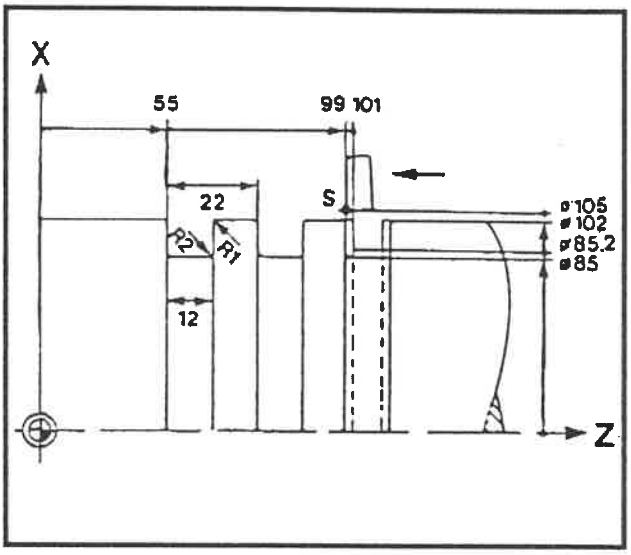
- This cycle is continued until the destination position given under G83 is reached.

- Traverse out of groove.

- Traverse to tool change point.

G83

Example 3
Grooves with several repetitions and changing tool definitions with the same tool. Workpiece zero point has been placed at the end of the workpiece in this example.
Please observe that there are only positive Z-values in this program.
The scratch point was the left-hand edge of the cut-off tool.

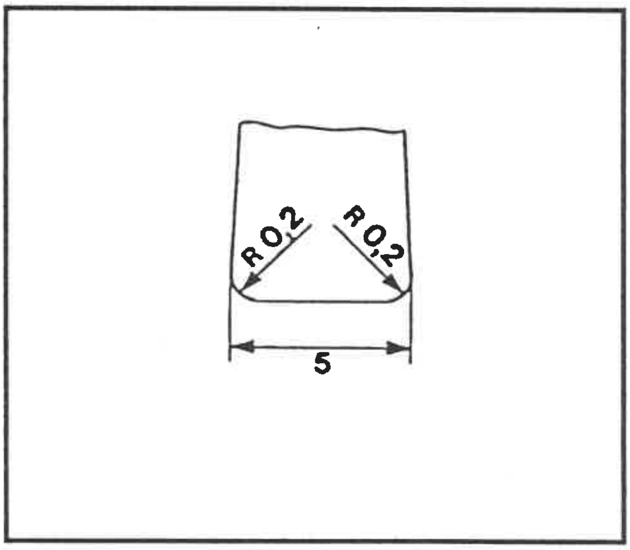


The **cut-off tool** (turret position 3) is used for three different tasks:

1. For finishing with the left-hand edge, tool no. T2103.
Here, the tool definition is
I = 0.2   K = 0.2

2. For finishing with the right-hand edge, tool no. T2203.
Here, the tool definition is
I = 0.2   K = -0.2

3. For recessing, tool no. T3.
Here, the tool definition is
I = 0.2   K = 5



Programming

```
N 0  G96  F0.12  S120  T3  M4  M8  M41
N 1  G0  X105  Z101
N 2  G83  Z57  K22
N 3  G86  X85.2  Z57  K8
N 4  T2103
N 5  G0  X102  Z50
N 6  G87  X102  Z55  I1
N 7  G87  X85  Z55 I2
N 8  G0  X105
N 9  T2203
N10  G0  X102  Z72
N11  G87  X102  Z67  I1
N12  G87  X85  Z67  I2
N13  G1  Z62  F0.5
N14  G0  X105
N15  T3
N16  G80
N17  G0  X200  Z300  M30
```

G83

Explanation

N0	The tool is called as a recessing tool.
N1	Rapid traverse to a point 1 mm above left-hand edge of first recess.
N2	Cycle call, last position in X and Z, offset or distance between two grooves is 22 mm.
N3	Recess cycle, the last groove is programmed, groove depth 8,2 mm (including stock allowance), groove width 8 mm.
N4	Call tool T2103 (finishing with left-hand edge), without tool change, i.e. tool remains, only new tool data and compensation take effect.
N5	Rapid traverse above left-hand edge of groove.
N6	Cycle radius, left-hand outside edge is rounded.
N7	Cycle radius, left-hand inside edge is rounded.
N8	Rapid traverse above groove.
N9	Call tool T2203 (finishing with right-hand edge) without change.
N10	Rapid traverse above right-hand edge of groove.
N11	Cycle radius, the right-hand outside edge is rounded.
N12	Cycle radius, the right-hand inside edge is rounded.
N13	Finishing base.
N14	Rapid traverse out of groove.
N15	Tool T3. Original data are effective once more.
N16	End of cycle.
N17	Traverse to tool change point, end of program.

G83

Cycle procedure

First, traverse to starting point.

The control system recognizes that in the cycle, a stock allowance must be made in Z (address K).
D = ZS - ZA = 44. In the "first run", this "allowance" is added to all Z-values, i.e. the groove at point Z = 57 + 44 = 101 is machined in accordance with the "contour description".

In G80, the initial machining point is engaged once more ("contour approach point"). In this case, 3 mm above the left-hand edge of the groove.

The second run with D = 22, i.e. 2nd groove is machined.

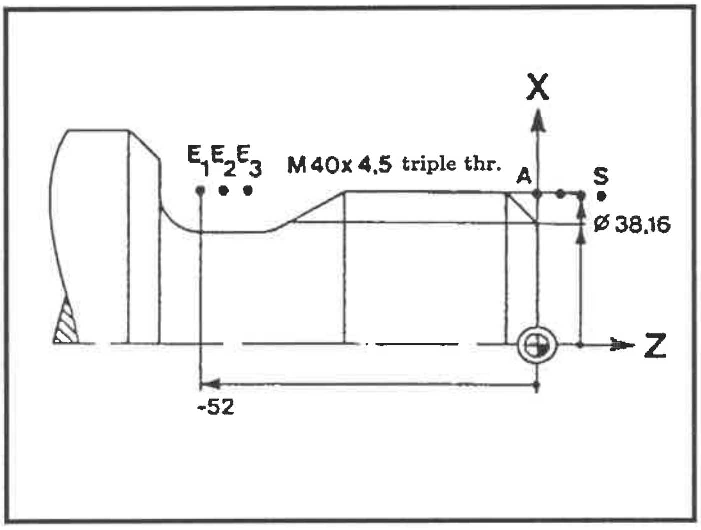
The third run with D = 0, i.e. the last groove is machined.

Traverse to tool change point.

G83

Example 4

Multiple start threading
G83 is used for repetition of the threading cut.



Programming

```
N0   G97  F0.5  S700  T9  M4  M7
N1   G0  X45  Z10
N2   G83  X45  Z13  K1.5
N3   G31  X40  Z-52  I0.24  K0.133
     P0.92  F4.5
N4   G80
N5   G0  X200  Z300  M30
```

Explanation

N0 Starting conditions.

N1 Traverse to starting point (X45, Z10).

N2 Multiple cycle data. Contour start point, approach in Z-direction.

N3 Threading cycle.

N4 End of multiple cycle.

N5 Traverse to tool change point, end of program.

Cycle procedure

- Traverse to starting point.

- The first thread is cut.

- Rapid traverse back to starting point. Main and feed spindles remain synchronized throughout the cycle.

- The control system effects 1.5 mm approach in Z-direction. Threading cut as above.

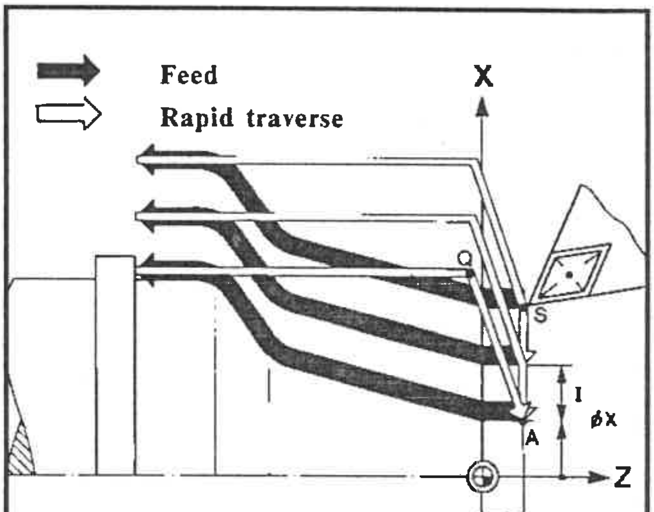
- The control system effects 1.5 mm approach in Z-direction, thus reaching the end point defined in N2. Last threading cut as above.

- Traverse to tool change point.

Note: This thread cannot be machined with G35, since F = 4.5 but P is only 0.92

Cycle contour G836

Programming for solid material CONTOUR PARALLEL cycle, G836
If a contour is to be turned from solid material using G83, the tool may run in the air without machining the workpiece.
This can be avoided by programming G836.
The next starting point cannot be engaged at the end of the cutting procedure (danger of collision!), so the point Q is determined in the cycle (see note).

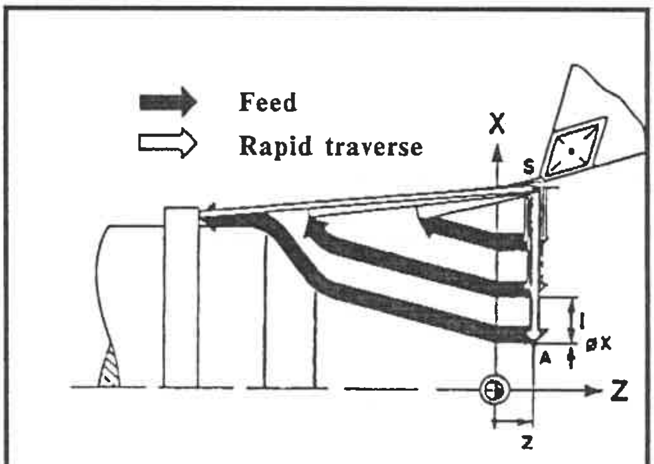


Traverse path with G83

Programming
N... G836 X... Z... I... K...

Movement sequence
In each cut, the control system checks whether the tool exceeds **one** coordinate of the starting point. If this is the case, the cut is terminated and the next approach made.

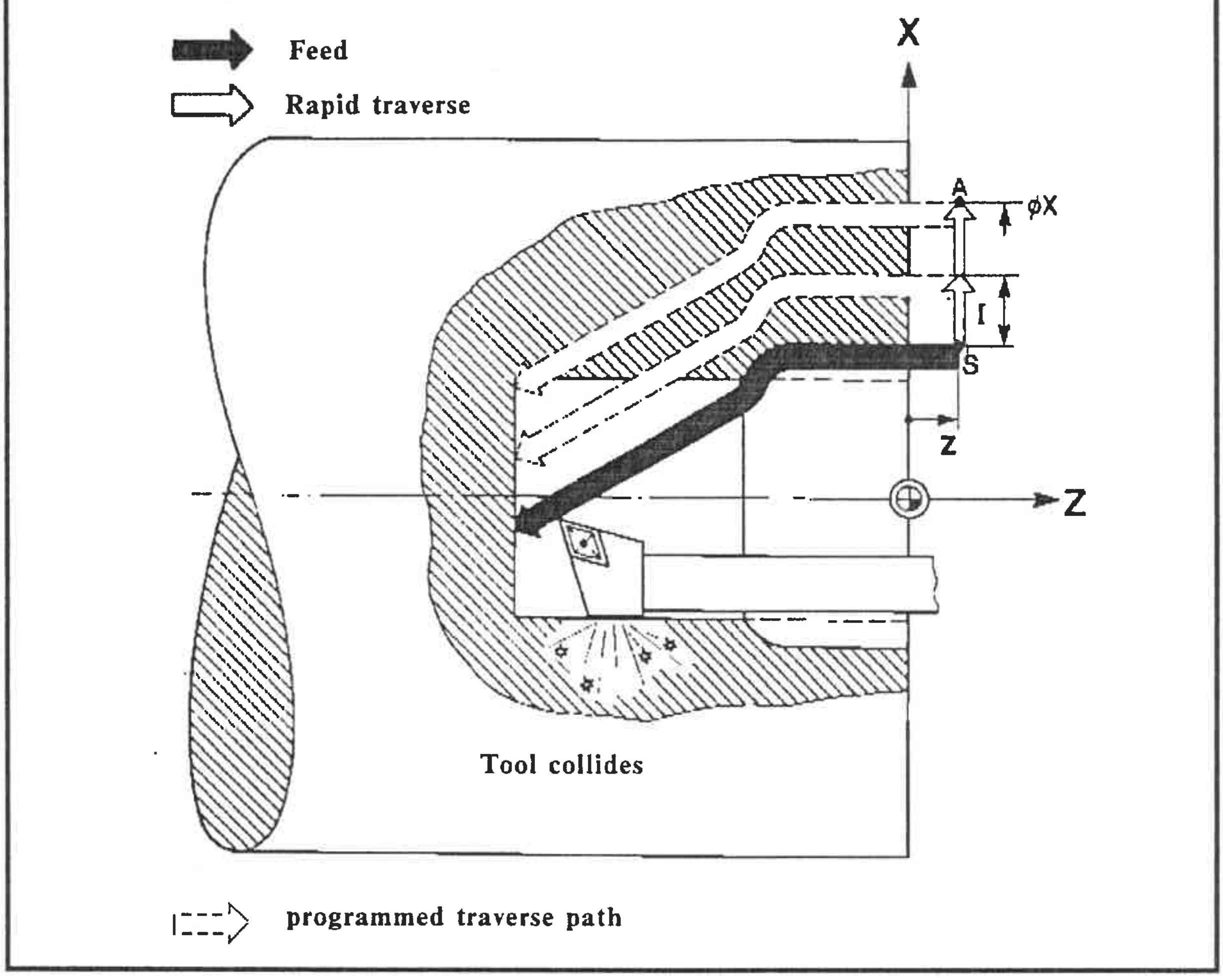
Otherwise as for G83.



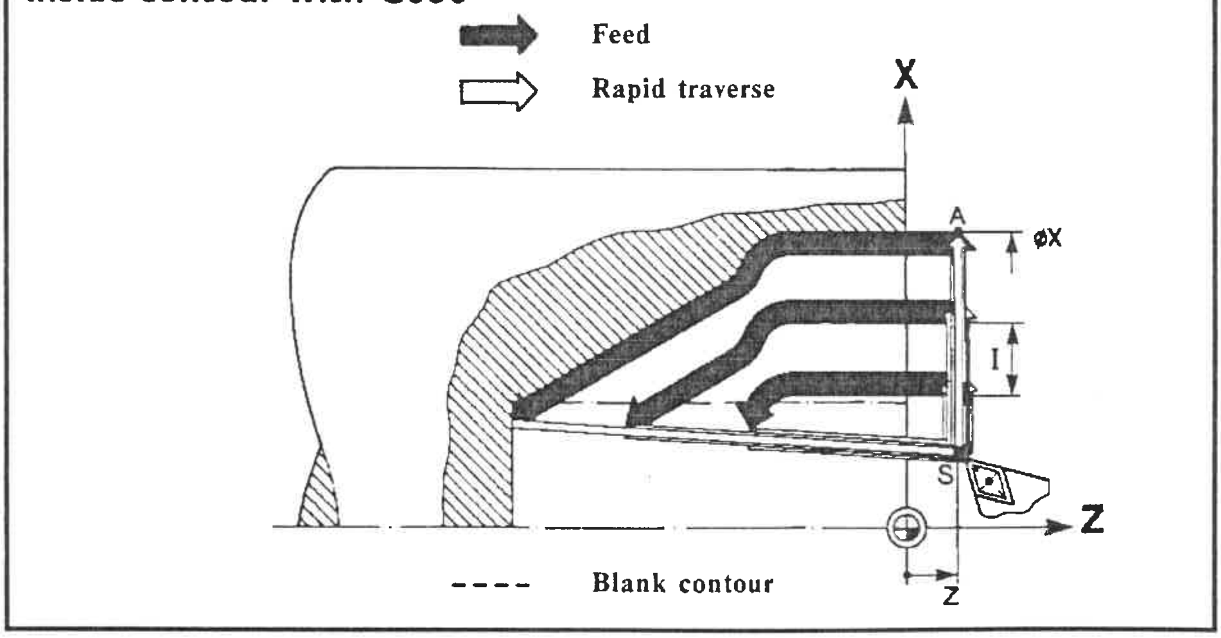
Traverse path with G836

G836

Inside contour with G83



Inside contour with G836

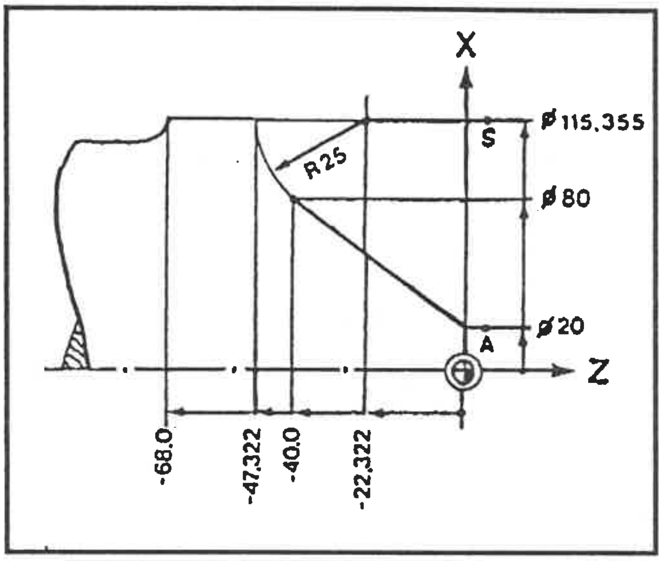


G836

Example

Programming

```
N 0  G96 F0.4  S150  T1  M4  M7
N 1  G0  X115.355  Z2
N 2  G836  X20  Z2  I5
N 3  G1  Z0
N 4  G1  X80  Z-40
N 5  G2  X115.355  Z-47.322  R25  B0
N 6  G80
N 7  G0  X200  Z20  M30
```



Explanation

N 0 Start conditions.

N 1 Rapid traverse to starting point.

N 2 Cutting cycle with limited traverse path, start point of the contour and cutting depth are determined.

N 3 Feed traverse along an imaginary contour.

N4,
N5 Programming the real contour.

N 6 End of cycle.

N 7 Rapid traverse return, end of program.

Cycle procedure

As traverse beyond the X coordinate is not possible, the tool approaches and cuts parallel to the contour until X once again reaches the value of the starting position.

G85

Undercut cycle E/F, DIN 76, G85
This cycle is used to produce: undercuts to DIN 509 form E, form F, with or without additional grinding, and thread clearance cuts to DIN 76.

Programming
The undercut always refers to right-angled, axis-parallel contours. For all undercut cycles, the undercut position (inside, outside, front and rear) must be programmed under Q or it must be recognizable from the tool dimensions I and K in the tool file or in G92.

If, for example, no tool is entered before G85 in subroutines, the position must be programmed with parameter Q:

Q1 = I+K+ Q2 = I-K+
Q3 = I+K- Q4 = I-K-
Confirm Q0 or Q = I and K of tool

Required addresses
After selecting G85, the control system requests the following inputs:

DIAMETER X:

LENGTH Z:

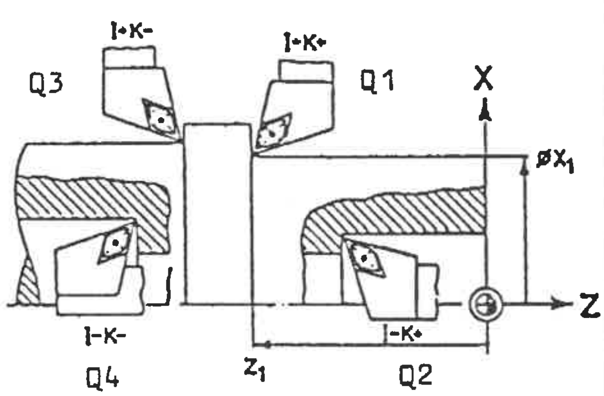
OUTSIDE = 1(3), INSIDE = 2(4) Q:

DEPTH,STOCK ALLOWANCE I:
Here, it is possible to program a grinding stock allowance for form E and F.
For thread undercuts the depth of the undercut must be programmed here.

WIDTH K:
No input for form E.
Input zero for form F.
For thread undercut: undercut width

SPECIAL FEED E:
for plunging straight lines and arcs

Sign of I and K



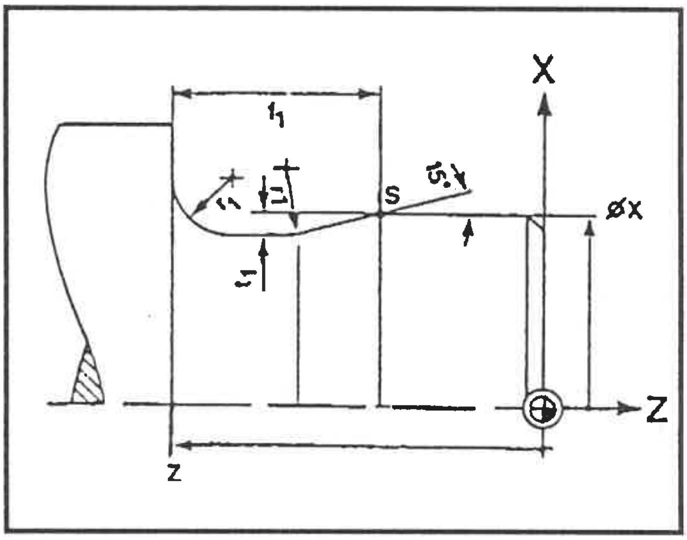
Cycle procedure
The tool traverses along a straight line to the contour intersection point.
At the end of the cycle, the tool is at

the end of the undercut movement.

G85

Undercut form E DIN 509, G85
Depending on diameter, the following dimensions are automatically observed:

X	r1	t1	f1
to 18	0.6	0.25	2
18 to 80	0.6	0.35	2.5
above 80	1	0.45	4



Required addresses
After selecting G85, the control system requests the following inputs:

DIAMETER X:

LENGTH Z:
Intersection point of the right-angled contours

OUTSIDE = 1(3), INSIDE = 2(4) Q:

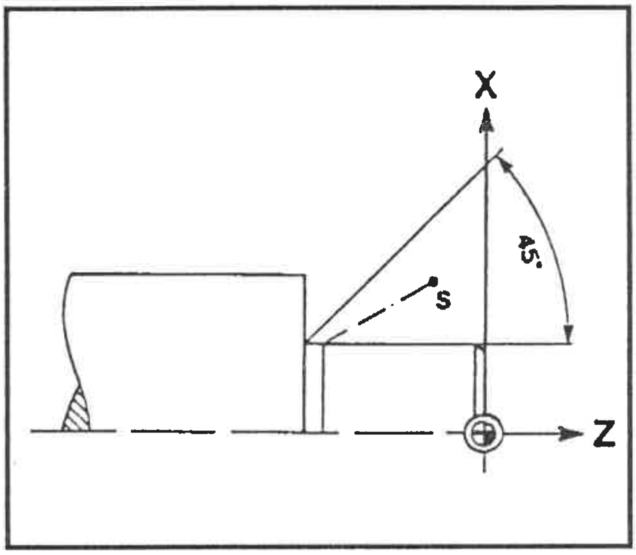
DEPTH, STOCK ALLOWANCE I:
Any stock allowance for grinding; entry for undercut depth is deleted.

WIDTH K:
not required

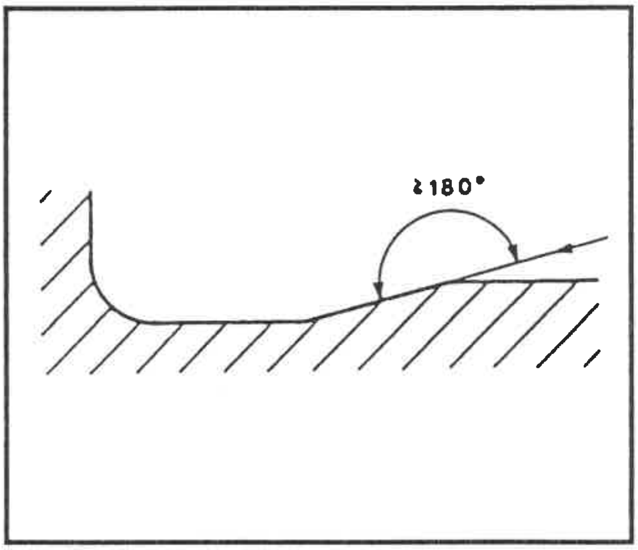
Cycle procedure
The undercut is made before the point programmed in X and Z is reached.
After the end of the cycle, the tool is at the end of the undercut contour (on the surface perpendicular to the approach direction).

G85

Separate undercut
The undercut can, of course, be machined separately.
The tool must first be brought into a position within a 45° angle to the adjoining contour.
Not possible with active SRK.



If SRK is being used in programming, the angle between the approach path and the following slanted section must be at least 180°.

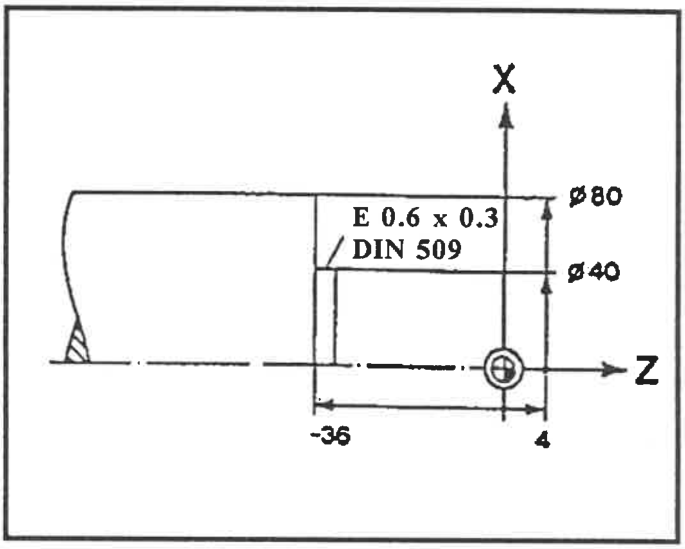


G85

Example
Machining a cylindrical stem with subsequent undercut to Form E DIN 509, and machining of the frontfaces:

Programming

```
...
N 12  G0   X40   Z4
N 13  G85  Z-36  Q1
N 14  G1   X80
...
```



Explanation
N12 Rapid traverse to starting point X40 Z4.

N13 Start cycle, corner point X40 Z-36.

N14 Straight line (linear interpolation) to X80.

Cycle procedure
- Rapid traverse to starting point S.

- Longitudinal movement up to corner point. X40 Z-33.5 is carried out.

- Undercut is made, tool is at end of frontface.

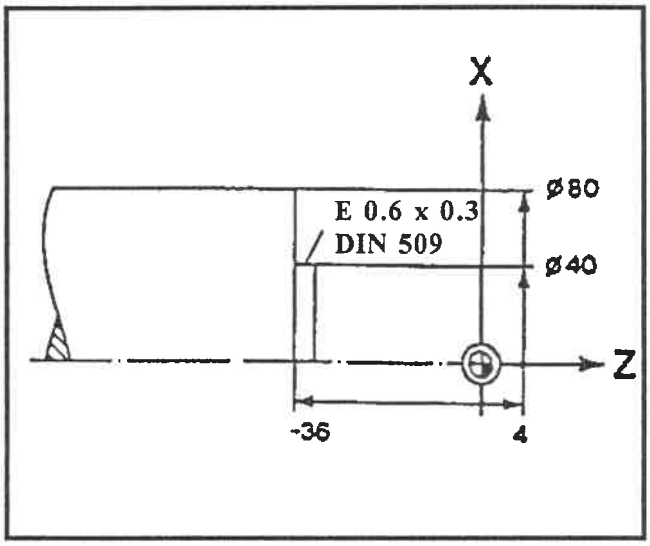
- Machine frontface.

G85

Example
Machining a cylindrical stem with grinding stock allowance 0.4 mm and subsequent undercut to Form E DIN 509, and machining the frontface.

Programming

```
...
N 12  G0   X40.8  Z4
N 13  G85  X40    Z-36  I0.4  Q1
N 14  G1   X80
...
```



Explanation

N12 The grinding stock allowance must be programmed at the starting point (as a diameter value!).

N13 In the "undercut" cycle, the corner point is programmed without stock allowance. The stock allowance is entered under address I (radius value!).

N14 Facing.

G85

Undercut Form F DIN 509, G85

Required addresses
After selecting G85, the control system requests the following inputs:

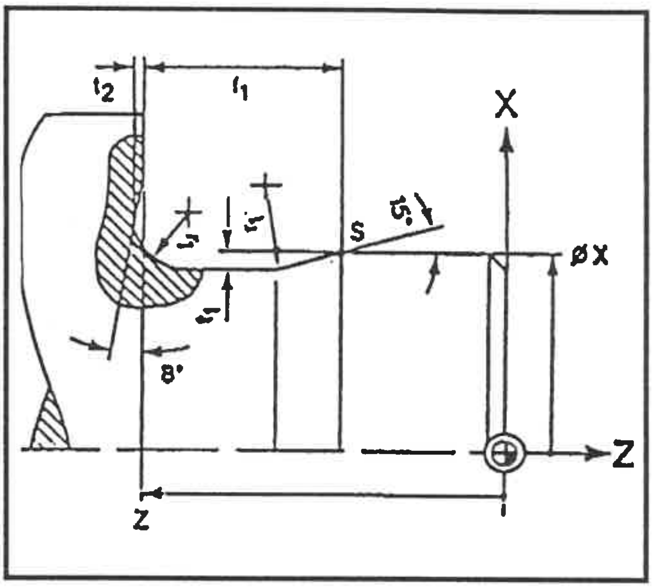
DIAMETER X:

LENGTH Z:
Intersection point of the right-angled contours.

OUTSIDE = 1(3), INSIDE = 2(4) Q:

DEPTH, STOCK ALLOWANCE I:
Any grinding stock allowance; input for undercut depth is deleted.

WIDTH K:
Input zero for form F



Depending on diameter X the following dimensions are automatically observed:

X	r1	t1	f1	t2
... 18	0,6	0,25	2	0,1
18 ... 80	0,6	0,35	2,5	0,2
80 ...	1	0,45	4	0,3

Note: The cutting radius of the tool must be smaller than the value entered under r1.
However, an error message appears only when SRK is active.

G85

Undercut form F DIN 509 with grinding stock allowance, G85
If a grinding stock allowance is to be included for diameter X, it can be programmed under I.

Required addresses
After selecting G85, the control system requests the following inputs:

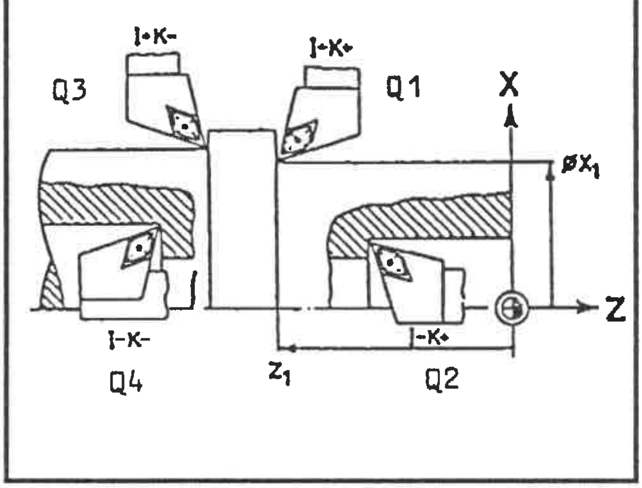
DIAMETER X:

LENGTH Z:
Intersection point of the right-angled contours.

OUTSIDE = 1(3), INSIDE = 2(4) Q:

DEPTH, STOCK ALLOWANCE I:
Grinding stock allowance

WIDTH K:
Here, zero must be entered



Sign of I and K

Programming
When traversing to the clearance cut, the overmeasure must be taken into consideration in the programmed X-value of the starting point S (**enter diameter** here)
The intersection point of the contour is programmed in the block with G85 without stock allowance.

Note:
In order to avoid inaccuracies, the programming of tool nose compensation (SRK) is useful, but not necessary.

G85

Thread undercut according to DIN 76, G85

Required addresses
After selecting G85, the control system requests the following inputs:

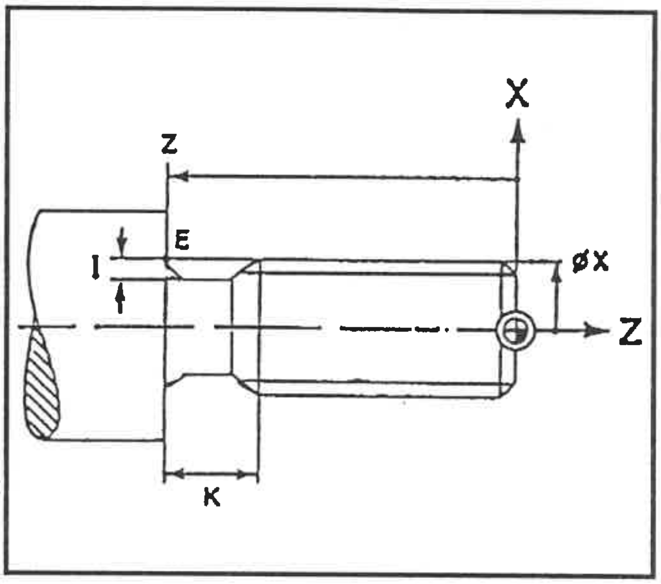
DIAMETER — X:

LENGTH — Z:
Corner point of the contour

OUTSIDE = 1(3), INSIDE = 2(4) — Q:

DEPTH; STOCK ALLOWANCE — I:
Undercut depth relative to the radius

WIDTH — K:
Undercut width

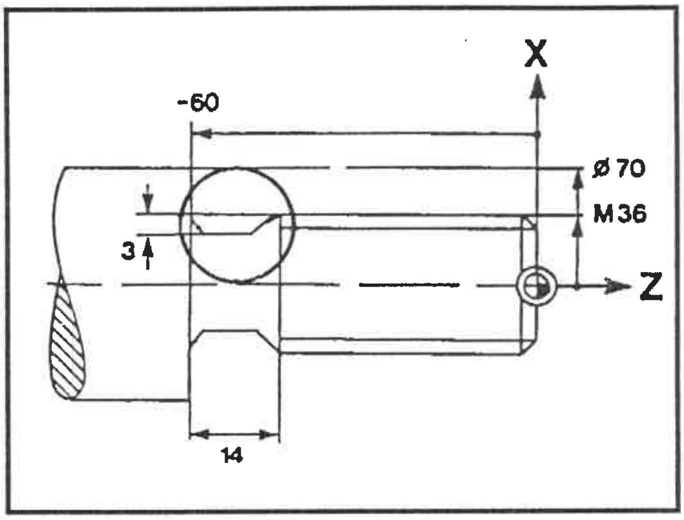


G85

Example
Machining a thread undercut to DIN 76

Programming

...
N12 G0 X36 Z5
N13 G85 Z-60 I3 K14 Q1
N14 G1 X75
...



Explanation

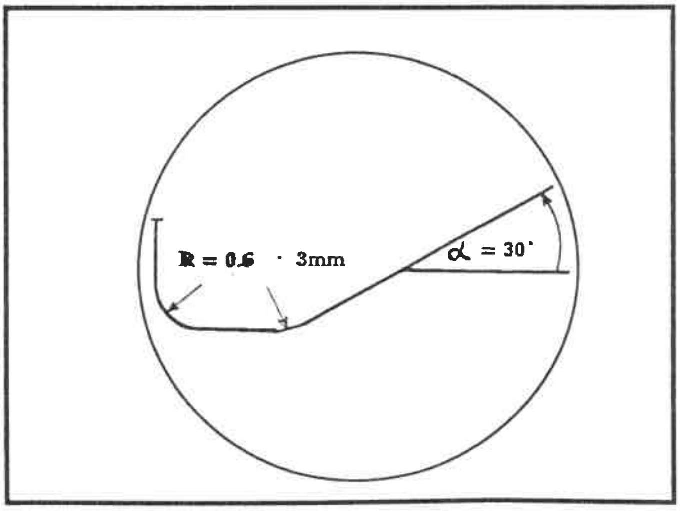
N12 Traverse to starting point.

N13 Call cycle, end of thread Z-60, depth I = 3 mm, width K = 14 mm.

N14 Straight line (linear interpolation).

Cycle procedure
- Rapid traverse to starting point S.
- The diameter of the entire threaded stem is machined up to the end point.
- Carry out undercut.
- Machine frontface.

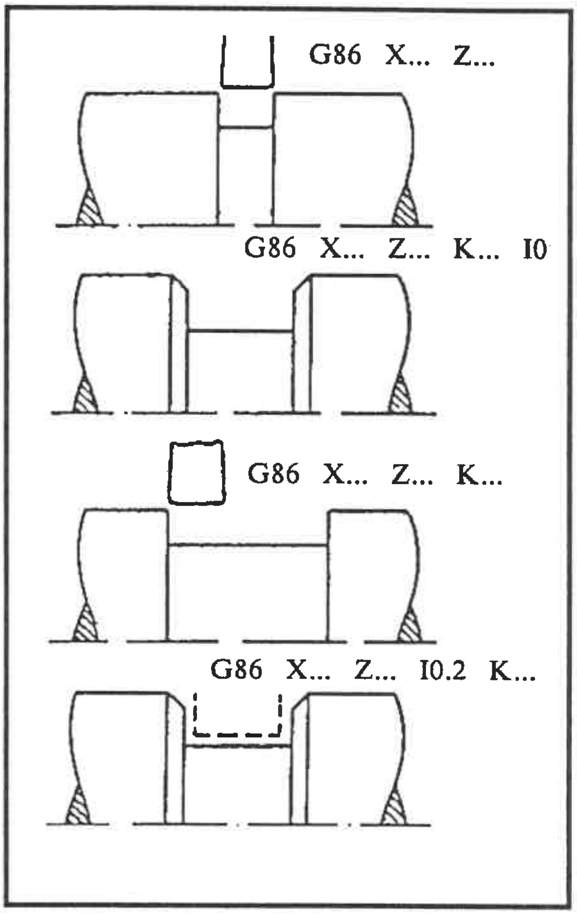
The radius R is calculated by multiplying the undercut depth by 0.6 ; the angle *a* is 30°.



Cycle grooving G86

Cycle GROOVING, G86
Cycle G86 can be used to machine grooves, for example for circlips to DIN 471 and DIN 472.

Recesses in radial direction



Possible forms
- Simple grooves.
  Recess width = tool width.

- Grooves with chamfers.
  Simple and wide recesses with chamfers on top edge.

- Grooves with several cuts.
  Groove wider than tool.

- Grooves with chamfer and finish cuts.
  Machining with stock and subsequent finishing.

Programming
Tool dimensions must be stored under X and Z.
The tool nose radius is stored under the address I, the cutting edge width under the address K.
As scratching is done with the left-hand edge of the tool, the dimensions stored under X and Z refer to the left-hand edge.

Machining direction
The grooves are always machined radially from the chuck side to the tailstock side. As alternatives, the grooving cycles G862 or G864 with freely definable contour are available for another machining direction.

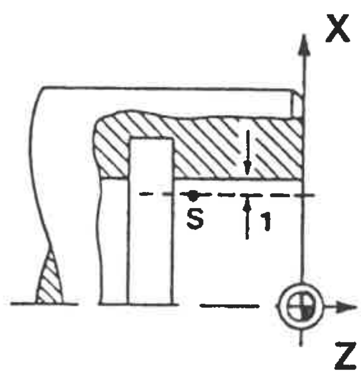
G86

Tool position before beginning of cycle

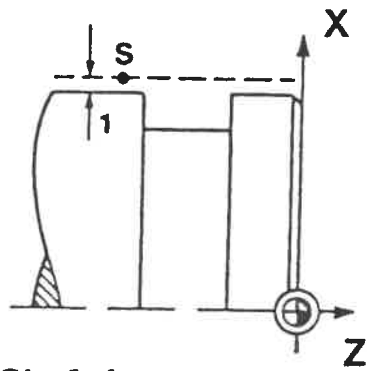
Transversal direction
Approx. 1mm before the workpiece, i.e. the shaft keyways, to a diameter approx. 2mm larger,

for drill hole grooves, to a diameter approx. 2 mm smaller.

Longitudinal direction
No exact positioning required.



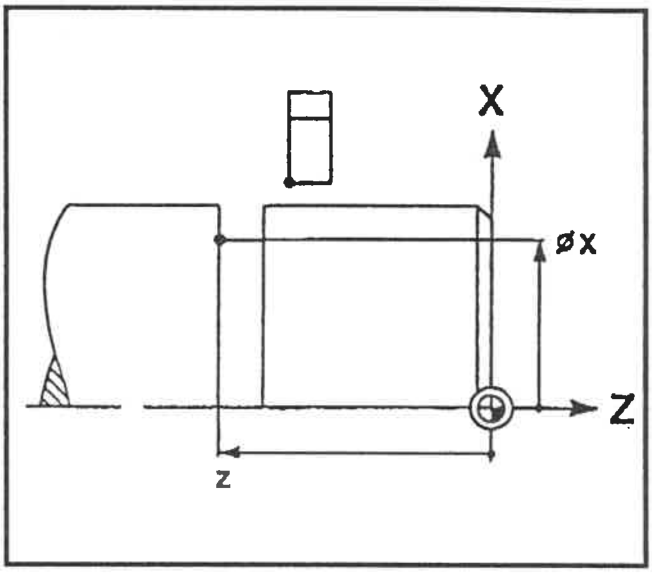
Drill hole groove



Shaft keyway

G86

Example
Simple recess with mit G86



Required addresses
After selecting G86, the control system requests the following inputs:

DIAMETER X:
left inside corner of groove

LENGTH Z:
left inside corner of groove

FINISHING STOCK ALLOW. I:
not required for simple groove

GROOVE WIDTH K:
not required for simple groove

PERIOD OF DWELL E:
If a finishing stock allowance is programmed under address I, then the control considers the value programmed under E as period of dwell in seconds at the recess corners to ensure that the corners are machined with high precision.
If no finishing stock allowance was programmed under address I, then the control treats the value programmed here as period of dwell (in seconds) for each recess, in order for the groove to be machined with high precision.
If address E is merely confirmed, the control system, by using the programmed feed, calculates a period of dwell exactly corresponding to one spindle rotation.

Cycle procedure
- Traverse in longitudinal direction to Z-position.

- Cut in up to programmed diameter.

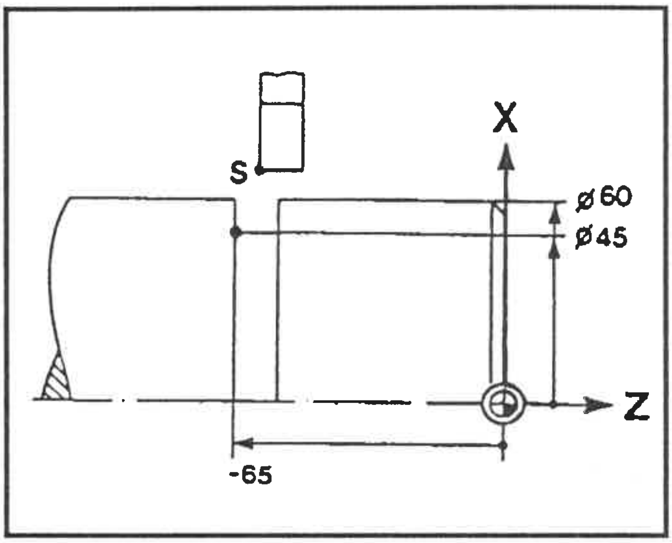
- Pull out in transversal direction up to start diameter.

- At the end of the cycle, the tool is directly above the recess.

G86

Example
Machining the illustrated groove
with tool width

Programming
N23 G0 X62 Z-64
N24 G86 X45 Z-65



Explanation

N23 Rapid traverse to starting position S.

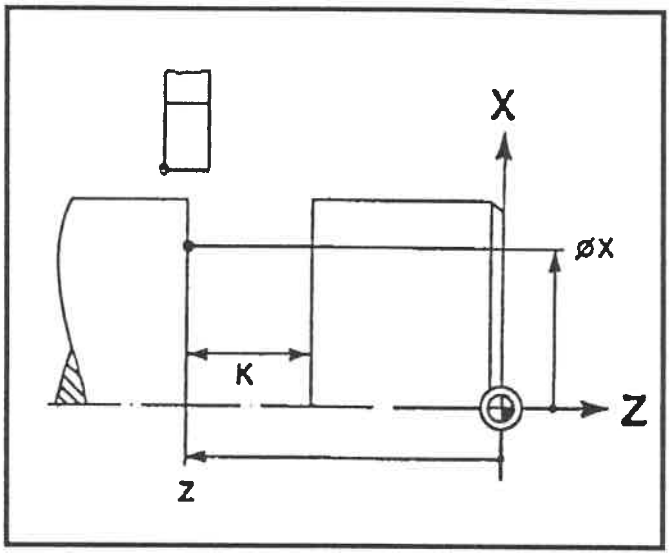
N24 Cycle recess, left inside corner of groove is programmed.

Cycle procedure:
- Rapid traverse to starting position S.
  Tool in Z above recess.
- Positioning.
- Groove to diameter X45.
- Pull out to diameter X62.

G86

**Groove with several cuts,
Cycle GROOVE, G86**

Required addresses
After selecting G86, the control system requests the following inputs:



DIAMETER X:
left inside corner of groove

LENGTH Z:
left inside corner of groove

FINISHING STOCK ALLOW. I:
not required

GROOVE WIDTH K:
Width of the groove.
K can also be entered as a negative value.
Then X and Z refer to the **right** inside corner of the groove. The contour is always machined from left to right, no matter whether addresses X and Z designate the left corner (K positive) of the right corner of the recess (K negative)

Note: The **left** bottom corner must be entered in the tool file, therefore **K** must be determined as a **positive** value in that place.

PERIOD OF DWELL E:
If a finishing stock allowance is programmed under address I, then the control considers the value programmed under E as period of dwell in seconds at the recess corners to ensure that the corners are machined with high precision.
If no finishing stock allowance was programmed under address I, then the control treats the value programmed here as period of dwell (in seconds) for each recess, in order for the groove to be machined with high precision.
If address E is merely confirmed, the control system, by using the programmed feed, calculates a period of dwell exactly corresponding to one spindle rotation.

Cycle procedure
- Traverse in longitudinal direction to the programmed Z-position.

- Cut in. (Machining direction from chuck side to tailstock side)

- Traverse in Z by 3/4 of the tool width.

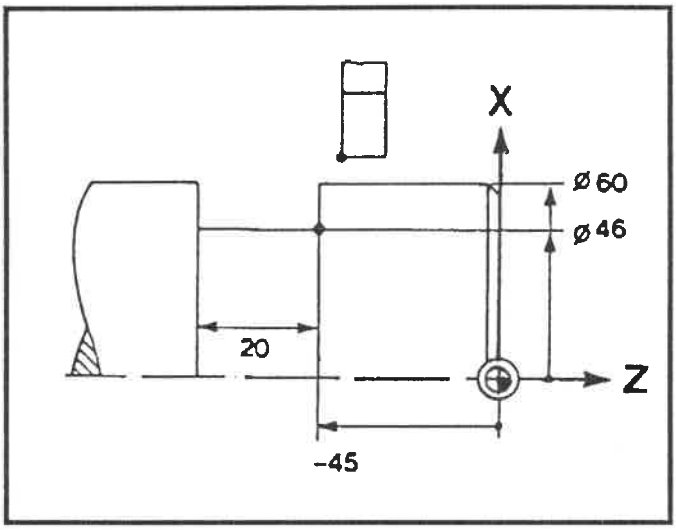
- Cut in.
etc.

The last traverse path in Z may be smaller than 3/4 of the tool width.

G86

Example
Groove is wider than tool:

Programming
N23 G0 X62 Z-42
N24 G86 X46 Z-45 K-20



Explanation

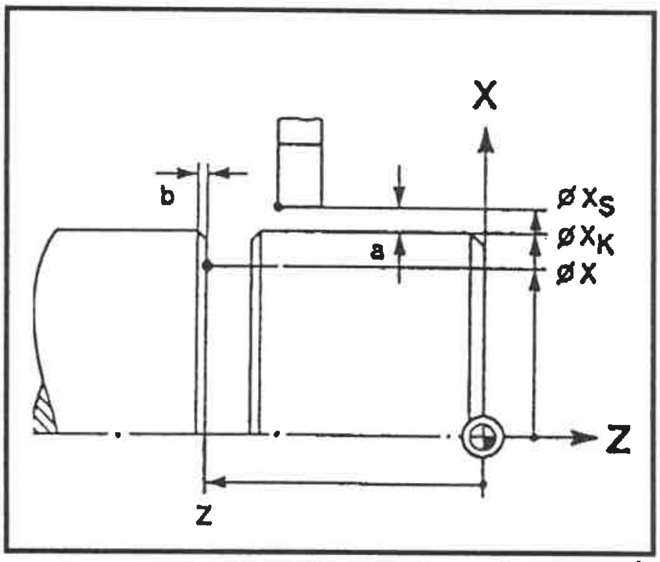
N23 Rapid traverse to starting position S.

N24 Cycle call, K is negative, so the right inside corner of the groove is programmed.

G86

Simple groove with chamfer,
Cycle GROOVE, G86
The width **b** of the chamfer depends on the distance **a** of the starting position:

a	b
mm	mm
1,3	0
1,2	0,1
1,1	0,2
1,0	0,3
0,9	0,4
0,8	0,5
0,7	0,6



Required addresses
After selecting G86, the control system requests the following inputs:

DIAMETER X:
left inside corner of groove

LENGTH Z:
left inside corner of groove

FINISHING STOCK I:
Enter zero

GROOVE WIDTH K:
not required

PERIOD OF DWELL E:
If a finishing stock allowance is programmed under address I, then the control considers the value programmed under E as period of dwell in seconds at the recess corners to ensure that the corners are machined with high precision.
If no finishing stock allowance was programmed under address I, then the control treats the value programmed here as period of dwell (in seconds) for each recess, in order for the groove to be machined with high precision.
If address E is merely confirmed, the control system, by using the programmed feed, calculates a period of dwell exactly corresponding to one spindle rotation.

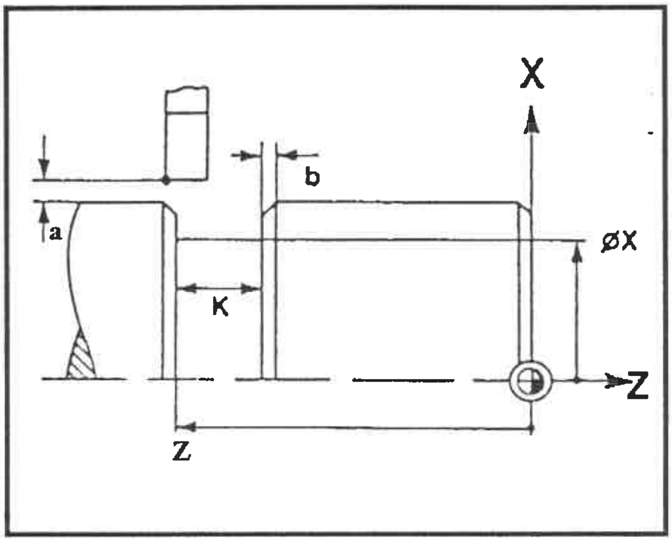
Starting position
The starting position is determined corresponding to the desired chamfer width **b** and the contour diameter X_K.
As **a** is entered as a radial value, 2 x **a** must be added to calculate the starting diameter.

$X_S = X_K + 2 \cdot a$

Cycle procedure
Entering I = 0 means that the groove edge will be provided with chamfers.
Otherwise as for simple groove, G86.

G86

Groove with chamfers,
Cycle GROOVE, G86
The width **b** of the chamfer is machined in accordance with the distance **a** of the starting position from the contour.



Required addresses
After selecting G86, the control system requests the following inputs:

DIAMETER X:
left inside corner of the groove

LENGTH Z:
left inside corner of the groove

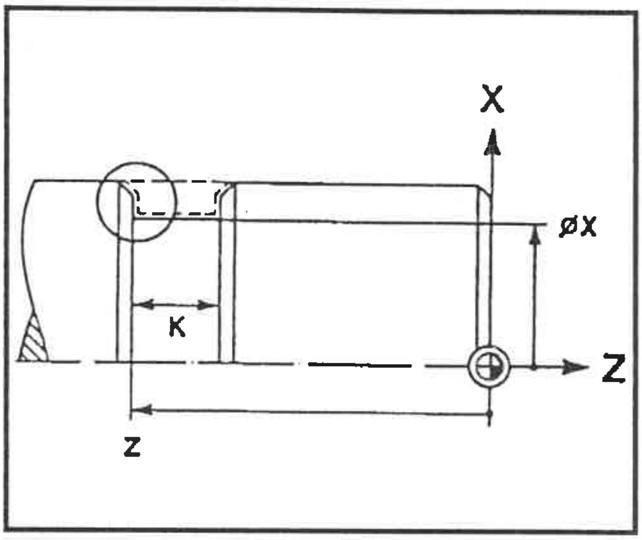
FINISHING STOCK I:
Enter zero

GROOVE WIDTH K:
Width of the groove (as for groove with several cuts).

PERIOD OF DWELL E:
If a finishing stock allowance is programmed under address I, then the control considers the value programmed under E as period of dwell in seconds at the recess corners to ensure that the corners are machined with high precision.
If no finishing stock allowance was programmed under address I, then the control treats the value programmed here as period of dwell (in seconds) for each recess, in order for the groove to be machined with high precision.
If address E is merely confirmed, the control system, by using the programmed feed, calculates a period of dwell exactly corresponding to one spindle rotation.

G86

Groove with chamfers, roughing and finishing,
Cycle GROOVE, G86
For the chamfered groove, an additional finishing stock allowance can be included on the sides and bottom.



Required addresses
After selecting G86, the control system requests the following inputs:

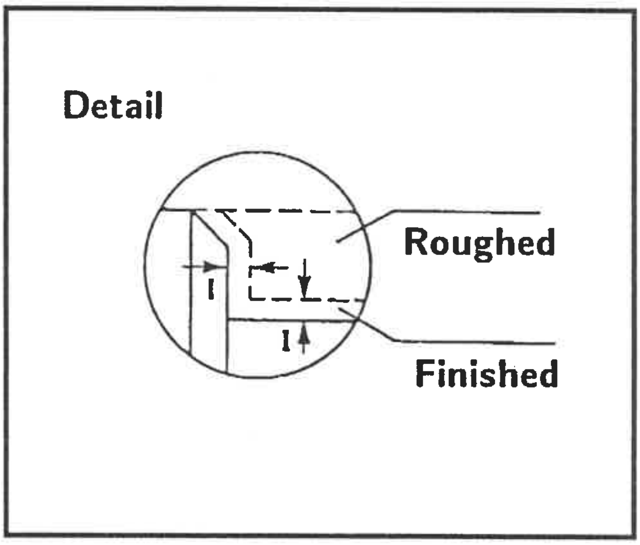
DIAMETER X:
left inside corner of groove

LENGTH Z:
left inside corner of groove

FINISHING STOCK ALLOW. I:
As desired

GROOVE WIDTH K:
Width of the groove

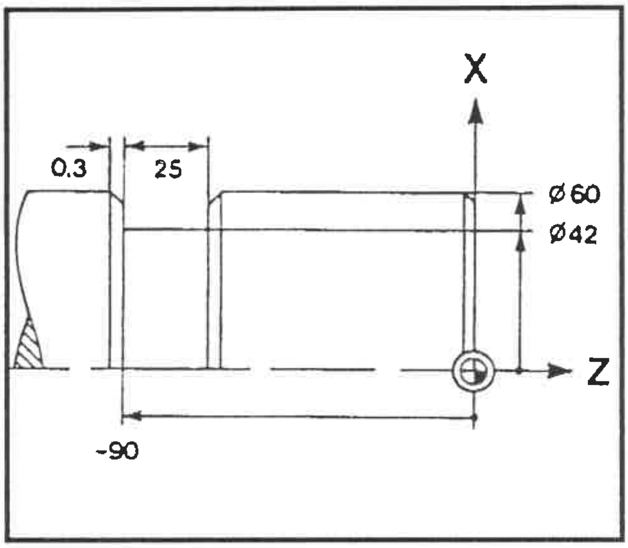
PERIOD OF DWELL E:
If a finishing stock allowance is programmed under address I, then the control considers the value programmed under E as period of dwell in seconds at the recess corners to ensure that the corners are machined with high precision.
If no finishing stock allowance was programmed under address I, then the control treats the value programmed here as period of dwell (in seconds) for each recess, in order for the groove to be machined with high precision.
If address E is merely confirmed, the control system, by using the programmed feed, calculates a period of dwell exactly corresponding to one spindle rotation.



Cycle procedure
First, roughing is done with stock allowance, then finishing. Finishing is always carried out from the outside to the center.

G86

Example
The groove is wider than the tool.
Finishing stock is 0.5 mm and chamfer is 0.3 mm wide. Rough and finish.



Programming
N27 G0 X62 Z-63
N28 G86 X42 Z-65 I0.5 K-25

Explanation

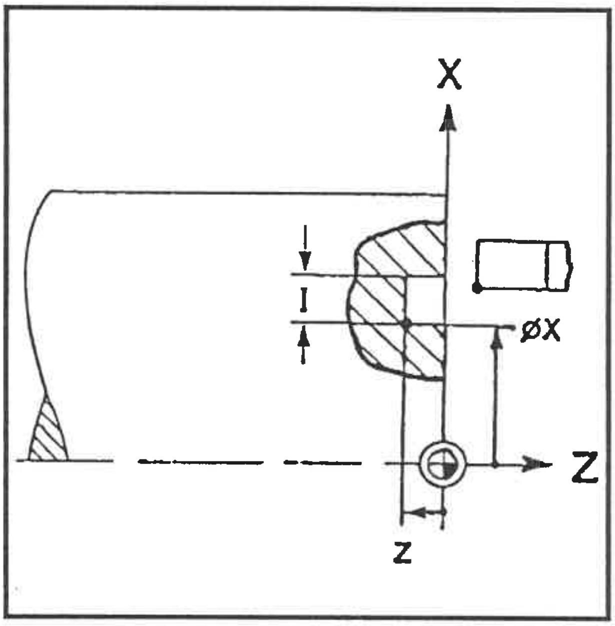
N27 Rapid traverse to starting position S.
Distance a from table 1.0 for chamfer width b of 0.3 mm.
$X_S = 60 + 2 \cdot 1.0 = 62$

N28 Call cycle, programming of **right** inside corner as K is negative; 0.5 mm stock allowance.

G86

Grooves in axial direction
Cycle GROOVE, G86
All groove cycles described so far can also be done in axial direction.
To do this, the tool width in the tool store must be programmed with negative sign.

Required addresses
After selecting G86, the control system requests the following inputs:



DIAMETER X:
Inside corner, in negative direction, of the groove

LENGTH Z:
Groove depth

FINISHING STOCK ALLOW. I:
Finishing stock allowance for chamfered groove

GROOVE WIDTH K:
Groove width

PERIOD OF DWELL E:
If a finishing stock allowance is programmed under address I, then the control considers the value programmed under E as period of dwell in seconds at the recess corners to ensure that the corners are machined with high precision.
If no finishing stock allowance was programmed under address I, then the control treats the value programmed here as period of dwell (in seconds) for each recess, in order for the groove to be machined with high precision.
If address E is merely confirmed, the control system, by using the programmed feed, calculates a period of dwell exactly corresponding to one spindle rotation.

Programming
N... G86 X... Z... (E...)

Machining direction
The grooves are always machined axially from the inside to the outside. As alternatives, the grooving cycles G861 or G863 with freely definable contour are available for another machining direction.

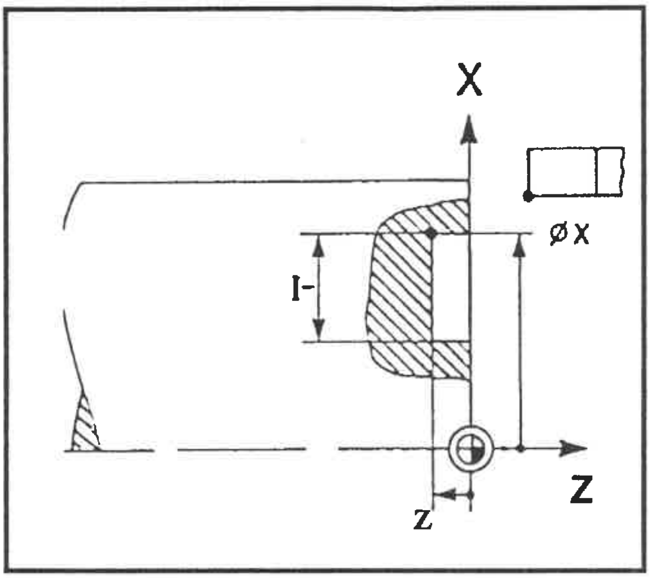
G86

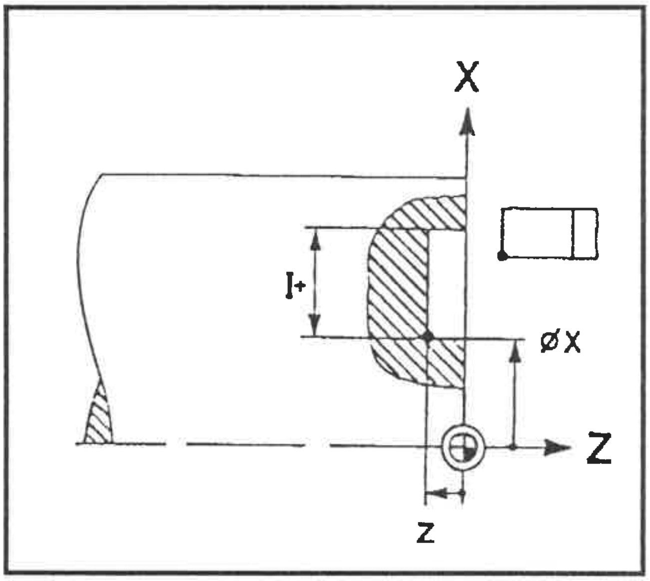
Axial groove with roughing and finishing:

I positive: The inside corner (in negative direction) of the groove is programmed (see fig.).

I negative: The inside corner (in positive direction) of the groove is programmed.

The contour is always machined from bottom to top, no matter whether addresses X and Z designate the recess width at the bottom (I positive) or at the top (I negative).





Axial groove G86 with chamfers
The width **b** of the chamfer is machined according to the distance **a** from the starting position.

Programming
Groove width equal to tool width:

N ... G86 X... Z... K0 (E...)

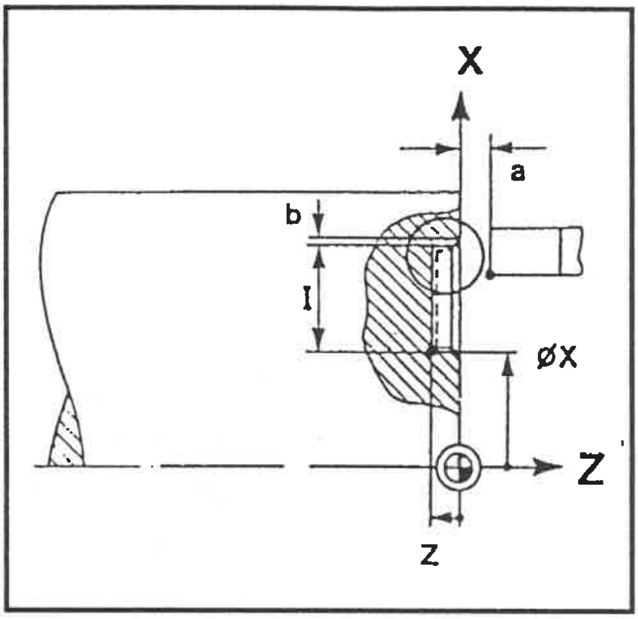
Groove wider than tool:

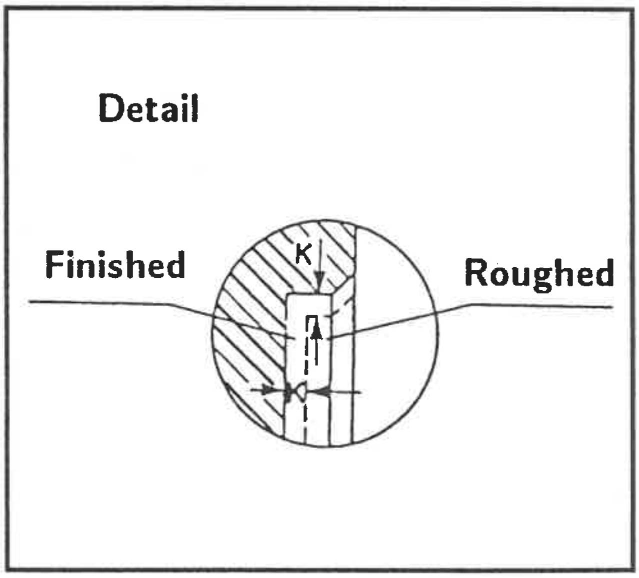
N ... G86 X... Z... I... K0 (E...)

G86

Axial groove G86 with chamfers and finish cuts

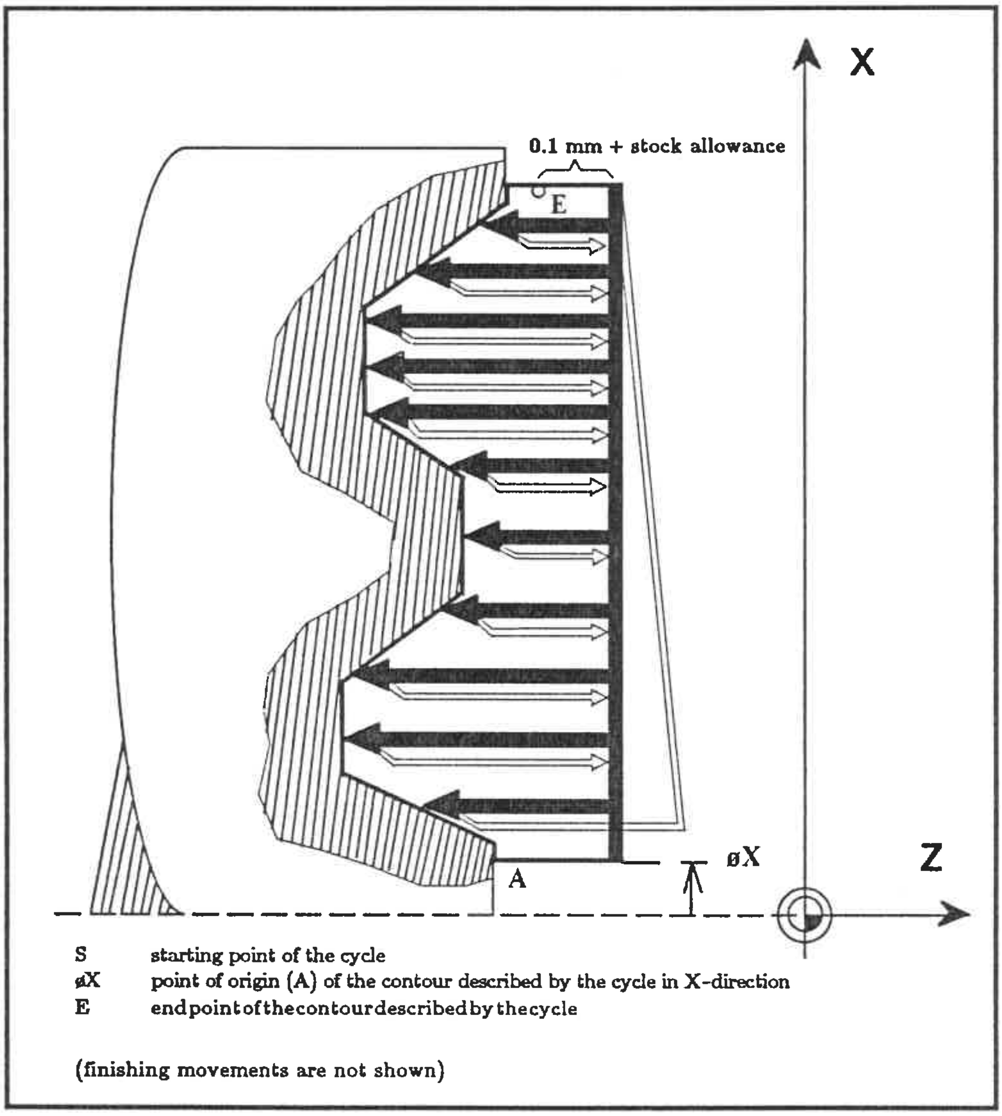
Programming
N... G86 X... Z... I... K... (E...)





Groove with contour (transversal) G861

Groove with contour (transversal), G861
G861 can be utilized to make a groove with freely programmable contour in the front face of the workpiece. This contour can have a maximum of 9 "valleys".



Required addresses
After selecting G861, the control system requests the following entries:

DIAMETER X:
X-coordinate at which the programmed contour of the groove begins (A).

G861

Important
Only the tool type "cutoff tool for axial machining" may be used in this cycle, that is WT21 or WT0.

Programming
When programming the contour, **keep in mind** that every hill in the contour (even limit elements) must consist of one path increment (G1) parallel to the axis.
The lowest point of the valleys (groove base) should also consist of path increments parallel to the axis (not a requirement). If the groove base consists of one path increment (bevel or radius) not parallel to the axis, the control system will automatically replace these path increments which are not parallel to the axis by a path increment which is parallel to the axis, since the tool cannot machine a bevel or radii in the groove base. In such a case the operator is warned "Remaining material not removed" to inform him that a programmed bevel or radius in the groove base has not been machined.
The contour is adapted to the chisel width so that the contour can be machined with different chisel widths, thus ensuring the best processing of the contour when the base of the groove was not programmed to be parallel to the axis by the operator.

The blocks following the cycle activation with G861 contain the description of the contour to which machining will be performed.
The approach to the contour must be programmed in the first NC block after the cycle activation, since only one approach path can be programmed (G0, G1 already go directly to the contour).
Furthermore this first NC block after the cycle activation must contain the tool nose radius compensation.
All following blocks are the contour description which is not to contain more than 80 NC blocks.
The cycle end must be programmed afterwards in a separate block.

If the workpiece is not to be machined to the finished size during this cycle, i.e. if further machining is planned with the cycle G863 for example, a stock allowance parallel to the contour must be programmed with G58 (see the programming example for G863).

The starting point of the cycle (S) and the end point (E) of the contour described by this cycle **must** have the same **X-coordinate**. The starting point in the **Z-direction** must be at least the minimum amount of 0.1 mm plus a programmed stock allowance away and to the right of the end of the contour.

Note
Both the left-hand and right-hand cutting nose tips are used for machining the contour. The active cutting nose tip is reversed at the boundary between two valleys. Consequently there is a line missing in the graphic simulation which corresponds precisely to a tool width since it is always the course of the left part of the chisel tip which is represented in the graphic simulation.

G861

Cycle sequence

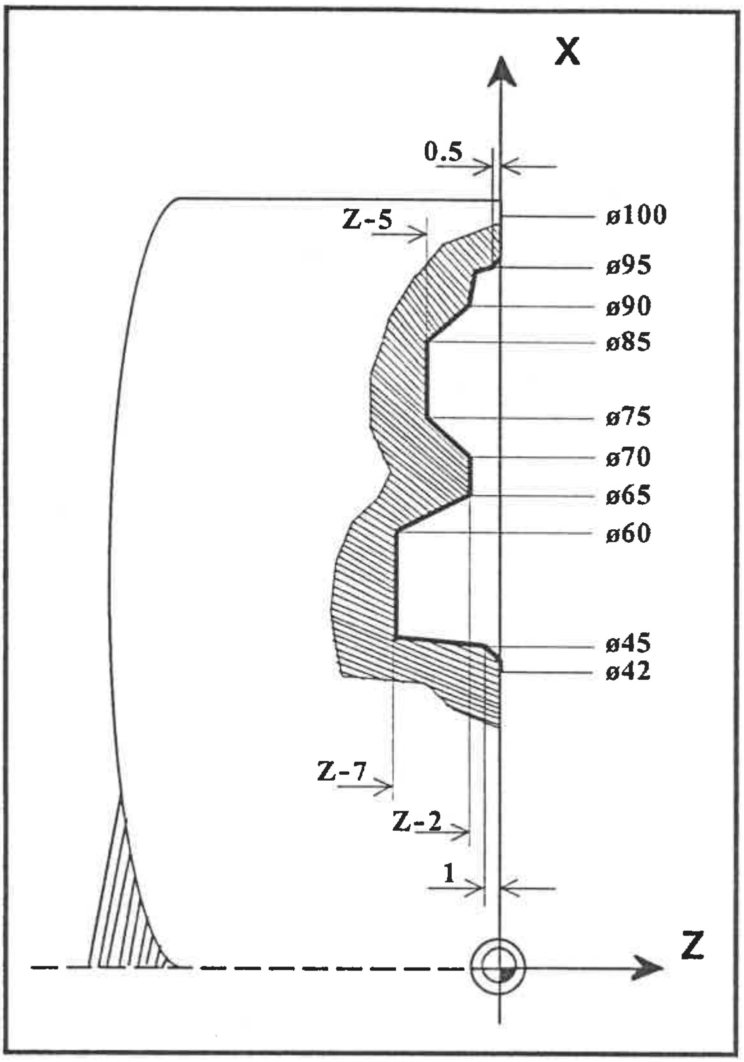
First the programmed contour is rough machined, i.e. each valley is recessed in order. The feed rate during this process is 0.75 times the tool width for each feed movement.
An automatic rest cut indexing with feed movements less than **0.75 times the tool width** ensures that each groove base is machined, irrespective of the position of the starting point. Where a number of valleys are involved, an automatic rest cut indexing is performed for each valley.
After each valley is machined, the control system automatically performs a finishing operation along the programmed contour of each valley.
Each of these finishing operations is conducted in two steps.
When all the machining is complete the tool is again at the starting point (S).
The contour is machined from the starting point (S) of the cycle to the point of origin (A) of the contour described by the cycle. The contour description program blocks have exactly the opposite sequence.

G861

Example
The contour (contour contains two valleys) is described in subprogram %8651.

Main program %2006
```
N 1  G96  S100  G94  F120  T3  M4
N 2  G0   X100  Z2
N 3  G861  X42
N 4  L8651
N 5  G80
N 6  G0  X100  Z2
N 7  M30
```

Subprogram %8651
```
N 1  G42  G0  X42  Z1
N 2  G1  Z0
N 3  G1  X45  B-1
N 4  G1  X?  Z-7  A5  B1
N 5  G1  X60  B1
N 6  G1  X65  Z-2  B1
N 7  G1  X70  B1
N 8  G1  X75  Z-5  B1
N 9  G1  X85  B1
N10  G1  X90  Z-2  B1
N11  G1  X95  Z?  A95  B1
N12  G1  X? Z0  A177  B-0.5
N13  G1  X100
N14  G40  G1  Z1
N15  M30
```



Explanation (main program)

N 1	constant cutting speed; feed in [mm/min]; tool file T3 turning direction CCW
N 2	rapid traverse to the cycle starting point (S)
N 3	cycle grooving (transversal); point of origin (A) of the groove contour in X-direction
N 4	subprogram activation (contour description of cycle)
N 5	cycle end
N 6	rapid traverse movement
N 7	program end

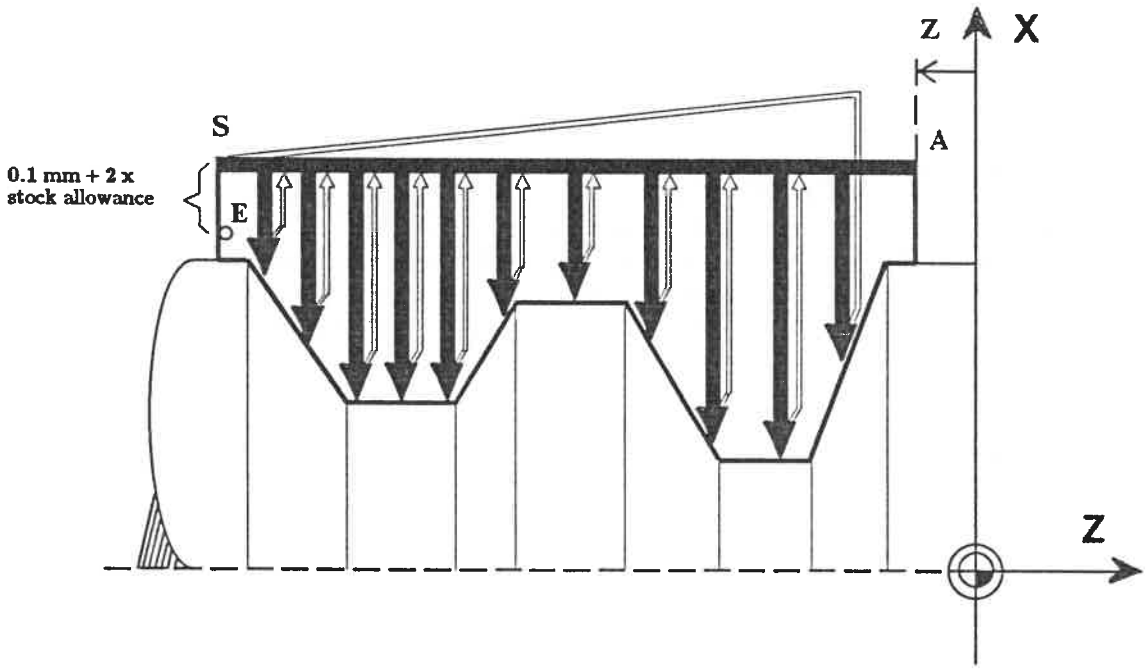
G861

Explanation (subprogram)

N 1	approach to the point of origin (A) of the contour in rapid traverse; activation of the SRK
N 2	straight line
N 3	straight line with chamfer (1 mm)
N 4	straight line with curve (R1) at an angle of 5o
N 5	straight line with curve (R1)
N 6	straight line with curve (R1)
N 7	straight line with curve (R1)
N 8	straight line with curve (R1)
N 9	straight line with curve (R1)
N10	straight line with curve (R1)
N11	straight line with curve (R1) at an angle of 95°
N12	straight line with chamfer (0.5 mm) at an angle of 177°
N13	straight line to starting point (S) of cycle
N14	straight line; deactivation of SRK
N15	subprogram end

Groove with contour (longitudinal) G862

Groove with contour (longitudinal), G862
G862 can be utilized to create a groove with freely programmable contour on the circumference of the workpiece. This contour can contain a maximum of 9 "valleys"



S starting point of the cycle
Z point of origin (A) of the contour described by the cycle in Z-direction
E endpoint of the contour described by the cycle

(finishing movements are not shown)

Required addresses
After G862 is selected, the control system requests the following entries:

LENGTH Z:
Z-coordinate at which the programmed contour of the groove begins (A).

G862

Important
Only the tool type "cutoff tool for machinings in radial direction" may be used in this cycle. Since machining in radial direction can be performed both from outside and from the inside, the tool type must be chosen accordingly. For external machinings WT3 or WT0 are available. Internal machining can only be performed with WT8.

Programming
When programming the contour, keep in mind that every hill in the contour (even limit elements) must consist of one path increment (G1) parallel to the axis.
The lowest point of the valleys (groove base) should also consist of path increments parallel to the axis (not a requirement). If the groove base consists of one path increment (bevel or radius) not parallel to the axis, the control system will automatically replace these path increments which are not parallel to the axis by a path increment which is parallel to the axis, since the tool cannot machine a bevel or radii in the groove base. In such a case the operator is warned "Remaining material not removed" to inform him that a programmed bevel or radius in the groove base has not been machined. The contour is adapted to the chisel width so that the contour can be machined with different chisel widths, thus ensuring the best processing of the contour when the base of the groove was not programmed to be parallel to the axis by the operator.

The blocks following the cycle activation with G861 contain the description of the contour to which machining will be performed.
The approach to the contour must be programmed in the first NC block after the cycle activation, since only one approach path can be programmed (G0, G1 already go directly to the contour).
Furthermore this first NC block after the cycle activation must contain the tool nose radius compensation.
All following blocks are the contour description which is not to contain more than 80 NC blocks.
The cycle end must be programmed afterwards in a separate block.
If the workpiece is not to be machined to the finished size during this cycle, i.e. if further machining is planned with the cycle G864 for example, a stock allowance parallel to the contour must be programmed with G58 (see the programming example for G864).

The starting point of the cycle (S) and the end point (E) of the contour described by this cycle must have the same Z-coordinate. The starting point in the X-direction must be at least the minimum amount of 0.1 mm plus two times the programmed stock allowance away and above (internal: below) the end of the contour.

Note
Both the left-hand and the right-hand cutting nose tip are used in machining the contour. The active cutting nose tip is reversed at the boundary between two valleys. Consequently there is a line missing in the graphic simulation which corresponds precisely to a tool width since it is always the course of the left part of the chisel tip which is represented in the graphic simulation..

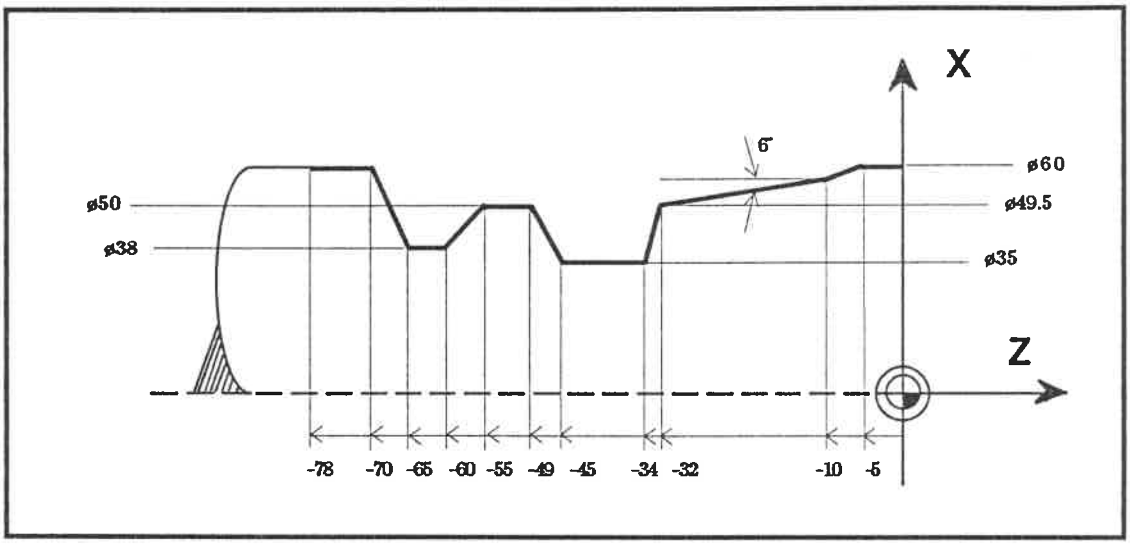
G862

Cycle sequence

First the programmed contour is rough machined, i.e. each valley is recessed in order. The feed rate during this process is **0.75 times the tool width** for each feed movement.
An automatic rest cut indexing with feed movements less than 0.75 times the tool width ensures that each groove base is machined, irrespective of the position of the starting point. Where a number of valleys are involved, an automatic rest cut indexing is performed for each valley.
After each valley is machined, the control system automatically performs a finishing operation along the programmed contour of each valley. Each of these finishing operations is conducted in two steps.
When all the machining is complete the tool is again at the starting point (S).
The contour is machined from the starting point (S) of the cycle to the point of origin (A) of the contour described by the cycle. The sequence of the program blocks of contour description is just the opposite.

G862

Example
The contour (which contains two valleys) is described in subprogram %8661.



Main program %2007
N 1 G96 S100 G95 F0.2 T3 M4
N 2 G0 X63 Z-78
N 3 G862 Z-3
N 4 L8661
N 5 G80
N 6 G0 X100 Z80
N 7 M30

Subprogram %8661
N 1 G42 G0 X60 Z-3
N 2 G1 Z-5
N 3 G1 X? Z-10 B1
N 4 G1 X49.5 Z-32 A-6 B1.5
N 5 G1 X35 Z-34 B1.5
N 6 G1 Z-45 B1.5
N 7 G1 X50 Z-49 B1.5
N 8 G1 Z-55 B1.5
N 9 G1 X38 Z-60 B1.5
N10 G1 Z-65 B1.5
N11 G1 X60 Z-70 B1.5
N12 G1 Z-78
N13 G40 G1 X62
N14 M30

G862

Explanation (main program)

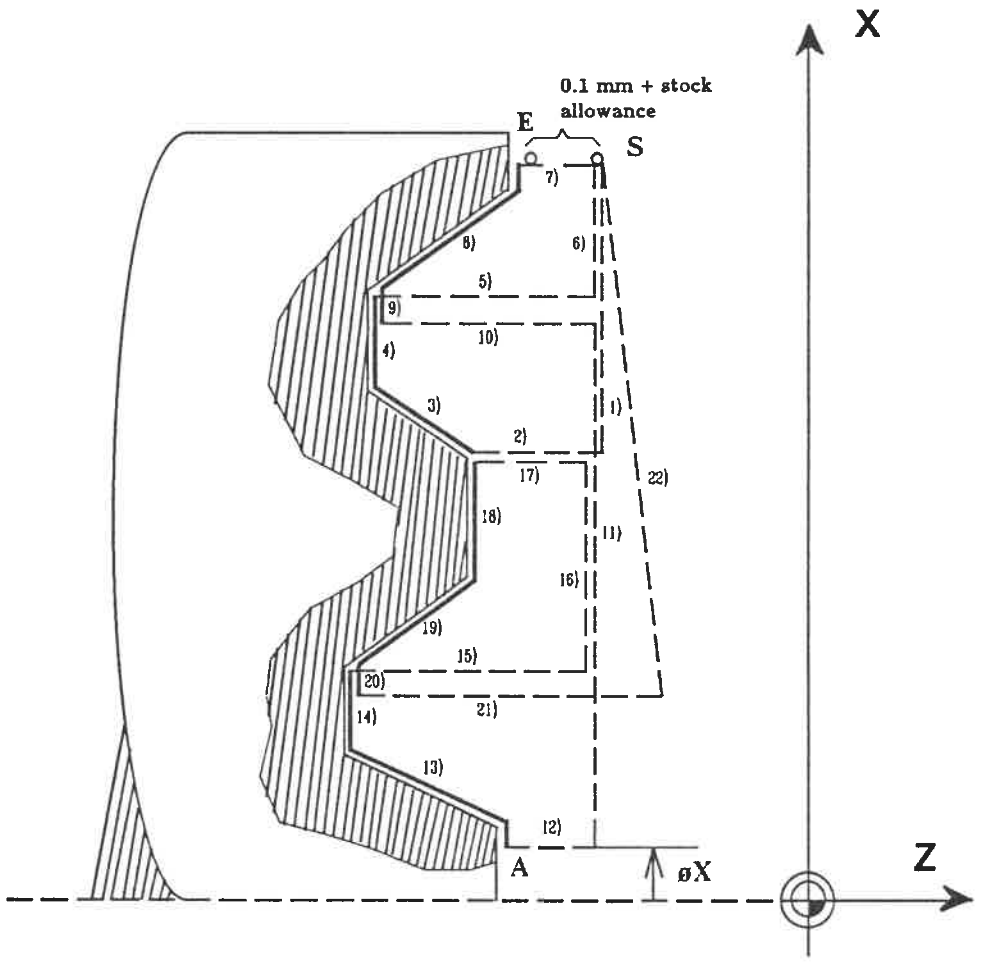
N 1	constant cutting speed; feed in [mm/rev]; tool file T3 turning direction CCW
N 2	rapid traverse to the cycle starting point (S)
N 3	cycle grooving (longitudinal); point of origin (A) of the groove contour in Z-direction
N 4	subprogram activation (contour description of cycle)
N 5	cycle end
N 6	rapid traverse movement
N 7	program end

Explanation (subprogram)

N 1	approach to the point of origin (A) of the contour in rapid traverse; activation of the SRK
N 2	straight line
N 3	straight line with curve (R1)
N 4	straight line with curve (R1.5) at an angle of -6°
N 5	straight line with curve (R1.5)
N 6	straight line with curve (R1.5)
N 7	straight line with curve (R1.5)
N 8	straight line with curve (R1.5)
N 9	straight line with curve (R1.5)
N10	straight line with curve (R1.5)
N11	straight line with curve (R1.5)
N12	straight line to the starting point (S) of the cycle
N13	straight line; switching off the SRK
N14	subprogram end

Groove finishing (traverse)
G863

Cycle groove finishing (traverse), G863
The function G863 can be utilized to finish to final size a keyway which is cut and rough-finished in the front face of the workpiece with cycle grooving G861.



S starting point of the cycle
øX point of origin (A) of the contour described by the cycle in X-direction
E end point of the contour described by the cycle
—— finishing operation
—— positioning movement

Required addresses
After selecting G863, the control system requests the following input:

DIAMETER X:
X-coordinate at which the programmed contour of the groove to be finished begins (A).

G863

Programming
The function G863 is programmed together with the cycle grooving G861. Address X has the same meaning as in G861. All programming regulations listed under G861 also apply, i.e. the starting point of the cycle (S) and the end point (E) of the contour described by this cycle **must have** the same **X-coordinate.**
In **Z-direction** the starting point must be at least the minimum measure of 0.1 mm plus a programmed stock tolerance to the right of the contour end. It is therefore advisable to write the contour in a subprogram that cycle G861 will be the first to access during the course of the program. The contour produced with G58 is recessed and then automatically rough-finished on the basis of a programmed stock allowance parallel to the contour. Afterwards the contour produced with G863 is finished to end size with G863 (G58 A0).

When some contour elements are to be machined using a feed value different from the one determined before the activation of the cycle, then this value can be programmed within the contour description. Programming is done under address F both for G94, that is feed rate in minutes of for G95, that is feed in revolutions.
It is also possible to have chamfers or roundings performed with special feed (address E of the geometry functions G1, G2, G3, G12 and G13). If the contour was written in a subprogram (as recommended) which is used for machining by both G863 and G861, then the changes in feed are not taken into consideration during the machining of the contour with cycle G861.

Cycle sequence
The contour is finished from both sides, one part being machined with the left-hand tool nose and one with the right-hand tool nose.
If the contour consists of a number of valleys, each valley is travelled in order beginning with the valley in the immediate vicinity of the starting point.
At the end the tool is at the starting point (see items 1 to 22 in the diagram on the preceding page).

G863

Example

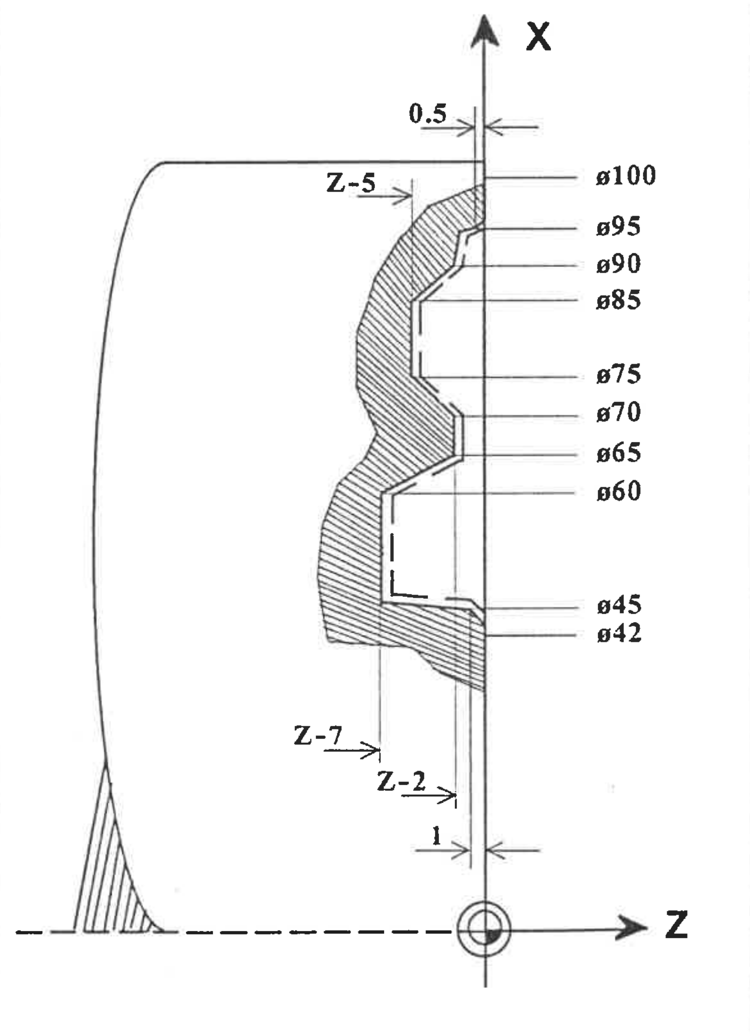
This example refers directly to the programming example for the function G861. The same contour is involved.
As the function G58 A0.3 is programmed before activating the cycle G861, the contour has a stock allowance parallel to the contour, i.e. the contour is recessed and rough-finished. Afterwards the same contour stored in subprogram %8651 is activated a second time as the contour of finishing cycle G863. Since G58 A0 was programmed before the cycle was activated, G863 is used to conduct the finishing to the final size.

Main program %2000

```
N 1  G96  S100  G94  F120  T3  M4
N 2  G0  X100  Z2
N 3  G58  A0.3
N 4  G861  X42
N 5  L8651
N 6  G80
N 7  G14
N 8  T8
N 9  G58  A0
N10  G0  X100  Z2
N11  G863  X42
N12  L8651
N13  G80
N14  G0  X150  Z80
N15  M30
```

Subprogram %8651

```
N 1  G42  G0  X42  Z1
N 2  G1 Z0
N 3  G1 X45  B-1  E10
N 4  G1 X?  Z-7  A5  B1 E10
N 5  G94 G1 X60 B1 E10 F100
N 6  G1 X65  Z-2  B1 E10
N 7  G1 X70  B1 E10
N 8  G1 X75 Z-5 B1 E10
N 9  G94 G1  X85 B1 E10 F100
N10  G1 X90 Z-2  B1 E10
N11  G1 X95 Z?  A95 B1 E10
N12  G1 X? Z0 A177 B-0.5 E10
N13  G1 X100
N14  G40  G1  Z1
N15  M30
```



rough-machining with G861 and G58 A0.3

machining with G863 to final size (G58 A0)

G863

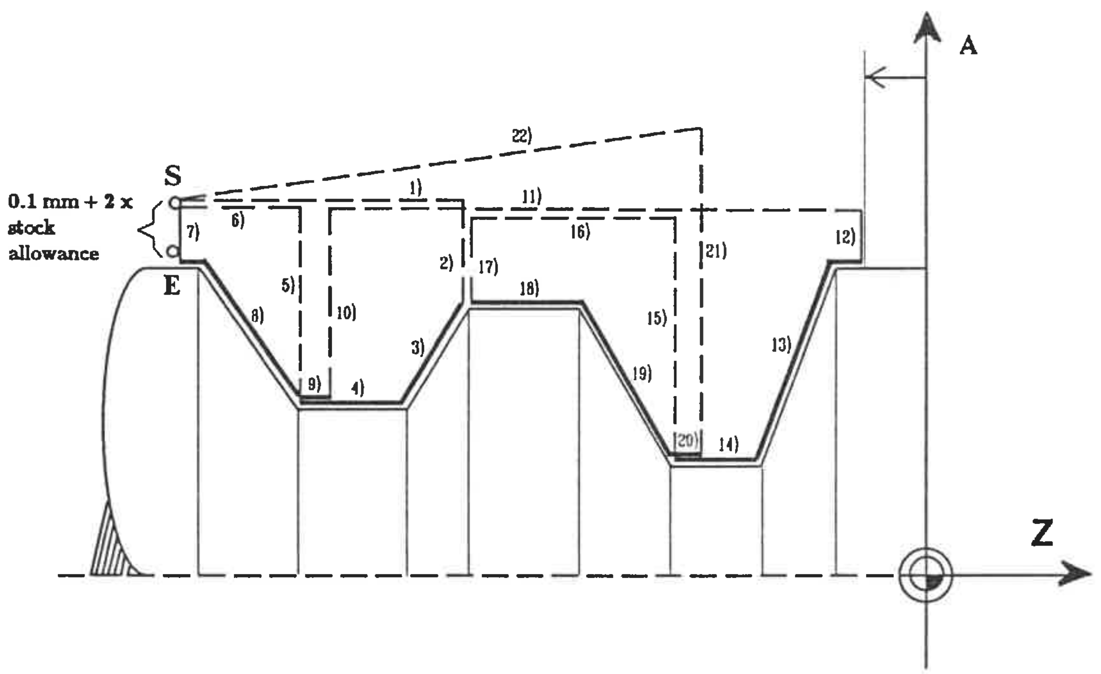
Explanation (main program)

N 1	constant cutting speed; feed in [mm/min]; tool file T3 turning direction CCW
N 2	rapid traverse to cycle G861 starting point (S)
N 3	stock allowance of 0.3 mm parallel to contour
N 4	cycle groove (traverse); point of origin (A) of groove contour in X-direction
N 5	subprogram activation (contour description of cycle; changes in feed are ignored by the control)
N 6	cycle end (G861)
N 7	move to the tool change point
N 8	tool file T8
N 9	selection of the stock allowance
N10	rapid traverse to cycle G863 starting point (S)
N11	cycle groove finishing (traverse); point of origin (A) of the contour to be finished in X-direction
N12	subprogram activation (contour description of cycle; changes in feed are taken into consideration and performed as programmed)
N13	cycle end (G863)
N14	rapid traverse movement
N15	program end

Explanation (subprogram)
(Refer to the key in the programming example for G861)

Groove finishing (longitudinal) G864

Cycle groove finishing (longitudinal), G864
The function G864 can be utilized to finish to final size a keyway cut and rough-finished in the circumference of the workpiece with cycle grooving G862.



S starting point of the cycle
Z point of origin (A) of the contour described by the cycle in Z-direction
E end point of the contour described by the cycle
finishing operation
positioning movement

Required addresses
After selecting G864, the control system requests the following entries:

LENGTH Z:
Z-coordinate which is the start of the programmed contour of the groove to be finished (A).

G864

Programming

The function G864 is programmed together with the cycle grooving G862. Address Z has the same meaning as in G862. All programming regulations listed under G862 also apply, i.e. the starting point of the cycle (S) and the end point (E) of the contour described by this cycle **must have** the same **Z-coordinate.**

In the X-direction the starting point must be at least the minimum measure of 0.1 mm plus **two times** the programmed stock allowance above (internal: below) the contour end.

It is therefore advisable to write the contour in a subprogram that cycle G862 will be the first to access during the course of the program. The contour produced with G58 is recessed and then automatically rough-finished on the basis of a programmed stock allowance parallel to the contour. Afterwards the contour produced with G864 is finished to end size (G58 A0).

When some contour elements are to be machined using a feed value different from the one determined before the activation of the cycle, then this value can be programmed within the contour description. Programming is done under address F both for G94, that is feed rate in minutes of for G95, that is feed in revolutions.

It is also possible to have chamfers or roundings performed with special feed (address E of the geometry functions G1, G2, G3, G12 and G13). If the contour was written in a subprogram (as recommended) which is used for machining by both G864 and G862, then the changes in feed are not taken into consideration during the machining of the contour with cycle G862.

Cycle sequence

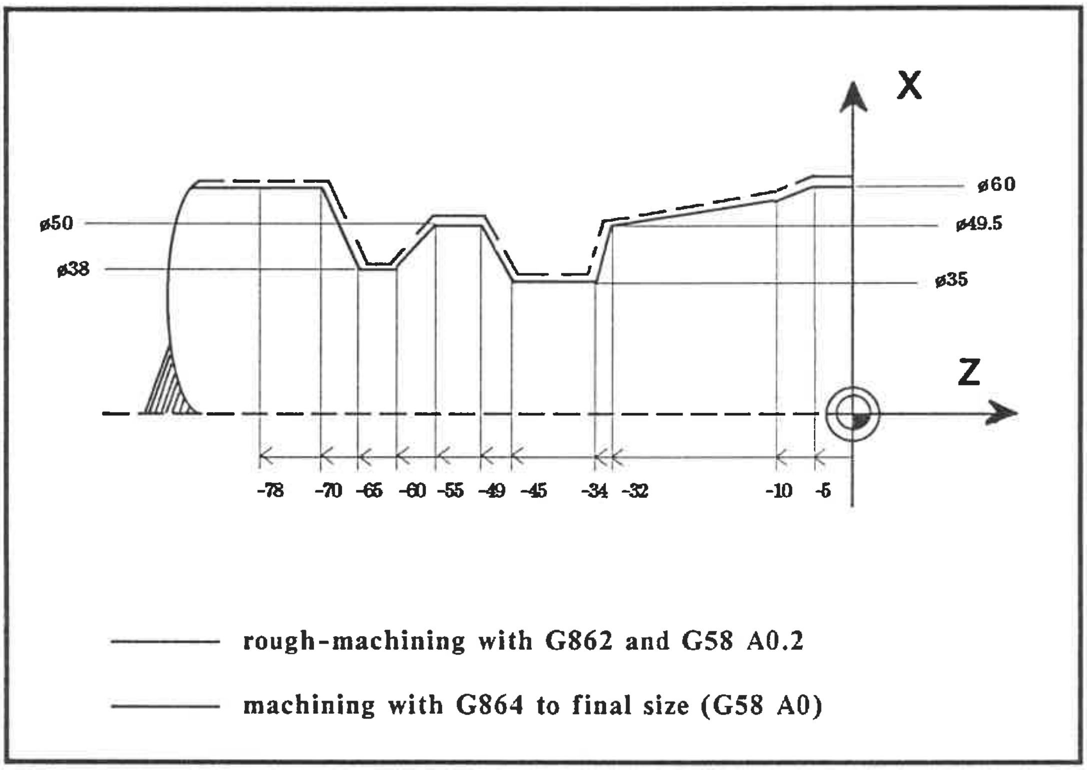
The contour is finished from both sides, one part being machined with the left-hand tool nose and one with the right-hand tool nose.

If the contour consists of a number of valleys, each valley is travelled in order beginning with the valley in the immediate vicinity of the starting point.

At the end the tool is at the starting point (see items 1 to 22 in the diagram on the preceding page).

G864

Example
This example refers directly to the programming example for the function G862. The same contour is involved.
As the function G58 A0.2 is programmed before activating the cycle G862, the contour has a stock allowance parallel to the contour, i.e. the contour is recessed and rough-finished. Afterwards the same contour stored in subprogram %8661 is activated a second time as the contour of finishing cycle G864. Since G58 A0 was programmed before the cycle was activated, G864 is used to finish to final size.



Main program %3000

```
N 1  G96  S100  G95  F0.25  T3  M4
N 2  G0  X63  Z-78
N 3  G58  A0.2
N 4  G862  Z-3
N 5  L8661
N 6  G80
N 7  G14
N 8  T8
N 9  G58  A0
N10  G0  X63  Z-78
N11  G864  Z-3
N12  L8661
N13  G80
N14  G0  X100  Z80
N15  M30
```

G864

Subprogram %8661

```
N 1  G42  G0  X60  Z-3
N 2  G1  Z-5
N 3  G1  X?  Z-10  B1 E0.05
N 4  G1  X49.5  Z-32  A-6  B1.5  E0.05
N 5  G1  X35  Z-34  B1.5  E0.05
N 6  G95 G1  Z-45  B1.5  E0.05  F0.2
N 7  G1   X50  Z-49  B1.5  E0.05
N 8  G1  Z-55  B1.5  E0.05
N 9  G1  X38 Z-60  B1.5  E0.05
N10  G95 G1  Z-65  B1.5  E0.05  F0.2
N11  G1  X60  Z-70  B1.5  E0.05
N12  G1  Z-78
N13  G40  G1  X62
N14  M30
```

Explanation (main program)

N 1	constant cutting speed; feed in [mm/rev.]; tool file T3 turning direction CCW
N 2	rapid traverse to cycle G862 starting point (S)
N 3	stock allowance of 0.2 mm parallel to the contour
N 4	cycle groove (longitudinal); point of origin (A) of the groove contour in Z-direction
N 5	subprogram activation (contour description of cycle; changes in feed are ignored by the control system)
N 6	cycle end (G862)
N 7	move to the tool change point
N 8	tool file T8
N 9	selection of the stock allowance
N10	rapid traverse to cycle G864 starting point (S)
N11	cycle groove finishing (longitudinal); point of origin (A) of the contour to be finished in Z-direction
N12	subprogram activation (contour description of cycle; changes in feed are taken into consideration and performed as programmed)
N13	cycle end (G864)
N14	rapid traverse movement
N15	program end

Explanation (subprogram)
(Refer to the key in the programming example for G862.)

Cycle radius G87

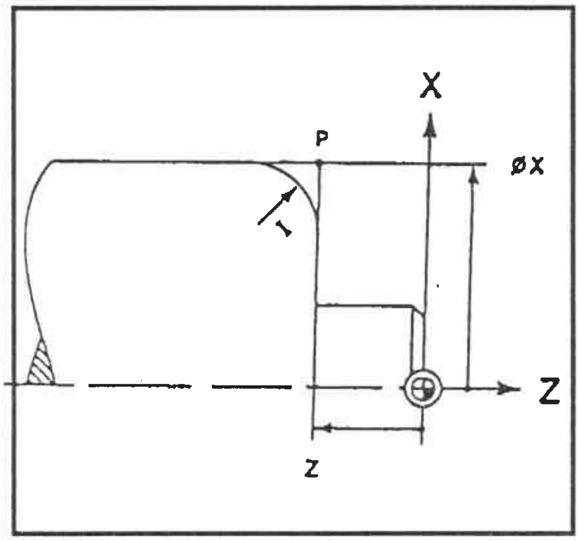
Transition radii, cycle G87
Transition radii at inside and outside corners can be automatically produced using G87. The effect is the same as in programming of G1 B+...
As opposed to G1 B+...,
- the direction of the roundance is obtained from the values of I and K in the tool file,
- separate traversing to the roundance is not possible,
- tool nose radius compensation is active even without being programmed,
- **only** the corners of contour parts parallel to the axis can be rounded.

Precondition:
Tool data X, Z, I and K must be stored in the tool file.
I and K must be entered bearing the correct sign.
The value may not be zero.

Required addresses:
After calling the cycle with G87, the following inputs are requested:

CORNER PT. P OF THE CONTOUR:

DIAMETER	X:
LENGTH	Z:
RADIUS	I:



Programming:
Contour parts parallel to the axis before a "radius" cycle need not be programmed!
After G87, coordinates X and Z of the corner point P must be programmed.
A straight line of length not equal to zero must be programmed before and after G87.
All other programming instructions will produce an error message.
If no tool has been called before G87, the control system computes a roughing tool with I+ and K+.
The control system recognizes the direction of the circular arc from the position of the starting point and the tool dimensions stored under I and K.

G87

The cycle:
The transition radii are machined in one cut.

Note
Large radii can produce excessive cutting depth.

Remedy:
First enter small radius, then program an additional machining step with large radius.

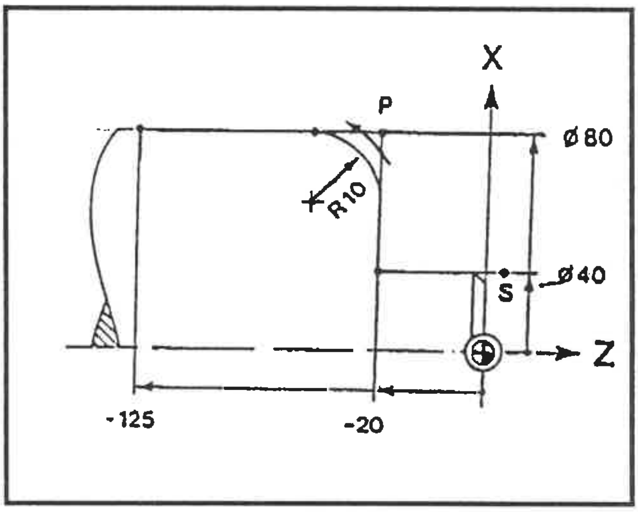
Note

If G81 or G82 stand before G87, then at least one block (i.e. G0 or G1) between them must be programmed with X <u>and</u> Z.

Examples
1. Machining an outside radius including contour parts (parallel to axis) adjoining the radius.

Programming:

...
N 23 G0 X40 Z2
N 24 G1 Z-20
N 25 G87 X80 I10
N 26 G1 Z-125
...



Explanation:

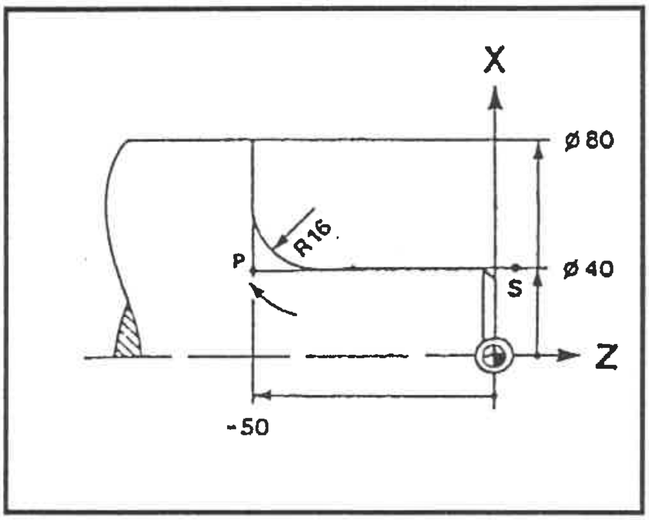
N23/ N24	Traverse to starting point, turn diameter 40.
N25	Cycle radius, the connection from diameter 40 to radius is produced automatically.
N26	Turn diameter 80 mm straight line up to Z-125.

G87

2. Machining an inside radius including the contour parts (parallel to axis) adjoining the radius.

Programming:

...
N 24 G0 X40 Z2
N 25 G87 Z-50 I16
N 26 G1 X80
...



Explanation:

N24 Traverse to starting point.

N25 Cycle radius, The connection from starting point to radius is produced automatically.

N26 Straight line up to diameter 80 mm.

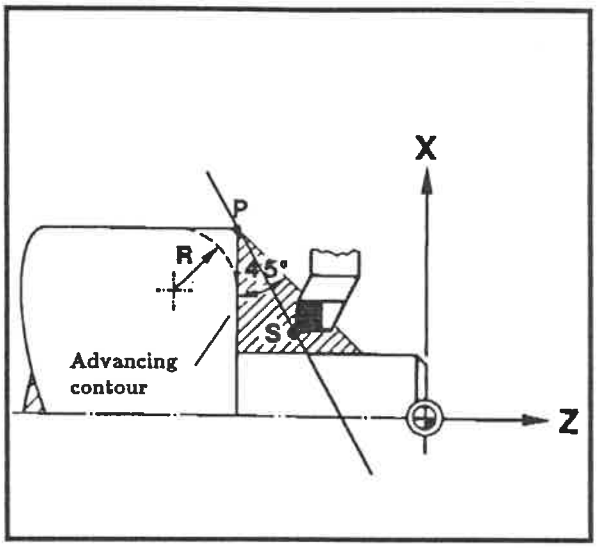
G87

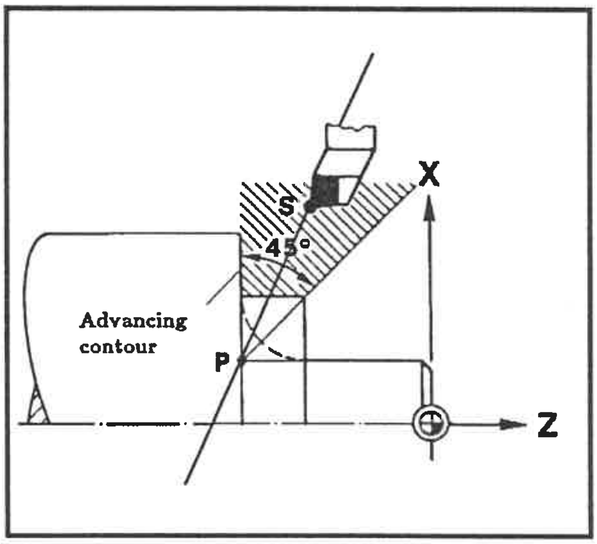
Separate traversing to transition radius
If you wish to machine a transition radius on a right-angled contour separately from the adjoining contours, the tool can be made to traverse to the radius separately.

The direction of the radius must be clearly determined.

The contour to which the control system advances will be the straight line (parallel to axis) which is less than 45° away from the line running from starting point to contour corner.

The starting point must lie within an area 45° from the straight line approached.



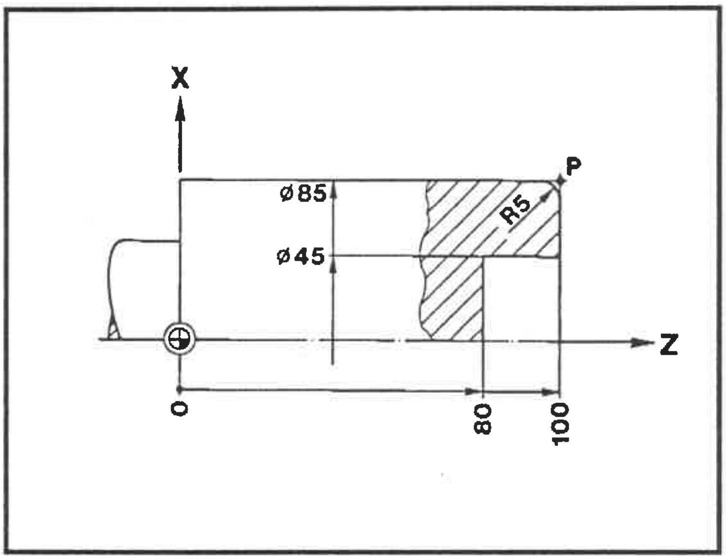


G87

Example
1. Machining an outside radius.
The angle between the adjoining contours is 90°.

Programming:

...
N 24 G1 X87 Z80
N 25 G87 X85 Z100 I5
N 26 G1 X40
N 27 G1 X120
...



Explanation:

N24 Facing, the connection to the radius is produced automatically.

N25 Cycle radius.

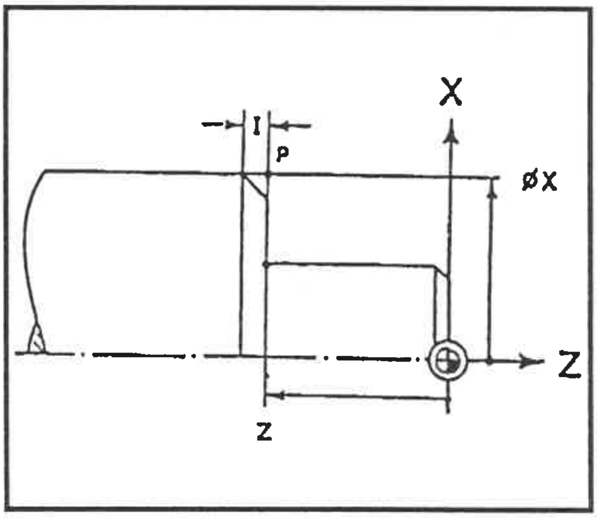
N26 Straight line, destination point required for calculation of the circle.

Cycle chamfer G88

Chamfers, cycle G88
Chamfers between axis-parallel contour parts can be automatically produced using G88. The effect is the same as in programming of G1 B-...
As opposed to G1 B-...,
- the direction of the chamfer is calculated from the values of I and K in the tool file,
- separate traversing to the chamfer is possible,
- tool nose radius compensation is active even without being programmed,
- only the corners of contour parts parallel to the axis can be chamfered.

Precondition:
Tool data X, Z, I and K must be stored in the tool file. I and K must be entered bearing the correct sign. The value may not be zero.

Required addresses:
After calling the cycle with G88, the following inputs are requested:



DIAMETER	X:
LENGTH	Z:
CHAMFER WIDTH	I:

Programming:
Axis-parallel contour parts before a "chamfer" cycle need not be programmed!
Only traverse paths with G0 or G1, of length not equal to zero, may be programmed directly before and after cycle G88!
If no tool has been called before G88, the control system computes a roughing tool with I+ and K+.
The chamfer is always outside.
The direction of the bevel is calculated from the position of the starting point and the tool dimensions stored under I and K.

Note
If G80, G81 or G82 stand before G88, a block (i.e. G0 or G1) with X **and** Z must be programmed between them.

G88

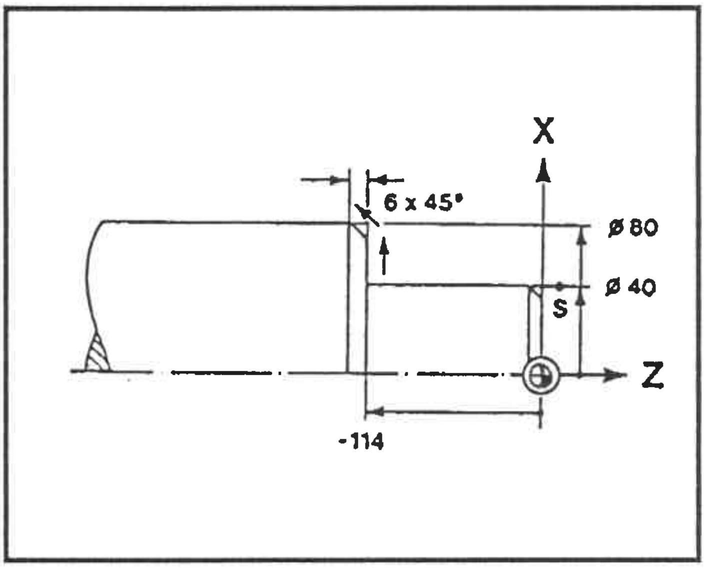
Cycle procedure:
Chamfers are machined in one cut **without dividing the cutting process**. Therefore, a cut with too large depth may be produced when the chamfer is too wide.

Remedy: Enter the small chamfer width first, then program an additional machining step with greater chamfer width.

Example:
Machining a chamfer including the adjoining axis-parallel contour parts.

Programming:

```
N24 G1 X40 Z-114
N25 G88 X80 I6
N26 G1 Z-130
```



Explanation:

N24 Longitudinal turning.

N25 Chamfer cycle, the right-angled connection is produced automatically.

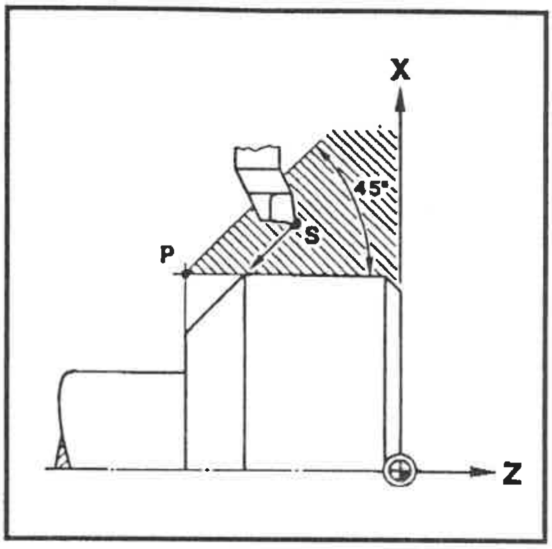
N26 Longitudinal turning.

G88

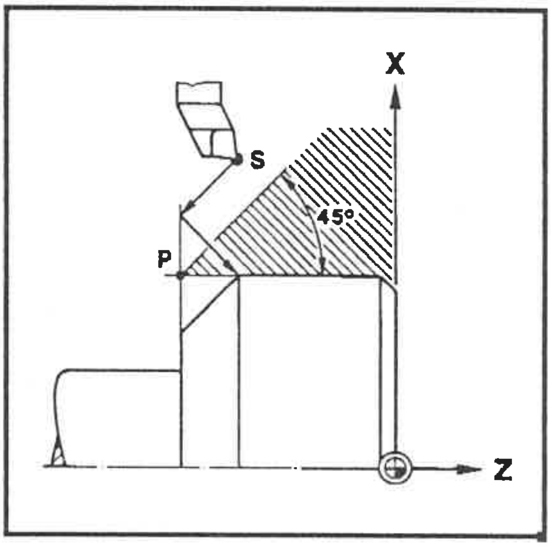
Separate traverse to a chamfer
If you wish to machine a chamfer on a right-angled contour separately from the adjoining contours, the tool may be traversed to the chamfer separately.

The control system calculates start and end point, whereby it selects the point nearest to the starting point to start, and the point farther away to finish.

The starting point S must lie within an angle area 45° from the straight line being approached.



The control system chooses as the approach contour the axis-parallel straight line which is less than 45° from the line running from starting point to contour corner.
If the starting point is selected outside this angle area, the chamfer will be machined in the wrong direction.

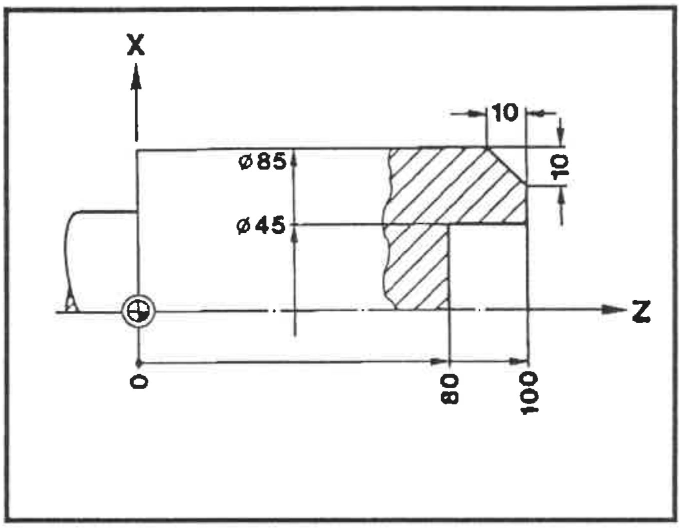


G88

Example
Machining a chamfer with both adjoining contour parts.

Programming:

```
...
N 24 G1  X85  Z40
N 25 G88  X85  Z100  I10
N 26 G1  X40
N 27 G1  Z120
...
```



Explanation:

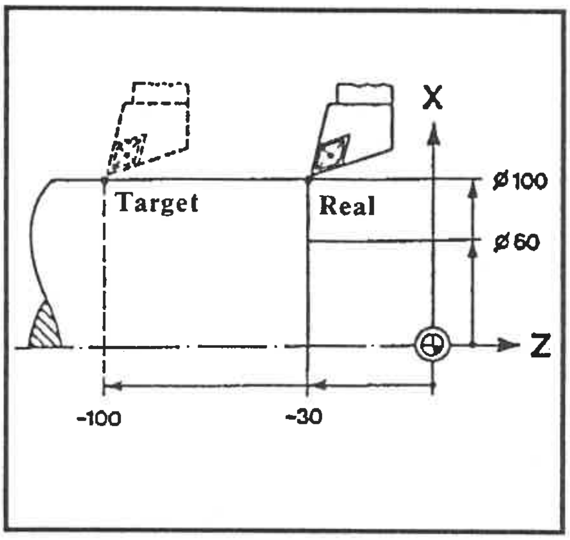
N24 Straight line.

N25 Traverse to chamfer, machine.

N26 Straight line, control system requires destination point in order to calculate the chamfer.

Dimensioning
G90, G91

Absolute dimension, G90
The reference point for all dimensions is the workpiece zero point. Diameter dimensions are programmed directly as diameters.



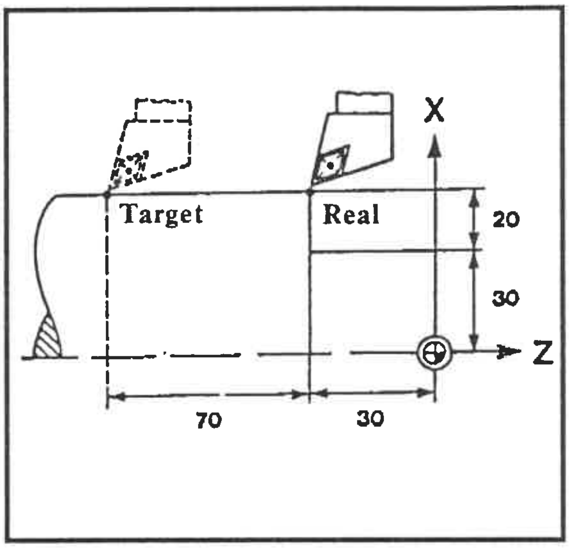
Required addresses
The control system requests no address.

Programming
G90 is active automatically after the control system has been switched on, it need not be programmed.

Information
After processing the function G14 (moving to the tool change point) during the parts routine, a change is automatically made to absolute dimensioning, i.e. the function G90 is active for the remainder of the routine unless G91 is programmed.

G91

Relative (incremental) dimension, G91
All dimensions refer to a **start point** of the **movement** to be performed, i.e. the amount (increment) is indicated by which the slide of the axis involved is to traverse.



Required addresses
The control system requests no address.

Programming
Functions **G90** and **G91** are active when stored and cancel one another.
Movements in direction of the X-axis must be programmed as radius values.

Note
If incremental dimensioning is utilized, all functions activated during the program which produce a coordinate shift are factored in by the control, just as with absolute dimensioning. These include zero point shifts, stock allowances, position corrections and tool-specific corrections.
When processing a program record with an incremental movement during which a coordinate shift is also active, the shift is carried out first and then the programmed incremental path.

Tool dimensions
G92

Changing the tool dimensions in the NC-program, G92
By programming G92, the tool type, tool color code for graphics, presetter dimensions, cutting edge dimensions and cutting edge angles A and B can be determined and changed in the NC program. When executing the block with G92, the values programmed for W, Q, X, Z, I, K, A, B, C, E and L are active. It depends on the position of the parameter N2 whether these values are accepted for the tool file. If the tool file is opened, then the programmed dimensions are written in the file and take effect. With the tool file being closed the data take effect, but they are not transferred to the tool file.
If any of the addressses requested for G92 is confirmed, the value from the tool file which is allocated to the relevant tool is accepted.

Required addresses:
After selecting G92, the control system requests the following inputs:

TOOL NUMBER:
Two-digit input if parameter N1600 = 0 or six-digit input of a tool identification number if N1600 = 1

TOOL TYPE W:
Definition of the tool type.
Note: In the parameter sets N1001 to N1064 the tool type is listed under the address WT although these addresses are identical.

COLOR CODE Q:
Color code which is used to represent the tool in the utilization graphic.
Note: In the parameter sets N1001 to N1064 the color code is listed under the address FC although these addresses are identical.

PRESETTER DIMENSION X:
Calculation value (from tool reference point to tool tip)

G92

PRESETTER DIMENSION Z:
Calculation value (from tool reference point to tool tip)

TOOL TIP I:
Distance from nominal tool tip to cutting edge center in X-direction.

TOOL TIP K:
Distance from nominal tool tip to cutting edge center in Z-direction.

ANGLE A:
Cutting edge angle A (varying meaning according to tool type)

ANGLE B:
Cutting edge angle B (varying meaning according to tool type)

RAKE ANGLE; TEETH C:
Number of teeth of the milling tool. It is only necessary to program this address if auxiliary drive 1 is fed with the function G193 (mm/tooth). The definition of a rake angle is of no consequence for this type of control.

DRILL DIAMETER E:
Measurement of diameter of milling tool or drill
Attention: The cutter or drill diameter is listed under address D in the parameter records N1001 to N1064. Nevertheless the same address is involved.

DRILL LENGTH L:
Length of the milling tool or drill.

Note
When executing G92, the D-tool offset corresponding to the tool number is automatically active. If another D-tool offset is desired, it can be called under the address D in the block with G92.
If D0 is programmed under that address, there is no active D-tool offset.

Other ways of input
Tool data can also be programmed manually in the parameters, or the parameter data may be entered by means of a punched tape or a DATAPILOT system.

Main spindle
G94, G95

Feed
Feed is programmed under address F.
There are two ways to program feed:
in **mm/min** and in **mm/rev.**.

1. Feed F in mm/min, G94
G94 causes the value programmed under F to be executed as feed in mm/min.

2. Feed in mm/rev., G95
G95 causes the value programmed under F to be executed as feed in mm/rev. G95 takes effect after switching on and need not be programmed.

Feed override
The programmed feed can be altered in percentages using the handwheel in AUTOMATIC or SINGLE BLOCK operating modes.
The override of the feed value may range from **-100%** to **+50%** relative to the current programmed feed value (100%), i.e. feed values from 0 to 150% can be set.

(See section 5.)

G96, G97

Spindle speed / cutting speed
Spindle speed or cutting speed is programmed under address S. When multistage machines are used, the desired gear stage must always be programmed first.
The gear stages are entered using M functions (see section 3.6).

1. Cutting speed S in m/min, G96 (V constant)
G96 causes the value programmed under S to be taken as speed in m/min.
The spindle speed depends on the X-position of the tool tip whose slide last called up G96 so that the cutting speed remains constant.
This is possible only within the speed range of the selected gear stage.
G96 and G97 cancel each other. Both functions are self-maintaning.
When switching over from G96 to G97, a new S-value must be programmed as the old value will otherwise be accepted with false designation.

2. Speed S in rev./min, G97
After programming G97, the value under address S is taken as speed/min.

Note
Speed may be limited by the chuck, workpiece or similar influences. In order to prevent excessive speed due to reduced diameters, the maximum speed should be entered under parameter N501 (see section 6) or it should be programmed using G26.

G96, G97

Example:

N10 G97 S500	Speed 500 rev/min from now onward.
...	
...	
N20 G96 S180	Cutting speed 180 m/min from now onward.

Gear stages in one-stage machining (no gear)

No gear shift commands allowed.

Gear stages in multiple stage machining

	Command	
Stage 1	M41	low speed high torque
to		
Stage 4	M44	high speed low torque

Speed override
The programmed S value can be altered in percentages in operating modes **AUTOMATIC** and **SINGLE BLOCK.**
Speed override is possible ranging from -50% to +50%, whereby the currently programmed speed is taken as 100%
(see section 5)

Speed override is not possible for the speed of the driven tool.

Milling with the C-axis
G100 - G103, G110 - G113, G152

General

The **ELTROPILOT L** control is designed for C-axis machining to permit milling motions on the frontface or circumference of a workpiece with the aid of a driven tool.

The C-axis machining describes motions one component of which represents a linear movement (X-direction during frontface machining or Z-direction during circumferential machining), the other component of which represents a circular movement around the axis of rotation. The C-axis is referred to the main spindle, and the drive of this C-axis is swivelled in (out) with the aid of function M14 (M15).

Important

During the processing of all contour descriptions the control moves the cutter middle point on the programmed contour. If keyways the width of the cutter diameter are to be produced, the programmed contour represents the middle line of the keyway. This case is unproblematic.

If concave or convex areas are to be milled out of the frontface or circumference, however, the programmer must keep in mind that the cutter middle point moves on the programmed contour but the cutting surface of the cutter is offset by the amount of its radius. The programer either calculates his contour description accordingly and has this calculation performed by the milling cutter radius compensation (see the functions G40 to G42, "Cutter radius compensation").

All programmed paths in conjunction with the C-axis have to be in **absolute** dimensioning (G90). Simplified geometry calculation (VGP) is not permitted in program parts with C-axis machining.

Milling feed

There are two possibilities for programmation of the feed of the milling tool: **mm/min** (independent by function G94) or **mm/tooth** (dependent from drive by function G193).

The feed for milling machining in connection with the c-axis refers all the time to the path curve of the milling center point (center point path). Independent thereby if the FRK is activated or not.

G100 - 103, G110 - G113, G152

Real value display by milling radius compensation
In the position display (real value display) the actual position of the milling center point (equidistant) is displayed, even with switched on milling radius compensation; this means these path positions are shown, to which the programmed feed refers.

Tool calculation with c-axis machining

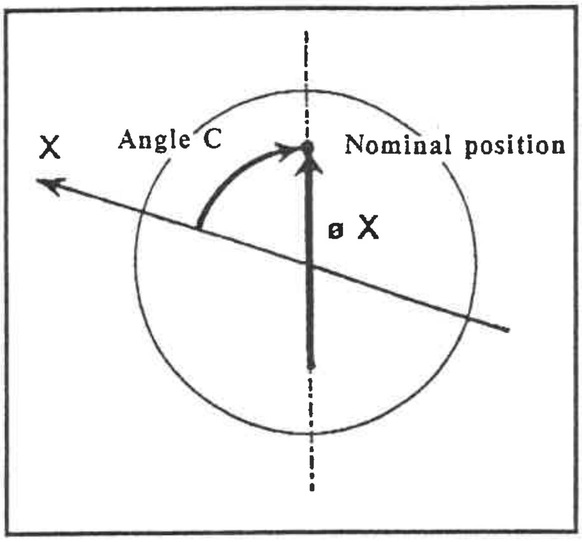
Front face machining:	Axial clamped tools are to be selected and calculated by the tool type WT24. The calculation value in the tool memory thereby is to be **X = 0.000.**
Circumference machining:	Radial clamped tools are to be selected and calculated by WT13.

Selection / Deselection of the milling radius compensation
The milling radius compensation may be selected resp. deselected only in this plane (front face or circumference) in which even the milling diameter is to be compensated; **under no circumstances** is the FRK to be selected when the milling cutter is being fed vertically into the particular machining level. There is no allowance to change the machining plane while the FRK is active (for detailed description of the FRK see G40 - G42).

Correction of the milling tool
The tool wear correction of the milling tool is to be executed exclusively under the address DD (therefor see also parameter N1101 - N1180.

Frontface machining G100

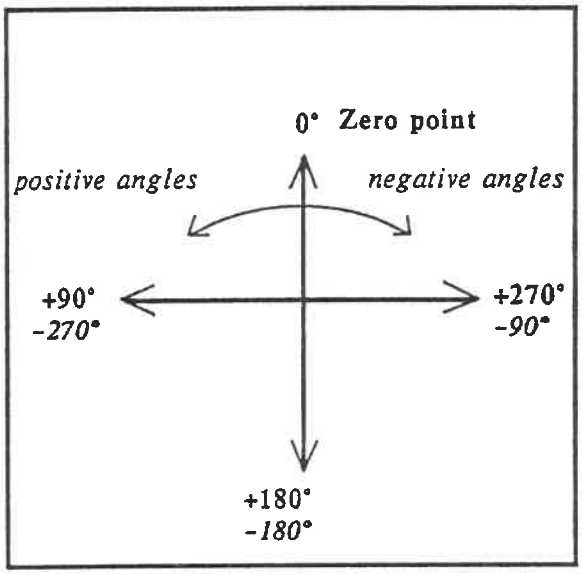
MILLING FRONTFACE rapid traverse, G100
If the tool tip of the driven tool is to be positioned in rapid traverse movement to a defined position on the frontface of a workpiece, then this positioning movement is to be programmed with G100.



Required addresses
After the selection of path condition G100, the control system requests the following inputs:

DIAMETER X:
Nominal position in X direction on the frontface, in diameter dimension.

ANGULAR VALUE C:
The angle dimension of the nominal position is entered under this address in degrees. Angular values ranging from -9999.999° to +9999.999° can be entered here.
The angles to be programmed on the frontface of the workpiece are represented in the drawing on the right. However, the actual direction of rotation of the spindle is in the opposite direction.



Example
N 10 G100 X30 C70

Treating this block, the slide moves to the programmed position in X-direction and at the same time the C-axis is positioned at the programmed angular value.

Note
1) Traverse movements via G100 usually do not follow a straight line since C-axis and X-axis independently move at the highest possible speed. It is, therefore, not possible to mill a keyway using G101 and then return into the keyway on the same path using G100.

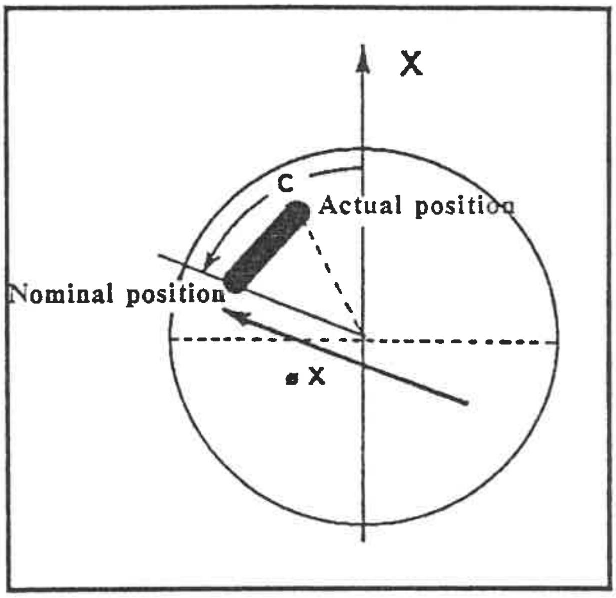
2) The speed values for rapid traverse movements are stored in the parameter memory and can be changed in the PARAMETER operating mode.
In the operating modes AUTOMATIC, SINGLE BLOCK and MANUAL MODE, the values for rapid traverse movements can be changed using the handwheel.

3) The feedrate override is limited to a maximum of 100%, even if a value exceeding 100% is displayed.

G101

FRONTFACE MILLING linear, G101
If a linear milling movement with given feed speed is to be executed on the frontface of the workpiece, using the driven tools, then this machining process is to be programmed with the function G101.

Required addresses
After selection of the path condition G101, the control system requires the following inputs:



DIAMETER X:
Nominal position in X direction on the frontface in diameter dimension.

ANGULAR VALUE C:
Under this address the angular value of the nominal position is entered in degrees.
The X-axis corresponds to the 0° mark.
Positive angles count in counter-clockwise direction, negative angles count in clockwise direction (see figure under G100).

Note
Please note that the tool must not be traversed further than the center of turning. Movements surpassing the center of turning which extend over the coordinate middle point (centre of rotation) in the X-direction must be stopped at the center and continued in opposite direction after the working spindle was turned around by 180°.

For example:

```
...
N...  G100  X100  C0
N...  G101  X0  C0
N...  G101  X0  C180
N...  G101  X100  C180
...
```

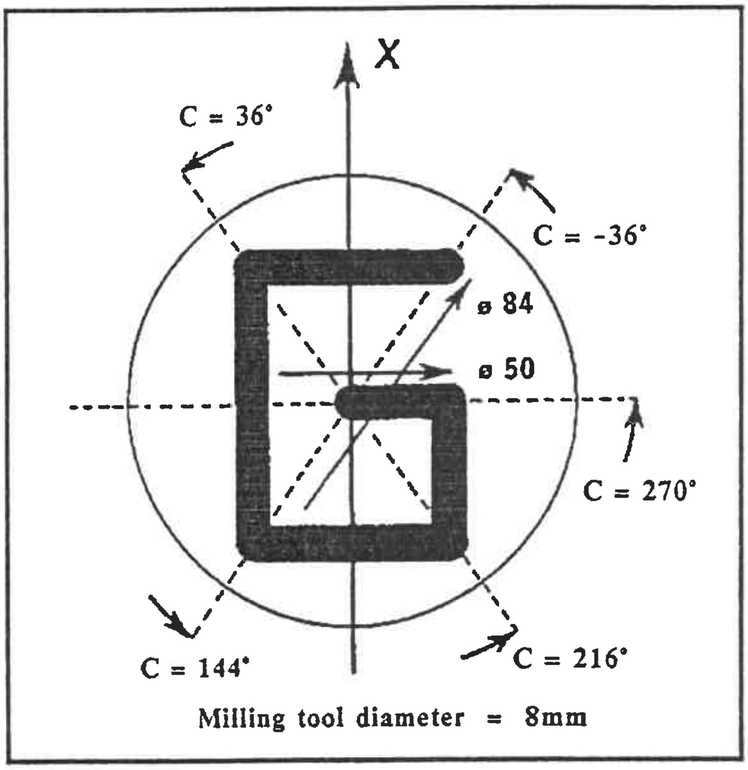
The depth of the produced keyway depends upon how far the workpiece was approached by the milling tool.
The breadth of the keyway to be milled depends on the geometry of the tool.

G101

Example
Milling of a keyway of 2 mm depth on the frontface of a workpiece.

Programming

```
...
N10 M14
N11 G100 X84 C-36
N12 G94 F30 G1 Z-2
N13 G101 X84 C36
N14 G101 X84 C144
N15 G101 X84 C216
N16 G101 X50 C270
N17 G101 X0 C270
N18 G0 Z30
...
```



Explanation

...

N10	Coupling in the C axis
N11	Positioning the milling tool in front of the piece to position diameter X = 84 mm and angle C = -36°
N12	Feedrate 30 mm/min; approach of milling tool 2mm into the piece
N13	Linear milling movement to position diameter X = 84 and angle C = 36°
N14	Linear milling movement to position diameter X = 84 and angle C = 144°
N15	Linear milling movement to position diameter X = 84 and angle C = 216°
N16	Linear milling movement to position diameter X = 50 and angle C = 270°
N17	Linear milling movement to position diameter X = 0 and angle C = 270° is maintained
N18	Moving out of the keyway to 30 mm in front of the piece

...

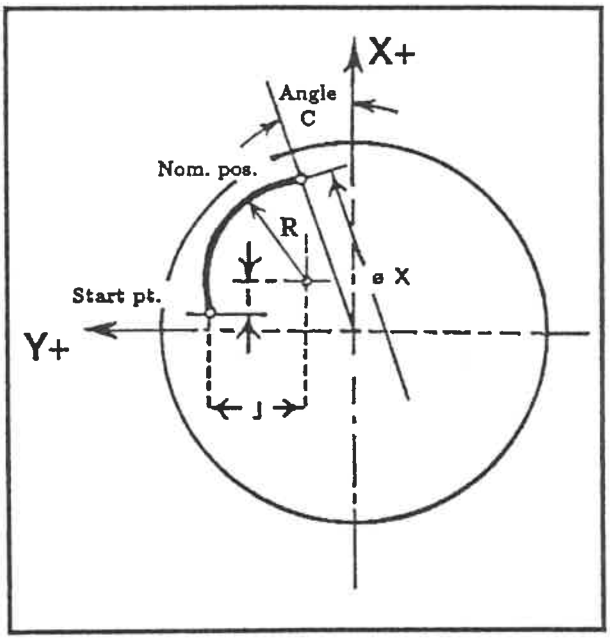
G102

FRONTFACE MILLING of a circular arc in clockwise direction (CW), G102

If, by means of the driven tools, a circular milling movement is to be performed on the front face of the workpiece, with given feed speed and in clockwise direction, then this machining process must be programmed using function G102.

Required addresses

After selection of path condition G102, the control system requires the following inputs:



DIAMETER X:

Nominal position in X direction on the front face, in diameter dimension.

ANGULAR VALUE C:

Under this addres, the angular value of the nominal position is indicated in degrees. The X axis corresponds to the 0° mark. Positive angles count in counter-clockwise direction, negative angles count in clockwise direction (see figure for G100)

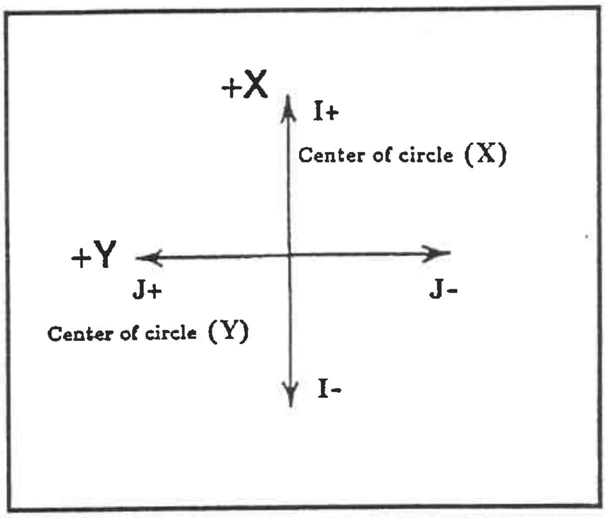
RADIUS R:

The radius of the circular arc to be milled is entered under this address. It is only permissible to use the radius entry if the circular arc to be milled has a traversing angle not equal to zero but smaller than/equal to 180° (0° < R ≤ 180°).
This form of entry is not permitted where larger traversing angles are involved. Furthermore, programming of complete circles is only permitted with the circle middle-point entries I and J. If the circle middle-point entries I and J are supposed to be or have to be used, the address R has to be confirmed; the control then offers the address parameters I and J. If a radius is entered, addresses I and J do not apply.

G102

CENTER OF CIRCLE (X) I:
Here the distance in X direction between the start point and the center of the circle is indicated. When viewing from the start point S parallel to the X-axis in direction of the center of the circle, then it applies:

In direction of the X-axis I+
In opposite direction of X-axis I-



CENTER OF CIRCLE (Y) J:
Here the distance between starting point S and the center of the circle in Y direction is indicated (see figure to the right). When viewing from the starting point S parallel to the Y-axis in the direction of the center of the circle, then it applies:

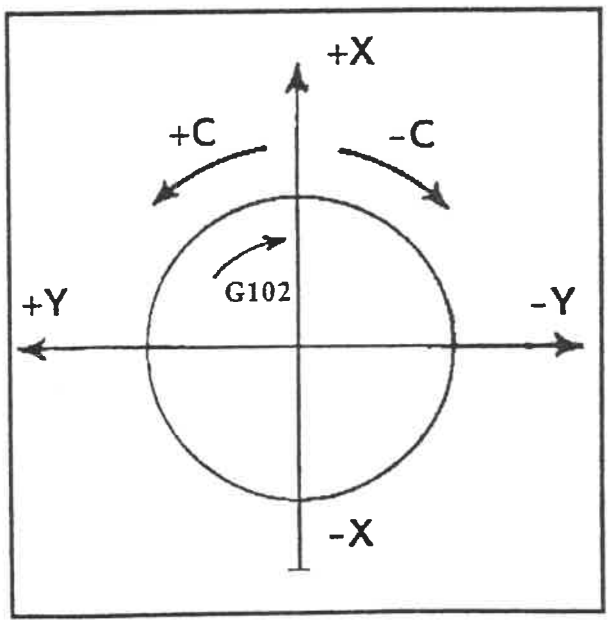
In direction of the Y-axis J+
In opposite direction of Y-axis J-

Programming complete circles
A distinction must be drawing between two possibilities when programing complete circles on the frontface of the workpiece:

1) The distance of the middle point of the circle from the axis of rotation is **greater** than the radius of the circle. Starting and ending angle have to be **identical** in this case.

2) The distance of the middle point of the circle from the axis of rotation is **smaller** than the radius of the circle. In this case the difference between starting and ending angle must be 360°.

Note
The tool must not be moved over the center of turning. Movements surpassing the center of turning in X-direction must therefore be stopped and continued in opposite direction after the working spindle has been turned around by 180°.

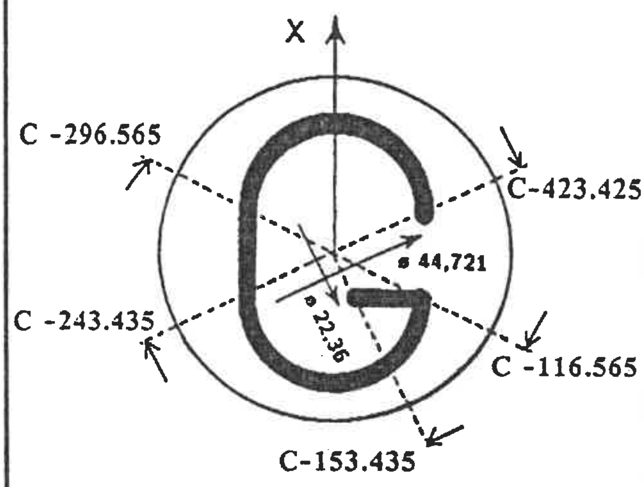


G102

Example
Milling of a keyway of 2 mm depth on the frontface of the workpiece.

Programming

```
...
N10  M14
N11  G100  X22.36  C-153.435
N12  G94  G1  F30  Z-2
N13  G101  X44.721  C-116.565
N14  G102  X44.721  I0  J20  C-243.435
N15  G101  X44.721  C-296.565
N16  G102  X44.721  I0  J-20  C-423.425
N17  G0  Z20
...
```



Milling tool diameter = 4mm

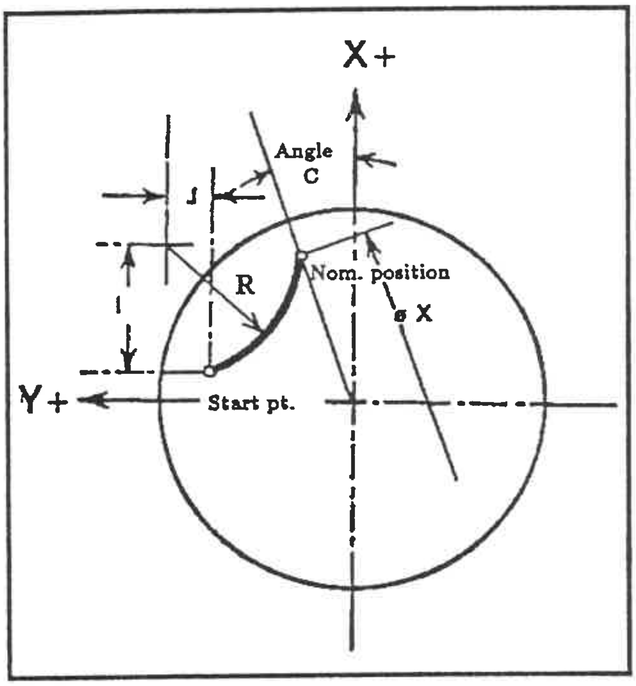
Explanation

N10	Swivelling-in of the C-axis 1 and reference run
N11	In rapid traverse to the frontface of the piece, diameter X = 22.36 and angle C = -153.435
N12	Feedrate 30 mm/min, approach of milling tool by 2mm into the piece
N13	Linear movement to angle -116.565° and diameter 44.721
N14	Circular movement (semi-circle) in clockwise direction to diameter 44.721 and angle -243.435°; distance starting point to center point J20 I0.
N15	Linear movement to angle -296.565° and diameter 44.721
N16	Circular movement (semi-circle) in clockwise direction to diameter 44.721 and angle -423.425°; distance starting point to center point J-20 I0
N17	Moving out of the keyway in rapid traverse to 20 mm in front of the piece

G103

FRONTFACE MILLING of a circular arc in counter-clockwise direction (CCW), G103
If, by means of the driven tools, a circular movement with given feed speed is to be performed on the frontface of a piece in counter-clockwise direction, then this machining process can be programmed with G103.

Required addresses
After selection of path condition G103, the control system requires the following inputs:



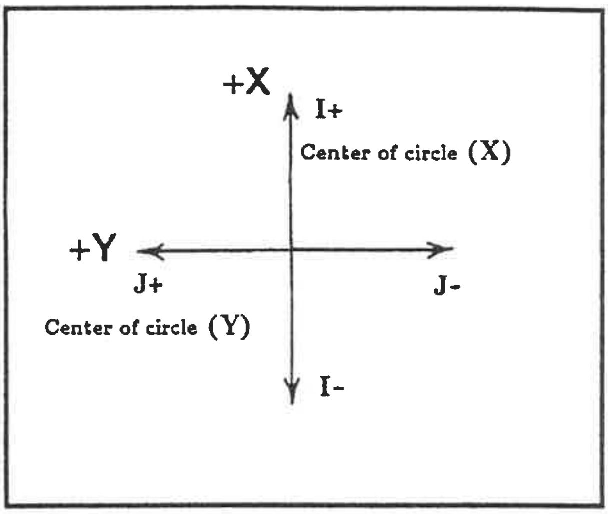
DIAMETER X:
Nominal position in X direction on the frontface, in diameter dimension.

ANGULAR VALUE C:
The angular value, in degrees, of the nominal position is entered under this address.
The X-axis corresponds to the 0° mark. Positive angles count in counter-clockwise direction, negative angles count in clockwise direction (see figure for G100).

RADIUS R:
The radius of the circle to be milled is entered under this address. The radius entry is only to be used if the circle to be milled represents a traversing angle not equal to zero but smaller than/equal to 180°
($0° < R \leq 180°$).
Where larger traversing angles are involved, this form of entry is not permitted. In addition, complete circles are only to be programmed with the circle middle-point entries I and J. If the circle middle-point entries I and J are to be or have to be used, the address R must be confirmed; the control will then offer the address parameters I and J. If a radius is entered, addresses I and J are not applicable.

G103

CENTER OF CIRCLE (X) I:
Here the distance between start point S and center of the circle in X-direction, is indicated. When viewing from starting point S parallel to the X-axis in direction of the centerof the circle, it applies:



In direction of the X-axis	I+
In opposite direction of X-axis	I-

CENTER OF CIRCLE (Y) J:
Here the distance between the start point S and the center of the circle, in C direction, is indicated (see figure to the right). When viewing from starting point S parallel to the Y-axis in direction of the center of the circle, then it applies:

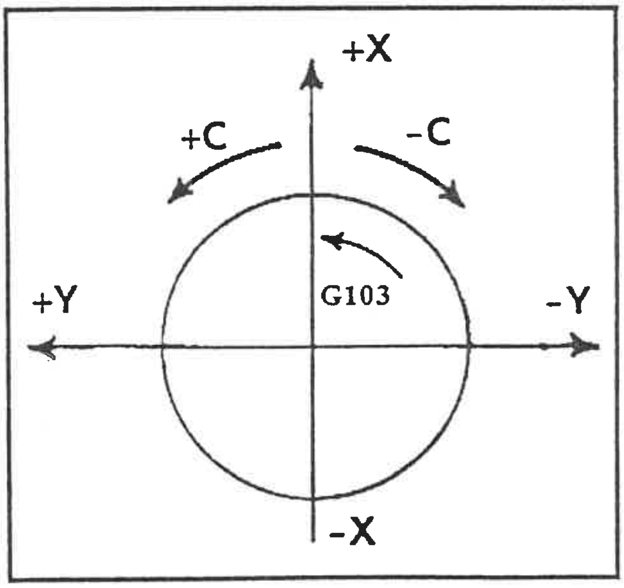
In direction of the Y-axis	J+
In opposite direction of Y-axis	J-

Programming complete circles
A distinction must be drawing between two possibilities when programming complete circles on the frontface of the workpiece:

1) The distance of the middle point of the circle from the axis of rotation is greater than the radius of the circle. Starting and ending angle have to be **identical** in this case.

2) The distance of the middle point of the circle from the axis of rotation is smaller than the radius of the circle. In this case the difference between starting and ending angle must be 360°.

Note
The tool must not be moved over the center of turning. Movements surpassing the center of turning in X-direction must therefore be stopped and continued in opposite direction after the working spindle has been turned a round by 180°.

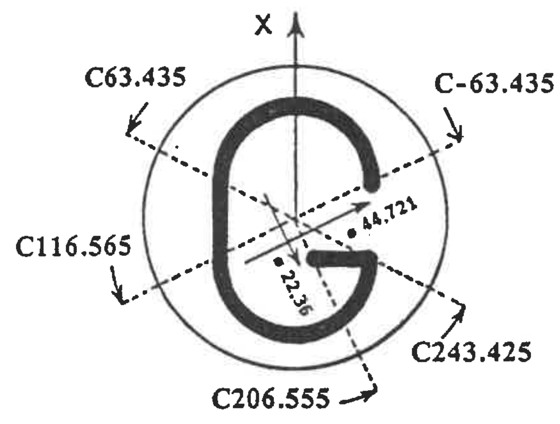


G103

Example
Milling of the same workpiece as in the previous example, but the circular movements are performed by programming function G103.

Programming

```
...
N10 M14
N11 G100 X44.721 C-63.435
N12 G94 G1 F30 Z-2
N13 G103 X44.721 I0 J20 C63.435
N14 G101 X44.721 C116.565
N15 G103 X44.721 I0 J-20 C243.425
N16 G101 X22.36 C206.555
N17 G0 Z20
...
```



Milling tool diameter = 4mm

Explanation

N10 Swivelling-in the C-axis 1 and reference run

N11 Moving in rapid traverse to the frontface of the piece, diameter X = 44.721 and angle -63.435°

N12 Feedrate 30 mm/min, approach of milling tool by 2mm into the piece

N13 Circular movement (semi-circle) in counter-clockwise direction to diameter 44.721 and angle 63.435°; distance start point - center J20 I0.

N14 Linear movement to angle 116.565° and diameter 44.721

N15 Circular movement (semi-circle) in counter-clockwise direction to diameter 44.721 and angle 243.425°; distance start point - center J-20. I0

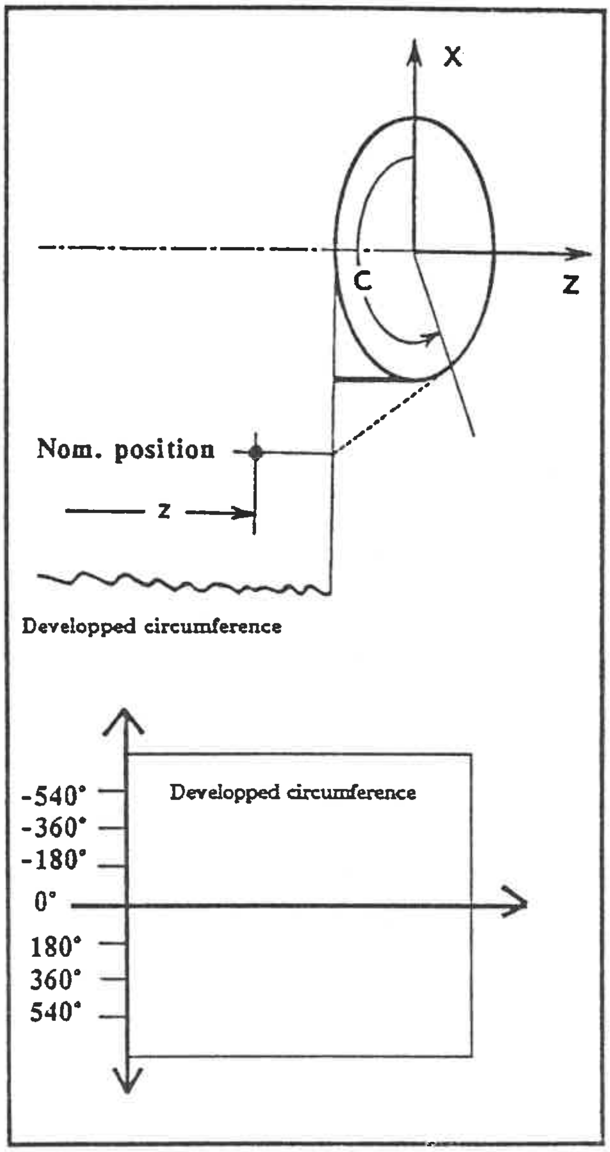
N16 Linear movement to angle 206.555° and diameter 22.36

N17 Moving out of the keyway in rapid traverse to 20mm in front of the piece

Machining on circumference G110

MILLING ON CIRCUMFERENCE in rapid traverse, G110
If the tip of the driven tool is to be positioned in rapid traverse at a nominal position on the circumference of a piece to be machined, then this positioning movement is to be programmed using G110.

Required addresses
After selection of path condition G110 the control system requires the following inputs:



LENGTH Z:
Nominal position in Z direction on the circumference of the piece.

ANGULAR VALUE C:
Under this address is indicated the nominal position as angle of rotation, in degrees, of the working spindle, referred to the zero point of the C-axis.
Angular values ranging from -9999.999° to +9999.999° may be programmed here.

Example
N 10 G110 Z-30 C70

Having treated this block, the slide moves in longitudinal direction to the programmed position and at the same time the C-axis is positioned to the programmed angular value.

Note
1) Traverse movements via G110 usually do not follow a straight line since C-axis and X-axis independently move at the highest possible speed. It is, therefore, not possible to mill a keyway using G111 and then return into the keyway on the same path using G110.

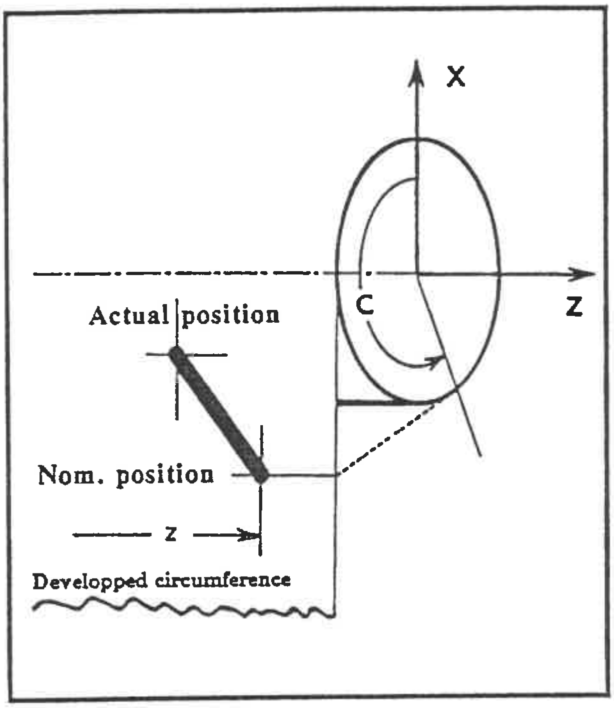
2) The rapid traverse speed values are stored in the parameter memory and may be changed in the PARAMETER operating mode. In the operating modes AUTOMATIC, SINGLE BLOCK and MANUAL MODE the rapid traverse speed value may be changed using the handwheel.

3) The feed override is limited to a maximum of 100% even if a value above 100% is displayed.

G111

MILLING ON CIRCUMFERENCE linear, G111
If, by means of the driven tools, a linear milling movement is to be performed on the circumference of a piece, with given feed speed, then this machining process is to be programmed using function G111.

Required addresses
After selection of path condition G111 the control system requires the following inputs:



LENGTH Z:
Nominal position in Z-direction on the circumference of the piece.

ANGULAR VALUE C:
Under this address the nominal position is indicated as angle of rotation, in degrees, of the working spindle, referred to the zero point of the C-axis. Angular values ranging from -9999.999° to +9999.999° may be programmed here.

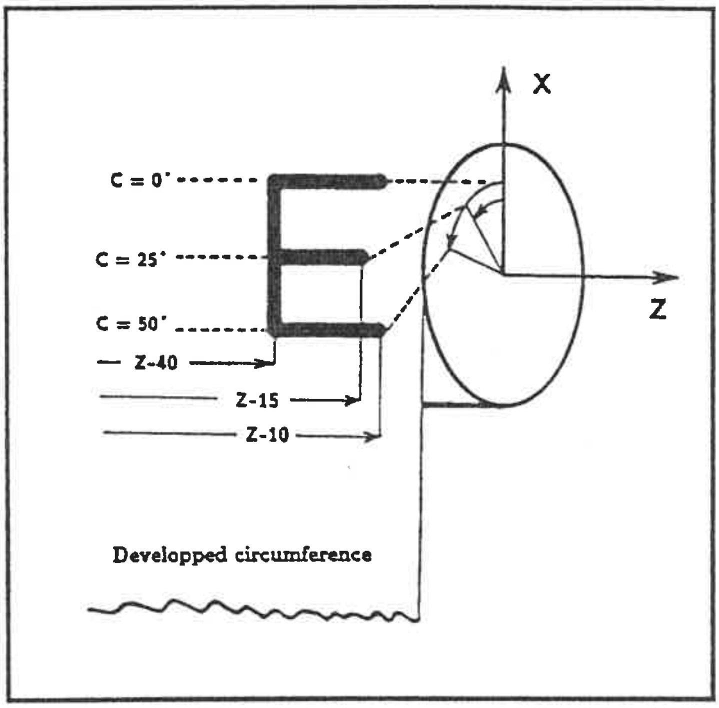
Note
The depth of the produced keyway depends on how deep the milling tool approaches the piece in X-direction.
The width of the keyway to be milled depends on the geometry of the tool.

G111

Example
Milling of a contour on the circumference
of a workpiece (diameter of the piece = 80mm)

Programming

...
N 6 M14
N 7 G110 Z-10 C0
N 8 G94 F30 G1 X76
N 9 G111 Z-40 C0
N10 G111 Z-40 C50
N11 G111 Z-10 C50
N12 G0 X84
N13 G100 Z-40 C25
N14 G1 X76
N15 G111 Z-15 C25
N16 G0 X100
...



Explanation

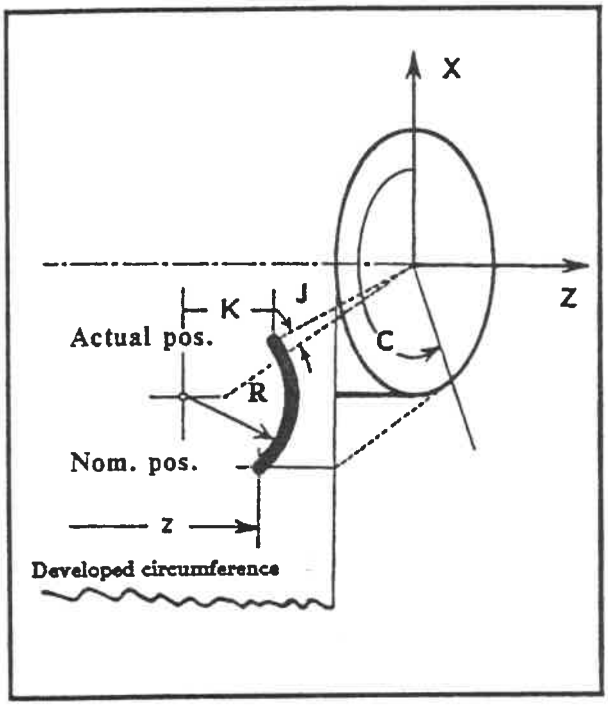
...

N 6	Swivelling-in of C-axis 1 and reference run
N 7	Move in rapid traverse above the circumference of the piece, length Z = -10 and angle C = 0°
N 8	Feedrate 30 mm/min, approach of milling tool 2mm into the piece (diameter X = 76)
N 9	Linear movement to Z-40 and angle of rotation C = 0°
N10	Linear movement to angle C = 50° and length Z-40
N11	Linear movement to length Z-10
N12	Moving out of keyway in rapid traverse to 2 mm above the circumference
N13	Positioning in rapid traverse to length Z-40 and angle C = 25°
N14	New approach of the milling tool 2 mm into the piece (diameter X = 76)
N15	Linear movement to length Z-15
N16	Moving out of the keyway in rapid traverse to 10 mm above the piece (diameter X = 100)

G112

MILLING ON CIRCUMFERENCE of a circular arc in clockwise direction (CW), G112
If, by means of the driven tools, a circular milling movement is to be performed on the circumference of the piece, with given feed speed and in clockwise direction, then this machining process is to be performed using function G112.

Required addresses
After selection of path condition G112 the control system requires the following inputs:



LENGTH Z:
Nominal position in Z-direction on the circumference of a piece.

ANGULAR VALUE C:
Under this address is indicated the angular value (in degrees) of the nominal position.
Angular values ranging from -9999.999° to +9999.999° may be programmed here.

RADIUS R:
The radius of the circle to be milled is entered under this address. The radius entry is to be used only if the circle to be milled represents a traversing angle not equal to zero but smaller than/equal to 180°
(0° < R ≤ 180°).
Where larger traversing angles are involved, this form of entry is not permitted. Complete circles are also only to be programmed with the circle middle-point entries K and J. If the circle middle-point entries K and J are to be or have to be used, the address R must be confirmed; the control will then offer the address parameters K and J. If a radius is entered, addresses K and J are not applicable.

G112

CENTER OF CIRCLE (Z) K:
Here the distance between the starting point (actual value) and the center of the circle in Z-direction is indicated.
When viewed from the starting point parallel to the coordinate axis in direction of the center of the circle, the following applies:

In direction of the Z-axis	K+
In opposite direction of the Z-axis	K-

CENTER OF CIRCLE (C) J:
Here the distance between the starting point (actual value) and the center of the circle is indicated as angle of rotation (see figure to the right)

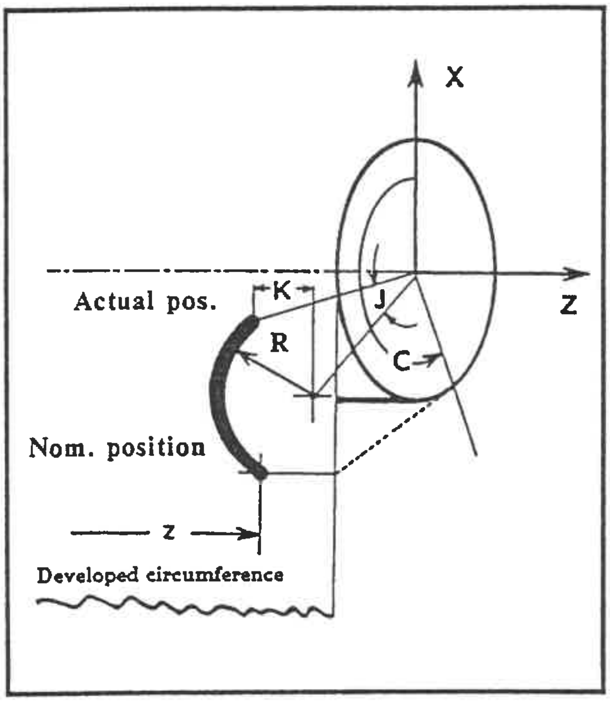
Angle in counter-clockwise direction	J+
Angle in clockwise direction	J-

Attention: This is an angular value and **not** the distance on the developed circumference.

G113

MILLING ON CIRCUMFERENCE of a circular arc in counter-clockwise direction (CCW), G113
If, by means of the driven tools, a circular milling movement is to be performed on the circumference of a piece, at a given feed speed and in counter-clockwise direction, then this machining process is to be programmed using function G113.

Required addresses
After selection of path condition G113 the control system requires the following inputs:



LENGTH Z:
Nominal position in Z-direction on the circumference of the piece.

ANGULAR VALUE C:
Under this address the angular dimension of the nominal position is indicated in degrees.
Angular values ranging from -9999.999° to +9999.999° may be programmed here.

RADIUS R:
The radius of the circle to be milled is entered under this address. The radius entry is to be used only if the circle to be milled represents a traversing angle not equal to zero but smaller than/equal to 180°
(0° < R ≤ 180°).
Where larger traversing angles are involved, this form of entry is not permitted. Complete circles are also only to be programmed with the circle middle-point entries K and J. If the circle middle-point entries K and J are to be or have to be used, the address R must be confirmed; the control will then offer the address parameters K and J. If a radius is entered, addresses K and J are not applicable.

G113

CENTER OF CIRCLE (K) K:
Here the distance between the starting point (actual value) and the center of the circle in Z-direction is indicated. When viewed from the starting point parallel to the coordinate axis in direction of the center of the circle, the following applies:

In direction of the Z-axis	K+
In opposite direction of the Z-axis	K-

CENTER OF CIRCLE (C) J:
Here the distance between the starting point (actual value) and the center of the circle is indicated as angle of rotation (see figure on the right).

Angle in counter-clockwise direction	J+
Angle in clockwise direction	J-

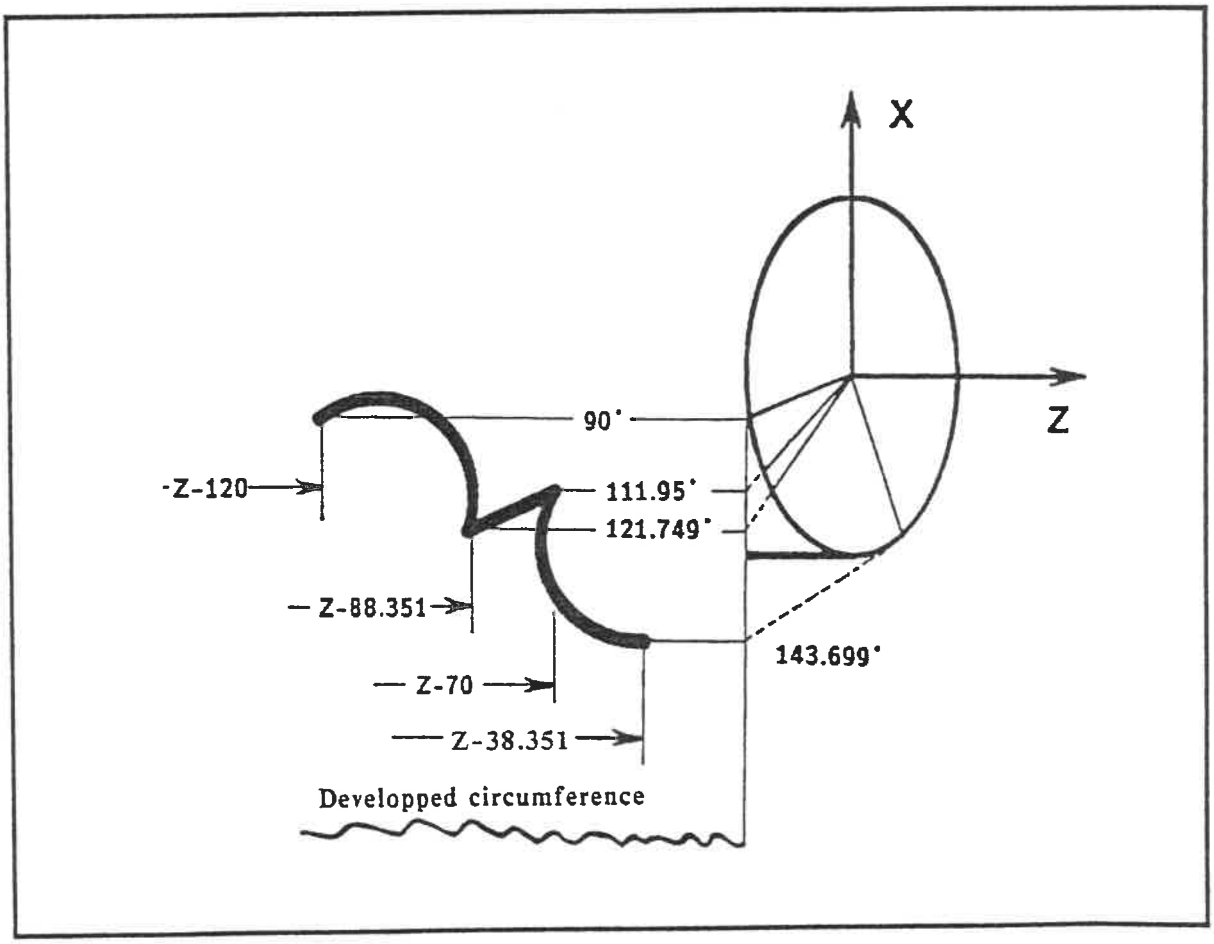
Attention: This is an angular value and **not** the distance of the developed circumference.

G112/113

Example
Milling of a keyway of 2 mm depth on the circumference of the piece (diameter = 84mm)

Programming

```
...
N 6  M14
N 7  G110  Z-120  C90
N 8  G94  F30  G1  X80
N 9  G112  Z-88.351  C121.749  K12.856  J21.95
N10  G111  Z-70  C111.95
N11  G113  Z-38.351  C143.699  K18.794  J9.799
N12  G0  X100
...
```



G112/113

Explanation

The diameter of the workpiece is 84 mm. Since a keyway with a 2 mm diameter is to be milled, the milling tool has to approach 2 mm into the workpiece. Therefore all calculations for the address J refer to a diameter of 80 mm. When the spindle is turned by 1°, then this angle of rotation corresponds to a distance of 0.6981 mm on the circumference (referred to a diameter of 80 mm).
Both circular arcs which are to be milled should correspond to a sector of 150° and they should each have a radius of 20 mm.

...

N 6	Swivelling-in of C-axis 1 and reference run
N 7	Move in rapid traverse above position C90 and Z-120
N 8	Feed 30mm/min; 2mm infeed to the piece (X80)
N 9	Circular arc in clockwise direction to position Z-88.351 and C121.749; distance between starting point and center of the circle in Z-direction 12.856 mm (K12.856) and sense of rotation 15.321 mm according to an angle of rotation of 21.95° (J21.95)
N10	Linear movement to position Z-70 and C111.95
N11	Circular arc in counter-clockwise direction to position Z-38.351 and C143.699; distance from starting point to center of the circle in Z-direction 18.794 mm (K18.794) and sense of rotation 6.84 mm corresponding to an angle of rotation of 9.799° (J9.799)
N12	Moving out of keyway in rapid traverse to 8 mm above the piece

...

Driven tool G126

Speed limitation of auxiliary spindle 1, G126
With this function the speed of the auxiliary spindle can be limited during the course of the program.

Required addresses
After selecting function G126, the control system requests the following input.

SPEED S:

Meaning
With this function the speed of the auxiliary spindle can be limited to a speed value in [1/min] entered under address S. This speed programmed under G126 takes effect until the end of the program or until replaced by renewed programming of G126.

Note
If the speed programmed with G126 is **higher** than the maximum speed fixed under **Parameter N541** (address **MD**), then the speed limitation from **Parameter N541** is treated with priority.

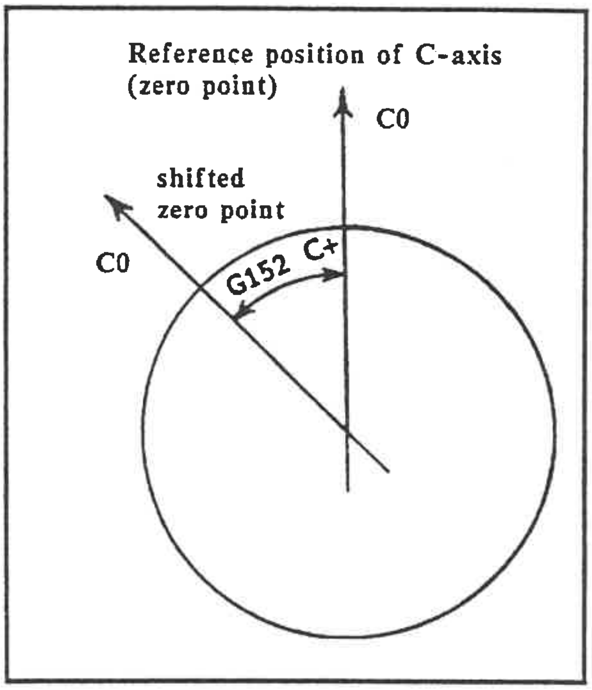
Zero point shift C-axis G152

Zero point shift of the C-axis, G152
With function G152 the zero point of the C-axis, angle of rotation of the NC part dividing mechanism, can be displaced.
The starting point for this zero point shift is always the reference point of the C-axis.

Required addresses:
After selection of path condition G152 the control system requires the following input:

ANGULAR VALUE C:
Under address C is indicated the angle of rotation in degrees, either with positive or negative sign, depending on the sense of rotation.
Angular values ranging from -9999.999° to +9999.999° can be indicated here.
It must be noted that the sense of rotation of the angle which is to be traversed theoretically on the frontface of the piece and which is programmed, is opposite to the actual sense of rotation of the spindle.

By entering the value 0 under address C a previously programmed zero point shift can be cancelled.

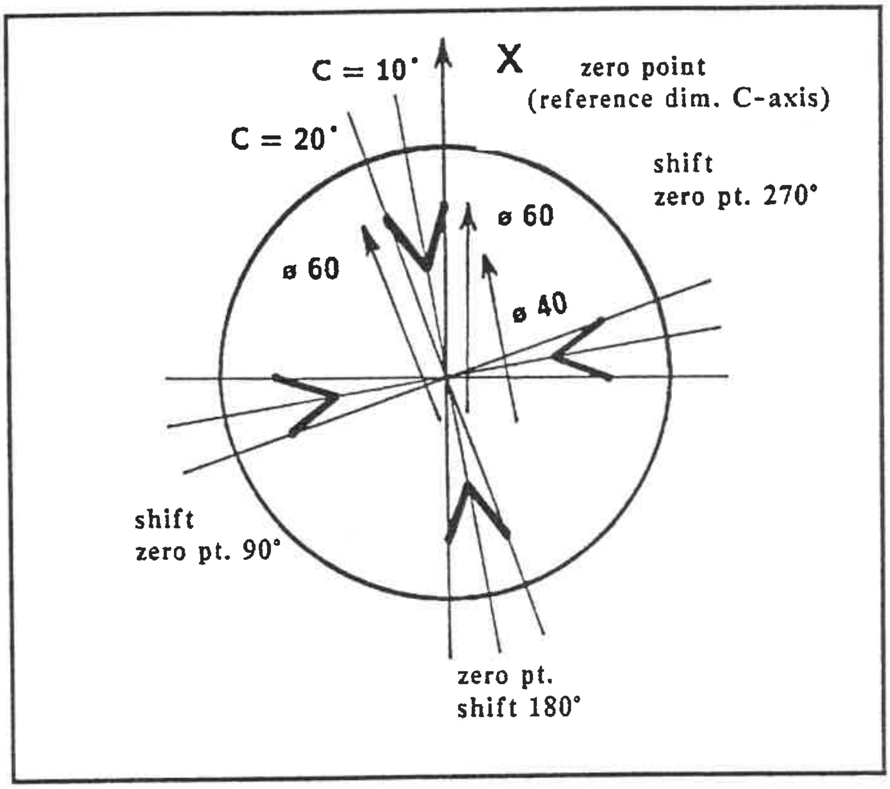


Range of application:
If the same contour is to be milled at different positions on the frontface or the circumference of the workpiece, then the contour to be milled can be programmed in a subprogram.
The subprogram with the contour to be milled is called up several times after the corresponding zero point shift of the C-axis.

G152

Example
Milling a contour several times on the frontface of a workpiece. The contour is written in the subprogram L2804 and is called up in the main program %1 after the corresponding zero point shifts of the C-axis (G152) were programmed.

Programming



Main program %1

```
...
N 5  M14
N 6  G94  F30
N 7  L2804
N 8  G152  C90
N 9  L2804
N10  G152  C180
N11  L2804
N12  G152  C270
N13  L2804
...
```

Subprogram L2804

```
N1  G100  X60  C0
N2  G1  Z-2
N3  G101  X40  C10
N4  G101  X60  C20
N5  G0  Z2
N6  M30
```

G152

Explanation

Main program

...

N 5	Swivelling-in of the C-axis
N 6	Feedrate 30 mm/min
N 7	Call-up of the subprogram (machining the contour for the first time)
N 8	Zero point shift of the C-axis to 90°
N 9	Call-up of the subprogram (machining the contour for the second time)
N10	Zero point shift of the C-axis to 180°
N11	Call-up of the subprogram (machining the contour for the third time)
N12	Zero point shift of the C-axis to 270°
N13	Call-up of the subprogram (machining the contour for the fourth time)

...

Subprogram L2804

N1	Positioning the C-axis to diameter X60 and to zero point (reference dimension)
N2	Approach by 2mm into the piece
N3	Linear milling movement to diameter X40 and angle 10° (viewed from the zero point)
N4	Linear milling movement to diameter X60 and angle 20° (viewed from the zero point)
N5	Moving out of the piece in rapid traverse
N6	End of the subprogram

Driven tool
G193, G195

Feed for auxiliary drive 1
There are three programming options for the feed of auxiliary drive 1: in mm/min (drive-independent with the function G94), in mm/rev and, where milling tools are involved, in mm/tooth also.

Feed of auxiliary drive 1 in mm/tooth, G193
The feed of the milling tool (auxiliary drive 1) programmed under address F is executed in mm/tooth by programming the function G193.

The feed version is permitted only for milling, i.e. it is imperative that the tool types **WT13**, **WT14** or **WT24** (milling tools) are selected. The special tool **WT0** can also be used if this is defined as a milling tool by appropriately programming the parameter addresses. Where other tool types are involved, the control sets the feed to zero. In addition, the number of teeth of the utilized milling tool must be specified beforehand under the address C in the tool file. Furthermore, the speed of auxiliary drive 1 has to be defined in advance in the parts program. The control uses the speed programmed with G197 or the cutting speed defined with G196 (which it also converts into a speed value in which the tool diameter is taken into account) and the number of the teeth on the milling tool in order to calculate a corresponding feed in m/min. Consequently, a feed in mm/min calculated in the control is indicated on the target-value/actual-value display, not a feed per tooth.

Feed of auxiliary drive 1 in mm/rev, G195
As a result of the function G195, the value programmed under F is executed as the feed in mm/rev referred to auxiliary drive 1. To this end, the speed of auxiliary drive 1 must be established beforehand in the parts program. Using the speed programmed with G197 or the cutting speed established G196 (also converted into a speed value by the control with the tool diameter being factored in), the control calculates a corresponding feed in mm/min. Consequently, the feed in mm/min calculated in the control is indicated on the target-value/actual-value display, not a revolving feed in mm/rev. This version of the feed is only to occur with driven tools, i.e. the selected tool type must be a milling tool or drill. If it is not, the control sets the feed value to zero.

Driven tool
G196, G197

Speed/cutting speed of auxiliary drive 1
The speed or cutting speed of the driven tool is programmed under the address S. Where machines with multi-stage auxiliary drive are involved, the desired gear stage is to be programmed beforehand. The gear stages are entered via M-functions (see Section 3.6).

Cutting speed S of auxiliary drive 1 in m/min, G196 (v-constant)
As the result of function G196 the value programmed under S is executed in m/min as the constant cutting speed of auxiliary drive 1.
This is therefore the relative working speed between the machine's drive surface and the workpiece to be machined.
The control uses the programming cutting speed S and the tool diameter indicated in the tool definition to calculate the necessary speed for the driven tool internally according to the relationship

$$v_s = \pi \cdot d \cdot n$$

The speed calculated in the control is therefore shown on the target-value/actual-value display, not the programmed cutting speed. This is only possible within the speed range of the selected gear stage.

Speed of the auxiliary drive 1, G197
The value for the speed of the auxiliary drive 1 can be programmed under address S by using function G197. The programmed value represents the number of revolutions per minute.

Notes on functions G196 and G197
G196 and G197 cancel one another. The two functions are self-maintaining.
A new S-value has to be programmed if you switch over from G197 to G196.
Defining the speed/cutting speed using the functions G196/197 is only permitted when driven tools are involved, i.e. the selected tool type must be a milling tool or drill with appropriately defined dimensions. Otherwise the control is unable to convert the cutting speed into a speed when the function G196 is involved and sets it to zero.
The speed can be limited by the drive mechanics, the workpiece, etc. To exclude the possibility of impermissible high speeds, particularly in the case of control-internal calculation of the speed from tool diameter and programmed cutting speed, the maximum speed should be preset in parameter N541 (see Section 6) or programming should be conducted with G126.

Second chuck
G226, G296, G297

Speed limitation of auxiliary spindle 2, G226
With this function the speed of the auxiliary spindle 2 can be limited during the course of the program.

Required addresses
After selecting function G226, the control system requests the following input.

SPEED S:

Meaning
With this function the speed of the auxiliary spindle 2 can be limited to a speed value in **[1/min]** entered under address S. This speed programmed under G226 takes effect until the end of the program or until replaced by renewed programming of G226.

Note
If the speed programmed with G226 is **higher than** the maximum speed fixed under **Parameter N581 (address MD),** then the speed limitation from **Parameter N581** is treated with priority.

Cutting speed S of the auxiliary spindle 2 in m/min, G296 (V - constant)
G296 causes the value programmed under S to be taken as speed of the auxiliary spindle 3 in m/min.
The spindle speed depends on the X-position of the tool tip whose slide last called up G396 so that the cutting speed remains constant.
This is possible only within the speed range of the selected gear stage.
G396 and G397 cancel each other. Both functions are self-maintaning.
When switching over from G396 to G397, a new S-value must be programmed as the old value will otherwise be accepted with false designation.
Speed assignment to the slides is achieved in the part program by means of the $-sign.

Speed of the auxiliary drive 2, G297
The value for the speed of the auxiliary drive 2 can be programmed under address S by using function G297. The programmed value represents the number of revolutions per minute.

PLC-FUNCTIONS
G600 to G699

G-functions to transfer the address parameters to the PLC, G600 - G699
The functions G600 to G699 constitute a range of 100 G-functions with 7 addresses each whose meaning is not defined by the NC. While the program is being processed in the operating modes AUTOMATIC and INDIVIDUAL RECORD, the values the operator programs under these addresses are transferred to the PLC.
The meaning of the individual G-functions and their address parameters is defined by appropriately configuring the machine interface (PLC).

Requested addresses
After one of these functions is selected the control asks the operator to enter the following address parameters; addresses which are not required can be confirmed.

WORD X:
WORD Z:
WORD C:
Values can be entered in the input format 4 positions before and 3 after the decimal point, with prefix.

WORD T:
WORD F:
A six-position integer without prefix can be entered under these addresses.

WORD S:
A four-position integer without prefix can be entered under these addresses.

Note:
The functions G600 to G699 can also be executed in the MANUAL CONTROL operating mode.

G900, G901

The funktions G900 to G999
Apart from their actual purpose, all functions from the "nine hundred range" except functions G908 and G915 also produce an interpreter stop, i.e. the function G909.

The functions G900 - G999 are devided into three groups:

G900 - G949	**pre-geometry functions**
G950 - G979	**geometry functions**
G980 - G999	**after-geometry functions**

ATTENTION:

Only one function from the group G900 to G999 may be programmed in one NC-block!

Calculation of return point for inspection cycle, G900
In the course of an inspection cycle the control system calculates the return point using the internally stored paths (use of manual direction keys). This can be realized using the function G900.

This function is implemented in the stategy program "Service 1" and is of no importance when editing parts programs (function is only mentioned here to complete the list).

Transfer of actual values to the variables memory, G901
By programming the function G901 the current actual values can be transferred to the variables memory.

Required addresses
After selecting function G901 the control system requests no other inputs.

Procedure
The actual value in X is transferred to the variable V937, the actual value in Z to the variable V938.

G902

Transfer of complete current zero point shift to variables memory, G902
All current zero point shifts can be transferred to the variables memory by programming the function G902.

Required addresses
After selecting function G902 the control system requires no other inputs.

Procedure
The sum of all current zero point shifts in X-direction is transferred to the variable **V937**, the sum of all current zero point shifts in Z-direction to the variable **V938**.

Speed monitoring G907

Blockwise deactivation of speed monitoring, G907
Each time there is a change from paths without spindle movement (G0) to paths with spindle movement, and also after the execution of T- or M-functions which interrupt the machining process as such, the speed is monitored internally by the control system. This means that a machining process including spindle movement will only be started after the spindle has reached the programmed speed (taking into account the defined tolerance range). This can lead to machining errors, e.g. when retracting in a threading cut. For applications like the one mentioned above there is a possibility to switch off speed monitoring blockwise via function G907.
Thus a machining process will be started as soon as the relevant G-function has been recognized by the control system even if the programmed nominal speed has not yet been reached.

Required addresses
After selecting the function G907 the control system requests no further inputs.

Programming
The function G907 is effective per block and has to be programmed in one NC-block together with the movement that is to be executed without speed monitoring.

Example
A tapered external thread is to be machined in longitudinal direction using a threading tool. Finally, the tool is to be retracted in the thread.

```
...
N...  T10
N...  G97 S300  M3
N...  G0  X...  Z...
N...  G33  X30  Z-20  F1.5
N...  M5
N...  G907  G33  X15  Z0  F1.5  M4
...
```

G908, G909

Set feed value 100%, G908

Additional programming of function G908 in a block including a slide movement will lead to the control system disregarding the feed changes via handwheel in AUTOMATIC MODE; instead the programmed feed value will be executed at 100%. This is true for feed speed and for rapid traverse movements..
Please note that the function G908 works only for one block each. That means that in the following block without G908 the programmed feed override will again be taken into account, otherwise G908 has to be programmed once more.

Note

If a feed override is entered via handwheel while executing a block with an active G908, this feed change will be stored and is active in the next block without G908.

Interpreter stop, G909

While executing NC-programs, the control system (pre-interpreter) reads and computes not only the next block to be executed by the machine, but as many as 15 to 20 blocks in advance.
This so-called pre-interpretation can be disabled by programming the function G909.
This can be necessary, for example, if variable assignments which may change in the course of a program are to be processed in a program block. Here, it may occur that a part of the program before the variable change is executed with the new variable assignment although the old value is required or vice versa.

Programming

The function G909 must be programmed before the variable assignment the new value of which is not to be processed by the control until the preceding part of the program has been executed by the machine.
The block in which G909 is programmed should not include any other functions.

G910, G911

Measuring the workpiece, G910
By the programming of function G910 the input of the primary element is activated for the measurement of the workpiece. At the same time a collision surveillance is switched on. When the primary element is deflected during pre-positioning, then an error message is displayed. Function G910 is active when stored. No other function may be programmed in a block containing G910.

Required addresses
After selection of function G910 the control system requests **no** other inputs.

Conditions
The condition is that the correct primary element is swivelled in and that the measuring electronic unit is switched on.

Measuring the tool, G911
By the programming of function G911 the input of the primary element is activated for the measurement of the tool. At the same time a collision surveillance is switched on. An error message appears when the primary element is deflected during pre-positioning.
Function G911 is active when stored. No other function may be programmed in a block containing G911.

Required addresses
After selection of function G911 the control system requests **no** further inputs.

Conditions
The condition is that the correct tool was swivelled in and that the electronic measuring unit is switched on.

G912, G913

Detection of the actual value, G912
The programming of function G912 leads to the switchover to detection of the actual value of the input of the primary element activated by G910 or G911. Function G912 is active when stored. No other function may be programmed in a block containing G912. The measuring distance must be chosen to guarantee the sufficient deflection of the primary element and it is programmed with a feedrate in minutes in connection with G1.

Required addresses
After the selection of function G912 the control system requires no further inputs.

End of measurement, G913
By the programming of this function the measurement is terminated. Besides, an error evaluation takes place. Function G913 is active when stored. No other function may be programmed in a block containing G913.

Required addresses
After selection of function G913 the control system requires no further inputs.

Tapping
G915

Monitoring the reversal of rotation during tapping, G915
(only when the main spindle is controlled via the AP186)
The cutting of threaded holes may involve the danger of breaking the tool when the tap is retracted while the sense of rotation of the spindle concerned is incorrect.
In order to avoid this, function G915 was created, combining **both functions** G907 (blockwise deactivation of speed monitoring) **and** G908 (blockwise setting of the feed value to 100%) (see descriptions below).

Blockwise deactivation of speed monitoring, G907
The speed of rotation is monitored internally by the control system each time there is a change from paths without spindle movement (G0) to paths with spindle movement, and also after the execution of T-commands, M-functions or pre-geometry functions G900 to G949, which interrupt the machining process as such, and after blocks which cause a machining operation of less than 20 ms.
This means that a machining process including spindle movement will only be started after the spindle has reached the programmed speed (taking into account the defined tolerance range).
For applications like the one mentioned above there is a possibility to switch off speed monitoring blockwise via function G907.
Thus a machining process will be started as soon as the relevant G-function has been recognized by the control system even if the programmed nominal speed has not yet been reached.

Set feed value 100%, G908
Additional programming of function G908 in a block including a slide movement will lead to the control system disregarding the feed changes via handwheel in AUTOMATIC MODE; instead the programmed feed value will be executed at 100%. This applies to feed speed and rapid traverse movements.
Please note that the function G908 works only block by block. That means that in the following block without G908 the programmed feed override will again be taken into account, otherwise G908 has to be programmed once more.

Note
If a feed override is entered via handwheel while executing a block with an active G908, this feed change will be stored and is active in the next block without G908.

G915

Required addresses
After selection of the function G915, the control systems requests no further inputs.

Meaning
The programming of function G915 causes the control to monitor the sense of rotation of the spindle.
The feed rate release (e.g. for retracting the tap from the workpiece) is given only after the actual sense of rotation (real sense of rotation) corresponds to the programmed sense of rotation.

Example

- main spindle

N10 T01
N11 G97 G00 X0 Z5 S1000 M03
N12 G908 G01 G95 Z-25 F1.25
N13 ***G915 G01*** G95 Z5 F1.25 M04
N14 G00 X100 Z50
N15 M30

Note:
Contrary to the other functions of the "nine hundred range" function G915 does not generate an interpreter stop.

G920, G921

Inactivating active zero point shift, G920
After the programming of function G920 the zero point shifts activated by the parts program are inactivated.
All traverse paths are now referred to the zero point of the machine. Function G920 is active until it is cancelled by function G980.

Required addresses
After selection of function G920 the control system requires no further inputs.

Note
If a traverse movement is to be performed without active zero point shift, then the programming can be simplified as shown in the example for G980.

Converting the system of dimensioning to slide position, G921
The programming of function G921 leads to the temporary cancellation of tool specific shiftings of the measuring system (these shiftings are, however, stored internally) such as

- ZERO POINT SHIFT
- TOOL LENGTHS
- TOOL CORRECTIONS
- POSITION CORRECTION

which are again effective when G921 is cancelled by G981.
The contours programmed afterwards no longer refer to the tool tip, but to a fixed point in the slide.
Thus the tool slide can be positioned in the workspace of the machine, e.g. for coupling and traversing a tailstock or a quill, without taking into consideration the tool-specific shiftings mentioned above.

Important: No tool change and no zero point shift may be programmed while function G921 is active since by programming G981 the previous tool-specific shiftings which are temporarily stored by G921 will be reactivated

- Danger of collision due to incorrect tool data-

Block display, tool life G940, G941, G943, G944

Switch off block display, G940
By programming the function G940 the block display (three future blocks of the parts program plus the current block and one already executed by the control system) can be switched off while in the AUTOMATIC CONTROL or in the SINGLE BLOCK operating mode.
This function is applied in the strategy programs supplied by the lathe manufacturer, customer-specific variable programs which begin with the letter "E" and are therefore protected (copy and editing protection), and in program parts which contain only numerical calculations without traversing paths of the machine.

Switch on block display, G941
The block display can be switched on again using function G941. The block display function is automatically active when switching on the control system.

Switching the tool life monitoring on, G943
By programming function G943, the tool life can be monitored both during feed in revolutions (G95) and during feed in minutes (G94).
This monitoring takes effect from the start of the program.

Switching the tool life monitoring off, G944
By programming function G944 the tool life monitoring programmed with G943 can be temporarily switched off for the tool in action at a given moment. This can be useful, for example, when a tailstock (optional equipment) is to be traversed in coupled-motion using a strategy program. Following that, the monitoring can be reactivated with G943.
The monitoring can also be switched on again (during inactive monitoring) by means of a programmed tool change.

Graphics
G970, G971

Sector limits for graphic representation, G970
The sector limits for the graphic representation can be altered via the parts program by programming the function G970.

Required addresses
After selecting the function G970, the control system requests the following addresses:

I: GRAPHIC END X>
X: GRAPHIC END X<
K: GRAPHIC END Z>
Z: GRAPHIC END Z<

Procedure
After executing the block with G970, the sector limits that are specified under the requested addresses are applied for graphic simulation. Simultaneously, the parameter block N70 is overwritten with the values programmed here.

Note
The function G970 must be programmed in a separate NC block. Function G970 takes no effect when the magnification function is switched on.

Dimensions of blanks for graphic representation, G971
The dimensions of blanks can be altered for the graphic representation via a parts program by programming the function G971.

Required addresses
After selecting the function G971, the control system requests the following addresses:

I: BLANK DIMENSIONS -X
X: BLANK DIMENSIONS +X
K: BLANK DIMENSIONS -Z
Z: BLANK DIMENSIONS +Z

Procedure
After executing the block with G971, the blank dimensions that are specified under the requested addresses are applied for graphic simulation. Simultaneously, the parameter block N71 is overwritten with the values programmed here.

Note
The function G971 must be programmed in a separate NC block. Function G971 takes no effect when the magnification function is switched on.

G972, G973

Clamping area length for graphic representation, G972
By programming the function G972 it is possible to use parts programs to define how far the graphic representation is to show the piece clamped into the chuck.

Required addresses
After selecting the function G972, the control system requests the following address:

CLAMPING SURFACE LENGTH Z

Procedure
After executing the block with G972 the clamping area length specified under the requested address is applied for the graphic simulation. Simultaneously the parameter block N72 is overwritten with the value programmed here.

Note
The function G972 must be programmed in a separate NC block. Function G972 takes no effect when the magnification function is switched on.

Scope of the graphic representation, G973
The scope of graphic simulation can be established via parts programs by programming the function G973.

Required addresses
After selecting the function G972 the control system requests the following address:

GRAPHICS <0=NOTHING, 1=BLANK, 3=WITH CLAMP, 7=TAILST> Q
Altogether the following possibilities exist, depending on the input under Q

0 = only traversing paths
1 = contour, blank
2 = contour of clamp
3 = contour of blank, clamp
4 = contour, tailstock
5 = contour, blank, tailstock
6 = contour, clamp, tailstock
7 = contour, blank, clamp, tailstock

Procedure
After executing the block with G973, the size of representation specified under address Q is applied for graphic simulation. Simultaneously, the parameter block N73 is overwritten with the value programmed here.

Note
The function G973 must be programmed in a separate NC block. This function takes no effect when the magnification function is switched on.

G980, G981

Reactivating the zero point shift, G980
After the programming of function G980, all zero point shifts programmed previously in the part program and inactivated by G920 are re-activated.

Example

N...
N... G920 G0 X150 Z200 G980
N...

The traverse path G0 X150 Z200 refers to the zero point of the machine.

Setting the system of dimensioning back to tool-specific shiftings, G981
Function G981 cancels the effect of G921, that is, the tool-specific shiftings

- ZERO POINT SHIFT
- TOOL LENGTHS
- TOOL CORRECTIONS
- POSITION CORRECTION

are again effective.

Simplified

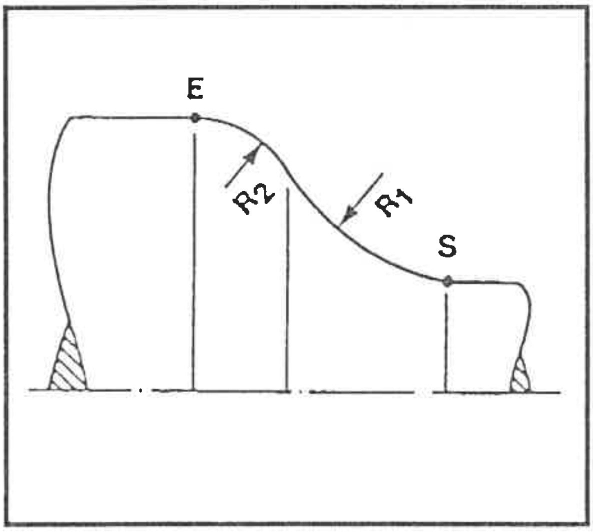
Geometry Programming

SGP

Introduction
Production drawings are not always dimensioned in a manner suited to NC. Programming of the contour is often only possible when further values such as intersection points, coordinates and circle centers are calculated by the user.
Simplified Geometry Programming (SGP) can often simplify the programming of difficult contours, as the control system calculates - where mathematically possible - values not included in the drawing. The control system can use, in advance, a **maximum** of 5 traverse paths in its calculations.

If this limit is reached before calculation of the block is completed, an error message is displayed. After the contour has been entered, this can be verified only after the cursor has once been moved away from the block entered last.

In the illustrated contour, it would suffice to program the coordinates of starting point S, the X-coordinate of the end point E and the two given radii, provided that there is a **tangential** transition at S and E.



Not required are:
circle centers, the point of tangential transition between the two circles, Z coordinate of the end point.

SGP calculates:
- coordinates entered as "?",
- chamfers and roundances with B,
- I and K for circles for which only R has been entered.

How to activate SGP is described on page 3.3-6.

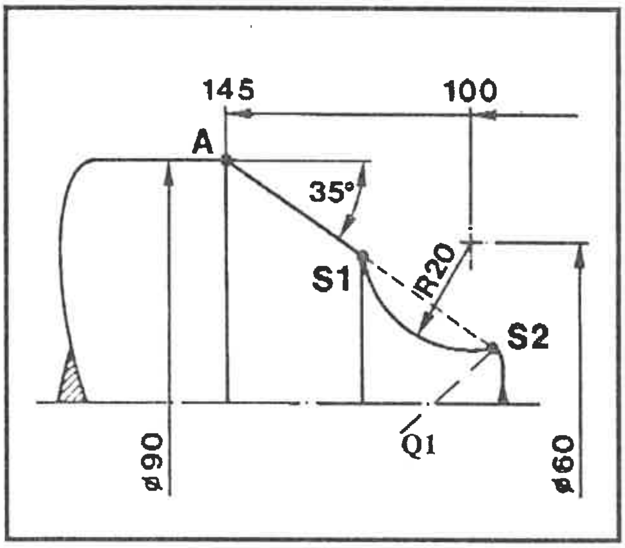
SGP

Programming
All contours can be described using path functions G1, G2, G3, G12, and G13.
The contours should be programmed in absolute dimensions (G90).
Programming with G91 is **only** possible when values given in incremental dimensions do **not** relate to a value to be calculated.

Alteration of the zero point, paths of length zero and blocks including cycle functions G80 to G83 may not be positioned directly before or after blocks in which SGP is used.

If several possible solutions arise when calculating an end point, the control system reacts to SGP as described in the following examples:

Example: non-tangential transition between straight line and circle
Known straight line data: start point A and pitch.
Known circular arc data: center point M and radius.
If the end point of the circular arc is unknown, the control system selects the point of intersection which is closer to the start point of the straight line, i.e. S1.

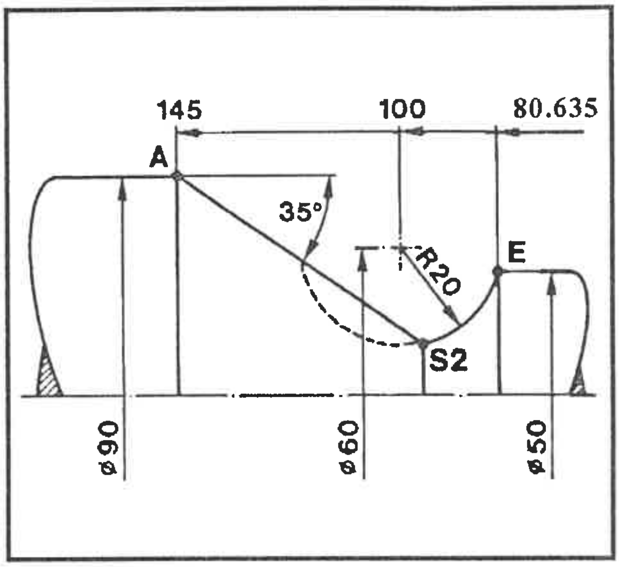


Each **non-tangential** transition must be programmed using B0. B0 is not necessary when: there is a transition between two straight lines, or when no destination coordinates have to be calculated in the preceding path, the current path **and** in the following path.

SGP

If the end point E of the circular arc is known, the point of intersection is selected which is closer to E, i.e. S2.

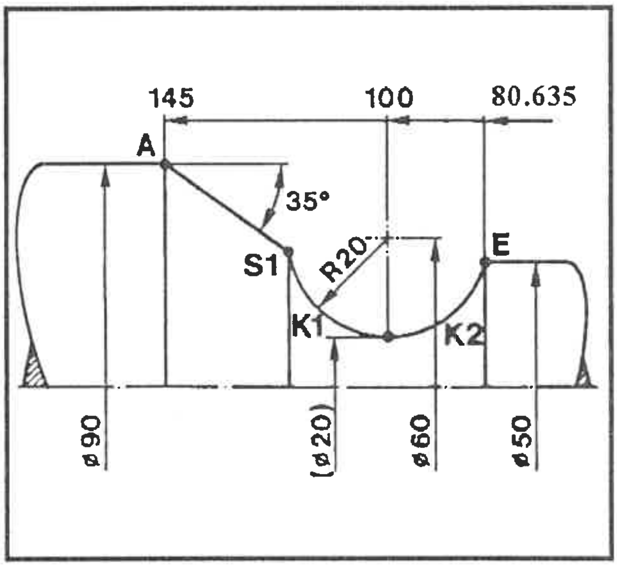
N 10 G1 X90 Z-145
N 20 G1 X? Z? A215 B0
N 30 G13 X50 Z-80.635 I30 K-100 R20
N 40 G1 Z0 A180



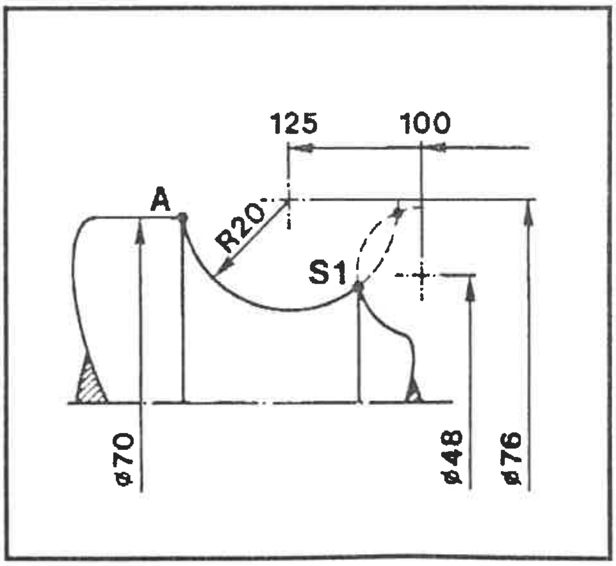
If intersection point S1 is desired, the circular arc must be programmed in two contour elements (e.g. K1, K2).

N 30 G13 X20 Z-100 I30 K-100 R20
N 35 G13 X50 Z-80.635 I30 K-100 R20
N 40 G1 Z0 A180

alternatively
N 10 G1 X90 Z-145
N 20 G1 X? Z? A215 B0 Q1
N 30 G13 X50 Z-80.635 I30 K-100 R20
N 40 G1 Z0 A180

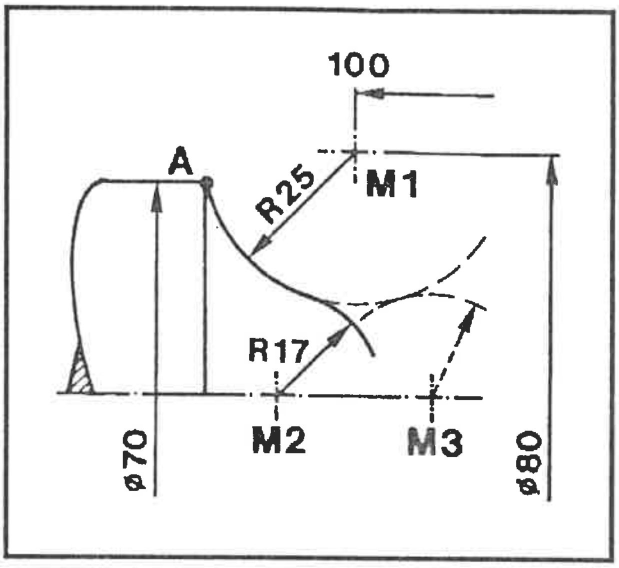


Circle-circle intersection point with non-tangential transition
The center points and radii of the circular arcs are known. Programming of such a contour will only remain error-free if start point A is entered or can be calculated.
The point of intersection is selected which will produce a shorter first circular arc, i.e. S1.



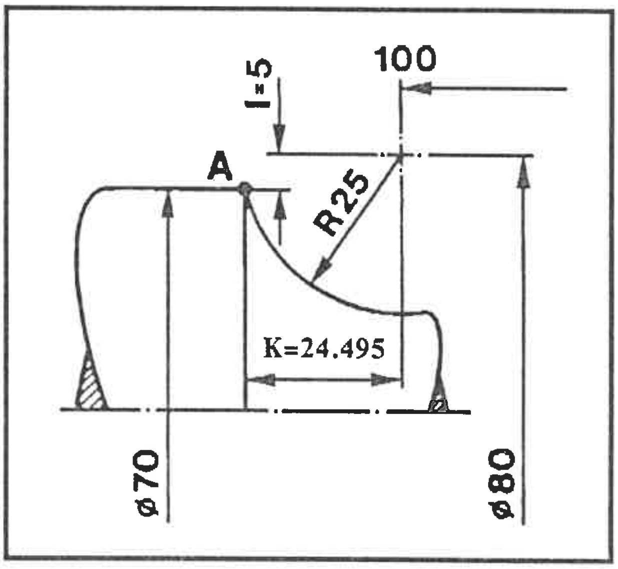
SGP

Circle-circle intersection point with tangential transition
Known data of first circular arc: Center point M1, radius and start point A.
Known second circle data: radius and only one center point coordinate.
Of the second circular arc, the center point is selected which is closer to start point A, i.e. M2.

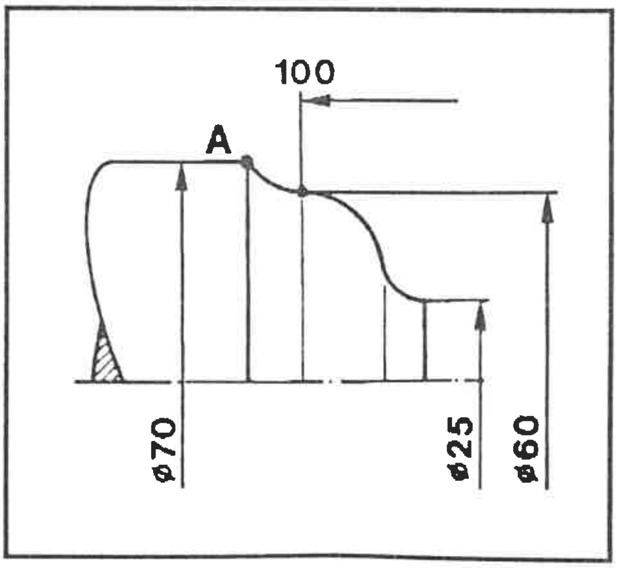


Excess data
If values are to be calculated for the start point A of a circular arc, I and K should be entered (if known).

Note: Entering more data than is required by the control system (excess data) will not lead to an error.



If start and end point of a contour element are not fully known, and if a point between them is known, it should be entered, whereby the contour element in question should be programmed in two parts.



SGP

Activating simplified geometry programming

If a contour is to be programmed, all dimensions given on the drawing should be entered in the control system.
Unknown addresses I, K, A and R are omitted, a question mark is entered for X- and Z-values.

E.g.
N... G2 X? Z50 R7 B0

	Press the G FUNCTION DIRECT softkey.	ENTER G FUNCTION:
	Enter digits (2)	
	Confirm.	G2 CIRCLE CW END POINT X:
>	Press "Continue".	The following submenu is displayed:

VARIABLE EXPRESS. "{...}"		SIMPLIF. GEOMETRY "?" <

	Press the SIMPLIF. GEOMETRY "?" Softkey.	END POINT X: ?
	Confirm.	END POINT Z:
	Enter digits.	END POINT Z: 50

SGP

Confirm. RADIUS R:

Enter digits. RADIUS R: 7

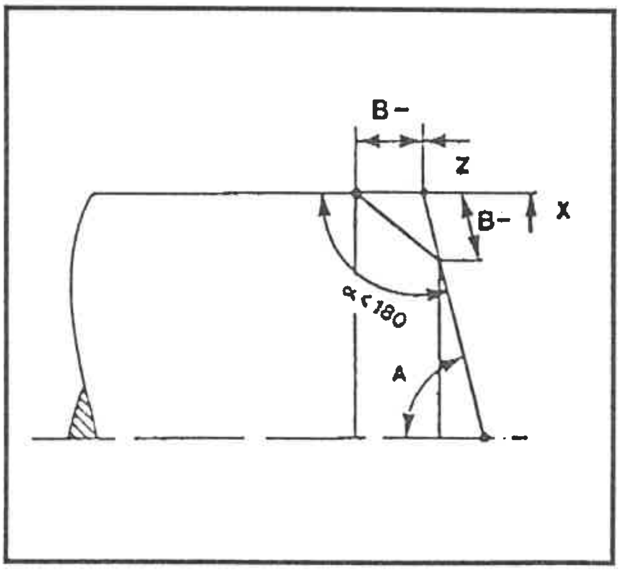
Confirm. CURVE = B+/N.TANG. = B0:
CURVE = B+/N.TANG. = B0: 0

Enter digits.

Confirm. Special feed E:

Confirm.

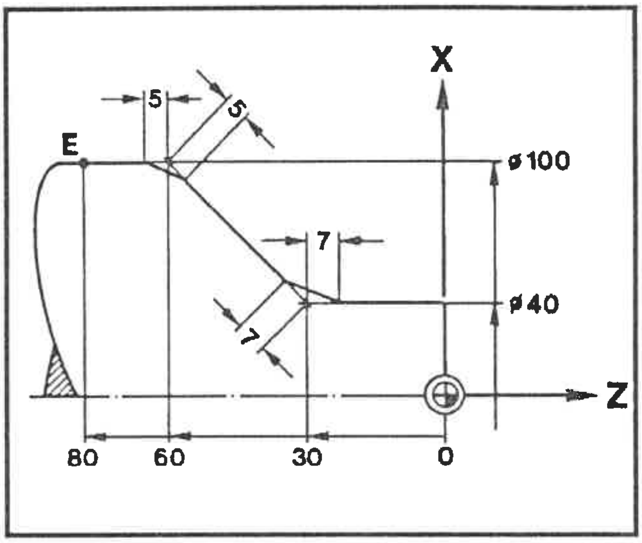
Example for straight line / chamfer
If a chamfer is located between two straight lines, it need not be programmed in a separate block with G88.
The contour is first programmed without the chamfer and, under X and Z, the value is programmed which would result without the chamfer.
In the block with G1, the chamfer width is entered under B with negative sign.
A separate feed for chamfer and roundance can be programmed under address E.
If the value is zero or merely confirmed upon entering, no other feed value is generated.



SGP

Programming

N 1 G0 X100 Z2 G96 S150 M4 T1 F0.3
N 2 G818 X40 I3
N 3 G1 X40 Z0
N 4 G1 Z-30 B-7
N 5 G1 X100 Z-60 B-5
N 6 G1 Z-80
N 7 G80
N 8 G0 X120 Z20
N 9 M30



Explanation

N 1	Rapid traverse to starting point.
N 2	Longitudinal turning cycle.
N 3	Straight line.
N 4	Straight line with chamfer, chamfer width (B-7).
N 5	Straight line with chamfer, chamfer width 5 (B-5).
N 6	Straight line.
N 7	End of longitudinal turning cycle.
N 8	Rapid traverse away from workpiece.
N 9	End of program.

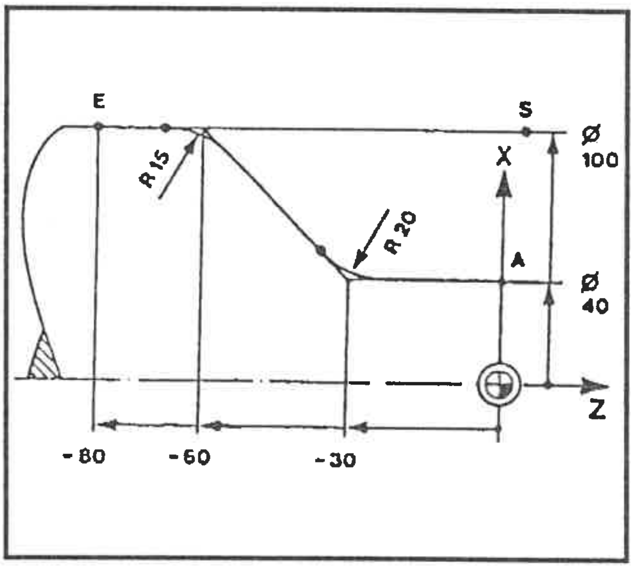
A separate feed can be programmed under address E.
If the value is zero or merely confirmed upon entering, no other feed value is generated.

SGP

Example of straight line / roundance
If a curve is located between two straight lines, it need not be programmed in a separate block with G2 or G3.
The contour element is first programmed without the curve and, under X and Z, the value is written which would result without the curve.
In the block with G1, the curve radius is entered under B+.

Programming:

```
N 0 G90  G95  F0.5  T1  M4  M8  G96  S150
N 1 G0  X100  Z2
N 2 G818  X40  I3
N 3 G1  X40  Z0
N 4 G1  Z-30  B20
N 5 G1  X100  Z-60  B15
N 6 G1  Z-80
N 7 G80
N 8 G0  X120  Z20
N 9 M30
```



Explanation

N 0 Start conditions.

N 1 Rapid traverse to starting point.

N 2 Longitudinal turning cycle.

N 3 Straight line.

N 4 Straight line with curve, curve radius 20 (B20).

N 5 Straight line with curve, curve radius 15 (B15).

N 6 Straight line.

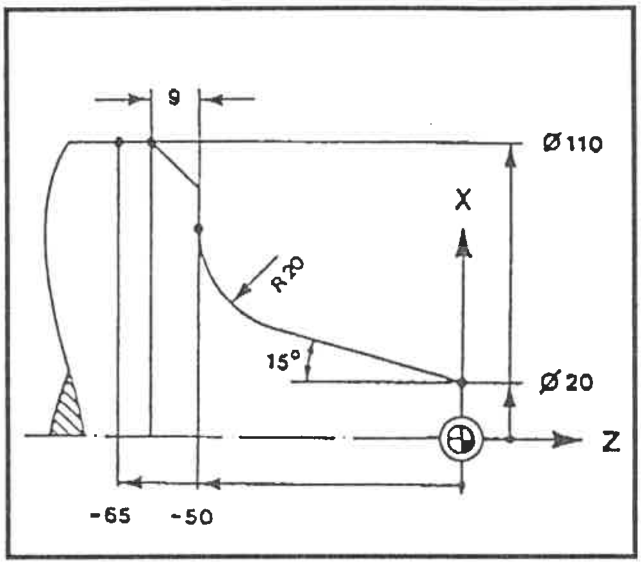
N 7 End of longitudinal turning cycle.

N 8 Rapid traverse away from workpiece.

N 9 End of program.

SGP

Example for chamfer and roundance
Turning a workpiece from solid material. Roundance and chamfer are programmed with G1.



Programming:

N 1 G0 X120 Z5 G96 S150 F0.3 M4
N 2 G818 X0 I3.5
N 3 G1 X0 Z0
N 4 G1 X20
N 5 G1 X? Z-50 A15 B20
N 6 G1 X110 Z? A90 B-9
N 7 G1 Z-65
N 8 G80
N 9 G0 X120 Z5
N10 M30

Explanation

N 1	Rapid traverse to starting point. Start conditions.
N 2	Longitudinal turning cycle with limited traverse path, start point of the contour, cutting depth 3.5 mm.
N 3	Straight line.
N 4	Straight line.
N 5	Straight line under 15° angle with subsequent roundance, curve radius 20 mm (B20), destination point in X unknown.
N 6	Straight line under 80° angle with subsequent chamfer, chamfer width 9 mm (B-9), destination point in Z unknown.
N 7	Straight line.
N 8	End of cycle.
N 9	Rapid traverse to starting point.
N10	End of program.

SGP

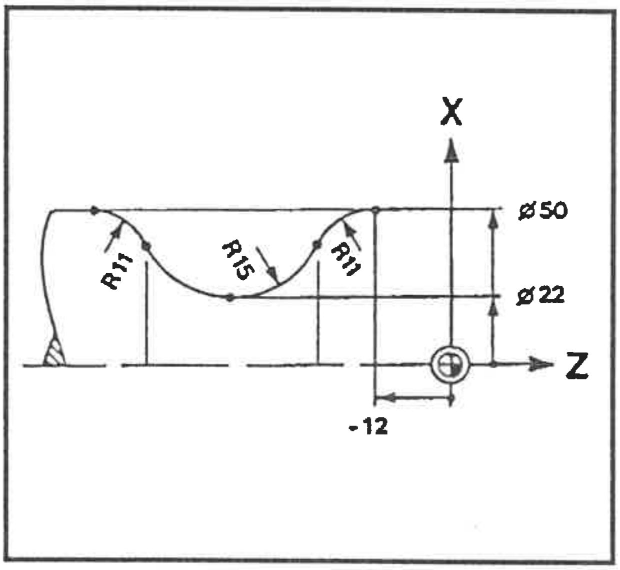
Circular arc as roundance or block?
As the previous examples have shown, it may be simpler and shorter to program a circular arc as a roundance, **but** this is bound with certain conditions:
The start and end point must have a **tangential** transition.
The contour elements involved must have a calculable point of intersection in X and Z.

Rule:
If more is known than just the radius, the circular arc should be programmed in a separate block using the corresponding G-function.

In the following example, the start point of the first arc with radius R11 is known. However, this start point is required by **SGP** in order to calculate the position of the circular arc with radius R15.

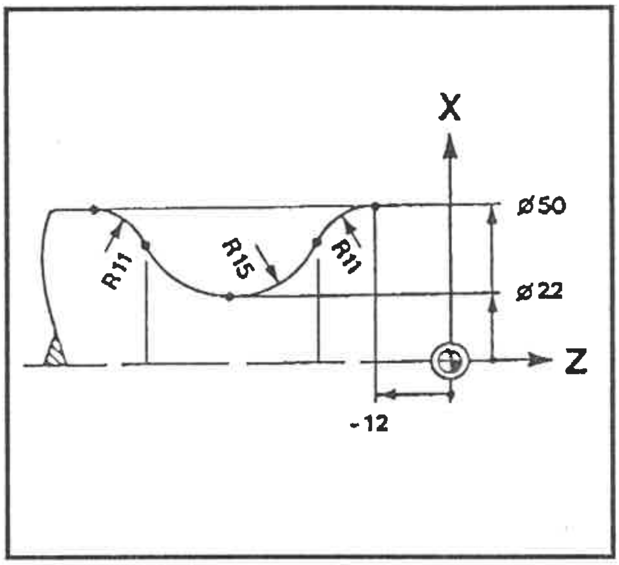
The circular arc must therefore be programmed in a separate block.

The two ways to program this are described in the following.



SGP

Programming (1st possibility)
Programming the circle and the roundance with radius R15 in one block.



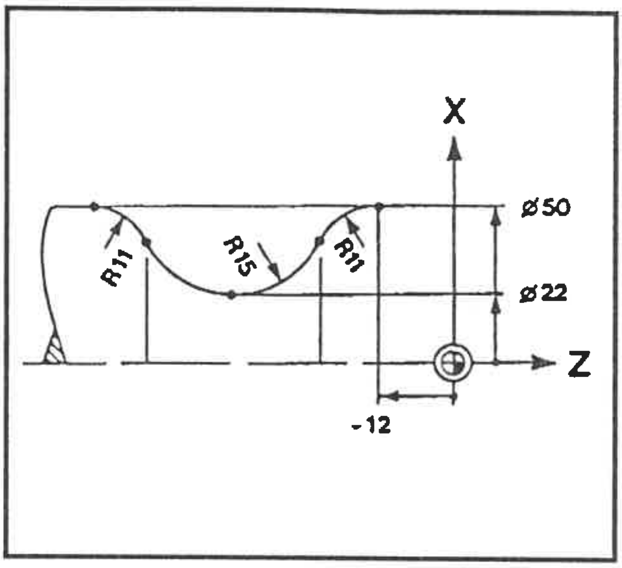

```
N 1 G0  X50  Z2
N 2 G1  X50  Z0
N 3 G1  Z-12
N 4 G3  X?  Z?  R11
N 5 G12  X?  Z?  I26  R15
N 6 G13  X50  Z?  I14  R11
N 7 G1  Z-80
N 8 G0  X100  Z50
N 9 M30
```

Explanation:

N 1	Rapid traverse to starting point.
N 2	Straight line.
N 3	Straight line.
N 4	Circular arc with radius R11, X- and Z-values of end point unknown.
N 5	Circular arc with radius R15, X- and Z-values of end point unknown. Center of circle (X) I26 (diameter 22 + 15).
N 6	Circular arc with radius R11, Z-value of end point unknown Center of circle (X) I14 (diameter 50 - 11).
N 7	Straight line.
N 8	Rapid traverse away from work-piece.
N 9	End of program.

SGP

Programming (2nd possibility)
Programming the circle with radius R15 in two blocks.



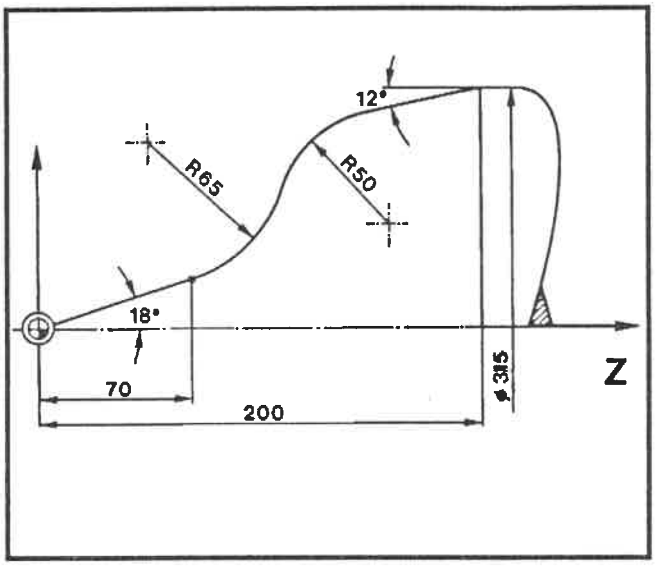

```
N 1 G0  X50  Z2
N 2 G1  X50  Z0
N 3 G1  Z-12
N 4 G3  X?  Z?  R11
N 5 G2  X22  Z?  R15
N 6 G2  X?  Z?  I15  K0
N 7 G13  X50  Z?  I14  R11
N 8 G1  Z-80  A0
N 9 G0  X100  Z50
N10 M30
```

Explanation:

N 1	Rapid traverse to starting point.
N 2	Straight line.
N 3	Straight line.
N 4	Circular arc with radius R11, X- and Z-values of end point unknown.
N 5	Circular arc with radius R15, X-value of end point known, Z-value unknown.
N 6	Circular arc, center of circle relates to starting point.
N 7	Circular arc with radius R11, Z-value of end point unknown, center of circle I14 (diameter 50 - 11).
N 8	Straight line.
N 9	Rapid traverse away from work-piece.
N10	End of program.

SGP

Example
In this example, the circular arc (R50) must be programmed as a roundance, although its size does not exactly make it a "typical roundance".
The start and end points have tangential transitions, and the intersection points of the contour elements involved are calculable. Apart from this, only R65 and R50 are known.



This example also illustrates the power of simplified geometry programming in the control system.

Programming
N 1 G1 X315 Z200
N 2 G1 A-12 B50
N 3 G2 X? Z70 R65
N 4 G1 X100 Z0 A-18
N 5 G0 X200 M30

Explanation

N 1 Straight line.

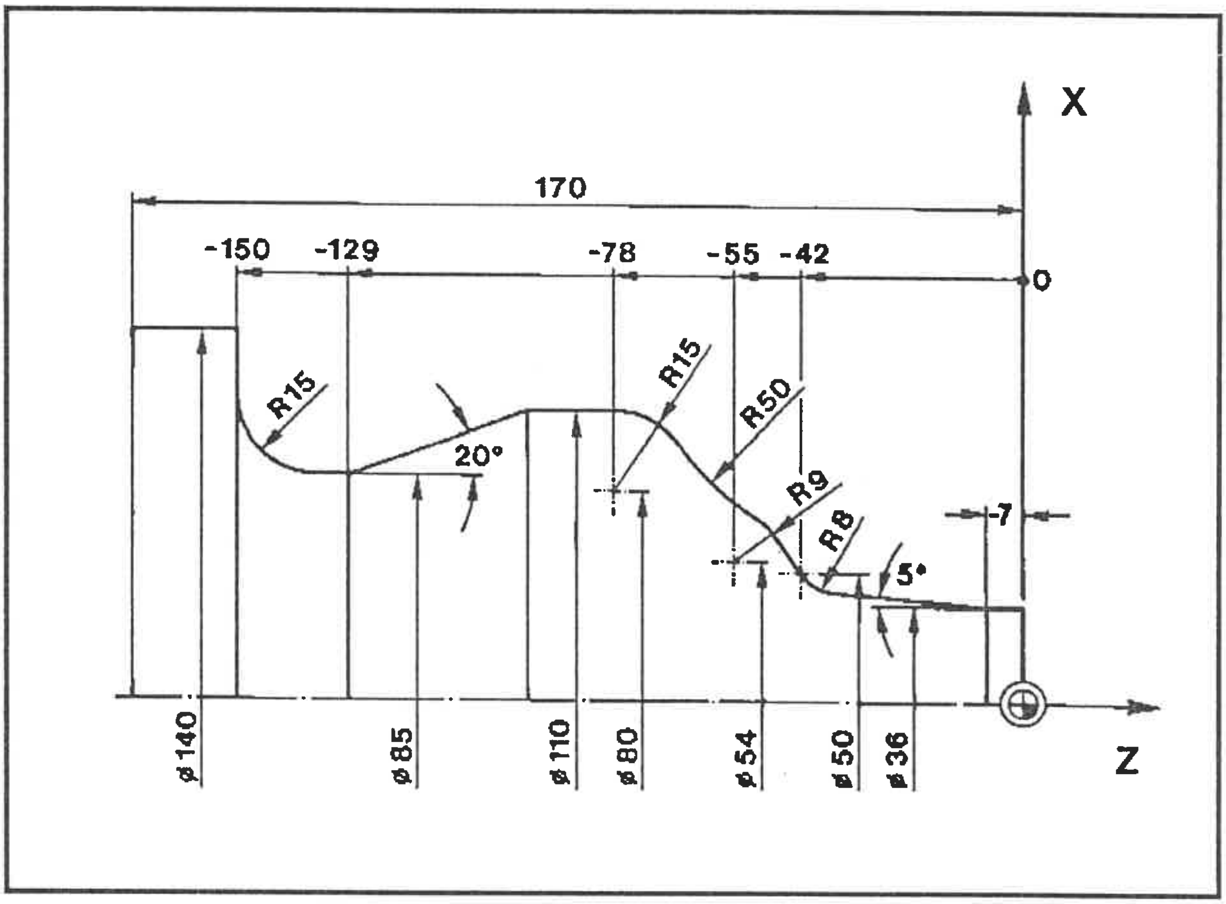
N 2 Straight line with roundance. The straight line is falling (A12). Roundance radius (B50). As neither X nor Z were programmed, the control system assumes that both must be calculated. G1 X? ? A-12 B50 should actually have been programmed.

N 3 Circular arc with radius R65, X-value of end point unknown.

N 4 Falling straight line (A18), end point known.

SGP

Example of circle/circle
In this example, R50 **must** be programmed as a **circle,** as the radii and centers of the two adjoining circular arcs are known.



Programming of the contour:

```
N 1 G0  X36  Z0
N 2 G1  Z-7
N 3 G1  X?  Z?  A5
N 4 G2  X50  Z-42  R8
N 5 G1  X?  Z?
N 6 G13  X?  Z?  I27  K-55  R9
N 7 G2  X?  Z?  R50
N 8 G13  X110  Z-78  I40  K-78  R15
N 9 G1  Z?  A0
N10 G1  X85  Z-129  A-20
N11 G1  Z-150  B15
N12 G1  X140
N13 G1  Z-170
N14 M30
```

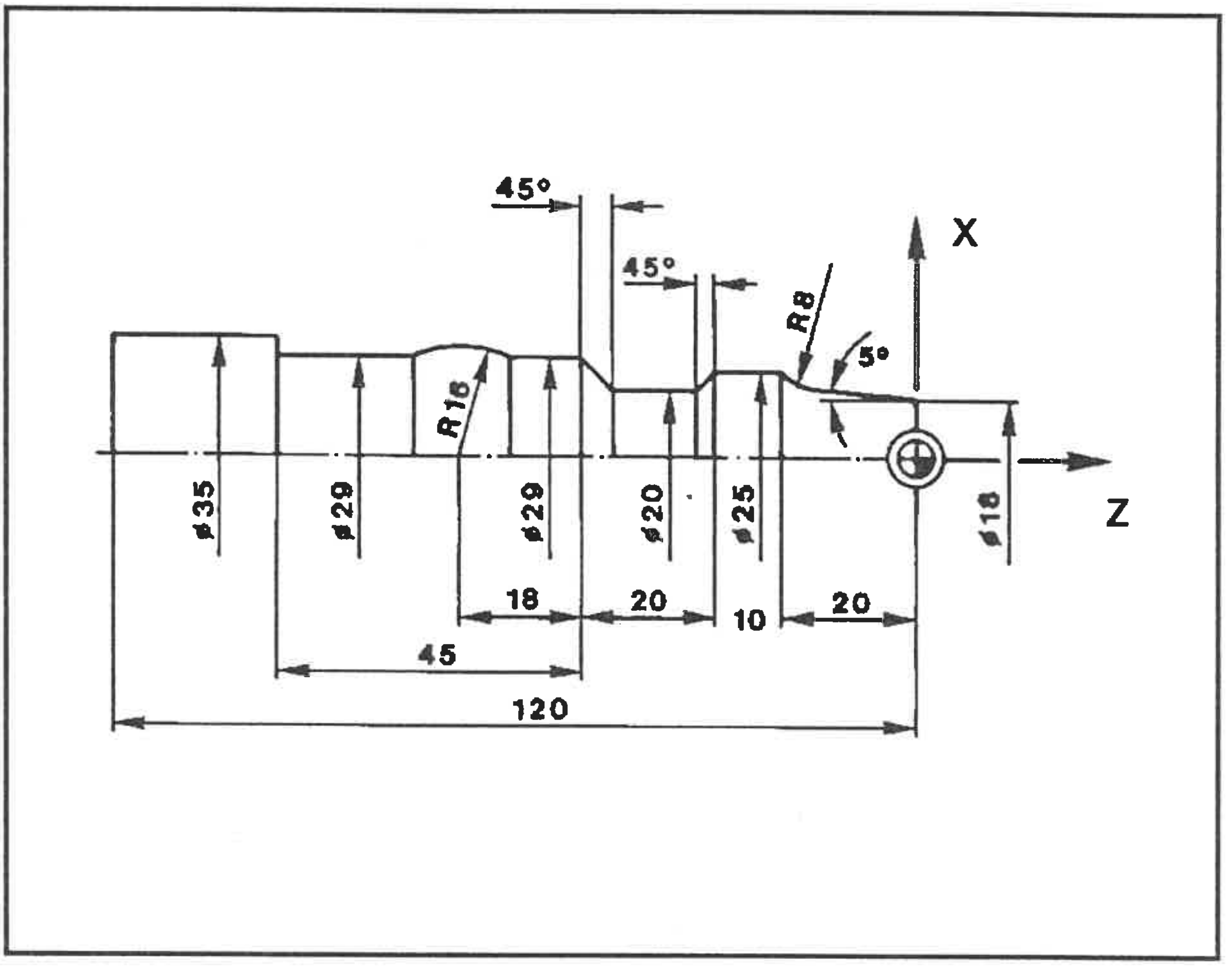
SGP

Explanation:

N 1	Rapid traverse to starting point.
N 2	Straight line.
N 3	Rising straight line (A5) X- and Z-values of end point unknown.
N 4	Circle with radius R8.
N 5	Straight line, X- and Z-values of end point unknown.
N 6	Circle, X- and Z-values of end point unknown, radius R9, center point coordinates known (I27 K-55).
N 7	Circle, X- and Z-values of end point unknown, radius R50.
N 8	Circle, X- and Z-values of end point known, radius R15, center point coordinates known (I40 K-78).
N 9	Straight line.
N10	Falling straight line (A-20).
N11	Straight line with roundance, roundance radius (B15).
N12	Straight line.
N13	Straight line.
N14	End of program.

SGP

Example:
Turning a shaft.



Programming:

```
N 0 G95  F0.3  S100  T1  M4  G96
N 1 G0  X18  Z5
N 2 G1  Z0
N 3 G1  A5
N 4 G2  X25  Z-20  R8  B0
N 5 G1  Z-30
N 6 G1  X20  Z?  A-45
N 7 G1  Z?  A0
N 8 G1  X29  Z-50  A45
N 9 G1  Z?  A0  B0
N10 G13  Z?  X29 I0  K-68  R16  B0
N11 G1  Z-95  M30
```

SGP

Explanation:

N 0 Start conditions.

N 1 Rapid traverse to workpiece.

N 2 Straight line.

N 3 Rising straight line (A5).

N 4 Circle with non-tangential transition to next contour (B0).

N 5 Straight line.

N 6 Falling straight line (A-45).

N 7 Straight line parallel to turning center.

N 8 Rising straight line (A45).

N 9 Straight line parallel to turning center, non-tangential transition to next contour element.

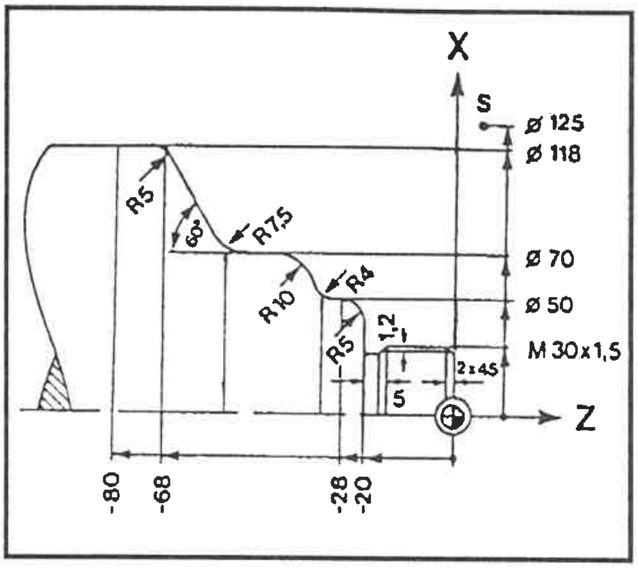
N10 Circle with radius R16, center point coordinates, non-tangential transition to next contour element.

N11 Straight line, end of program.

SGP

Example
Turned part with thread.

Start conditions, stock allowance, tool change and cutting conditions are established in the main program.
The contour is programmed in the subprogram.



Programming:

Main program % 4711

```
N 0 G90  G95  G96  S180  F0.45  T1  M4  M7
N 1 G0  X125  Z3
N 2 G818  X0  Z3  I5
N 3 L777/1
N 4 G80
N 5 G0  X250  Z100
N 6 G96  G0  Z3  S220  T2  F0.15
N 7 L777/2
N 8 G0  X250  Z100
N 9 G97  G0  X35  Z5  S550  T3
N10 G31  X30  Z-19  I0.24  K0  P0.92  F1.5
N11 G0  X250  Z100  M30
```

Subprogram L %777

```
N 0  G42  G1  X0  Z0
N 1  G88  X30  I2
/1N 2  G85  Z-20  I1.2  K5
/2N 3  G1  Z-20
N 4  G87  X50  I5
N 5  G1  Z-28
N 6  G2  X?  Z?  I4  K0
N 7  G13  X70  Z?  I25  R10
N 8  G1  Z?  A0  B7.5
N 9  G1  X118  Z-68  A60  B5
N10  G1  Z-80  G40
N11  M30
```

SGP

Main program

N 0	Start conditions.
N 1	Rapid traverse to starting point.
N 2	Cutting cycle with the addresses of the starting point of the contour and the cutting depth.
N 3	Call subprogram (with deletion level 1 active).
N 4	End of cycle.
N 5	Rapid traverse to tool change point.
N 6	Start conditions for finishing.
N 7	Call subprogram (with deletion level 2 active).
N 8	Rapid traverse to tool change point.
N 9	Start conditions for threading.
N10	Threading.
N11	Rapid traverse to tool change point, end of program.

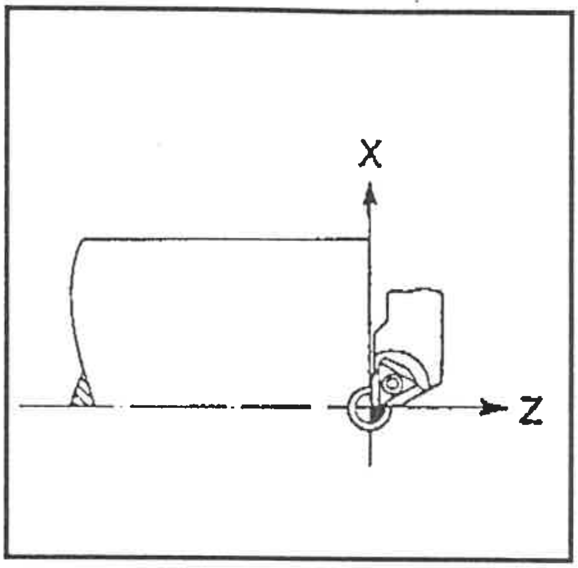
Subroutine

N 0	Starting point, switch on SRK.
N 1	Straight line with subsequent chamfer (programmed with chamfer cycle).
N 2	Undercut cycle, can be deleted with deletion level 1 active.
N 3	Path for roughing tool deleted with deletion level 2.
N 4	Radius cycle.
N 5	Straight line.
N 6	Circle.
N 7	Circle.
N 8	Straight line with roundance.
N 9	Straight line with roundance.
N10	Straight line, switch off SRK.
N11	End of program.

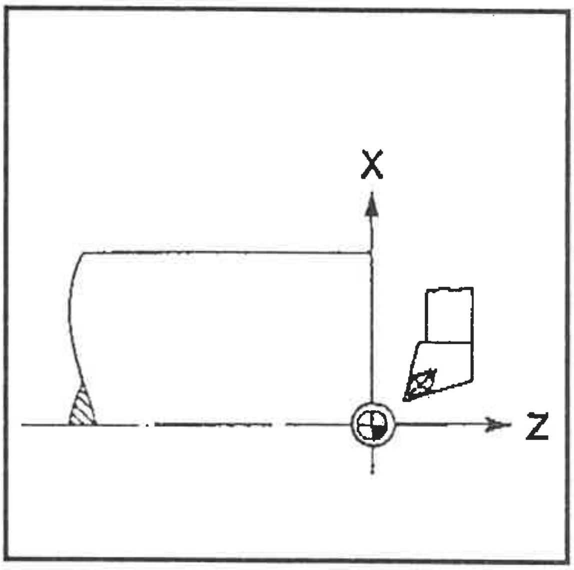
3.4 Tool data/Tool change

T

General
In order to be able to execute a program with different tools, the programmed workpiece contour may not be dependent upon tool geometry. If, for example, the workpiece zero point is established with the finishing tool, the scratch allocates the value zero to the momentary position of the machine slide.



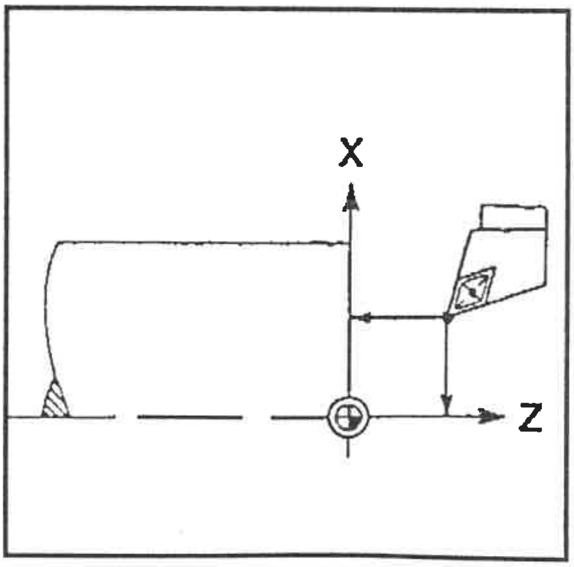
The roughing tool, for example, will not scratch at the same position.
If the roughing tool is to be positioned at workpiece zero point, the slides must be in a different position!
This slide position correction must be made for all tools. To do so, the control system is equipped with a tool compensation facility (standard).



Tool compensation value
The length difference in Z-direction as well as the offset in X-direction are called compensation values. They can be evaluated automatically by the control system.

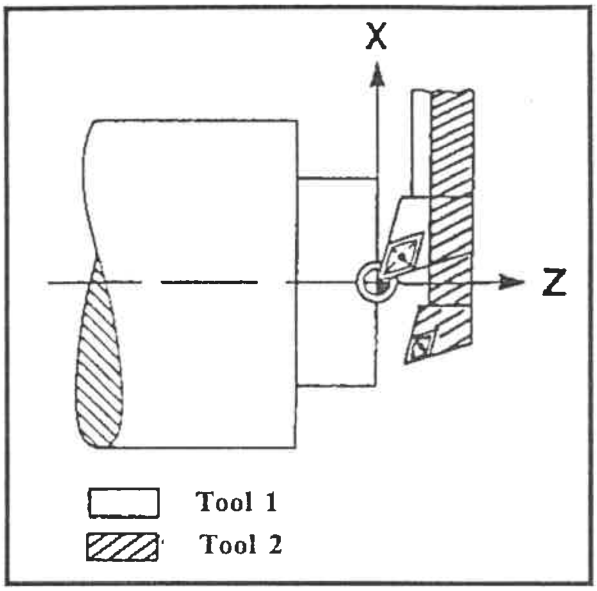
Maximum values for X and Z ± 9999.999 mm.

The sign determines the direction of compensation.



T

Example
Compensation in X
Tool 2 must be compensated **in the direction** of the X-axis, i.e. +.
Compensation in Z:
Tool 2 must be compensated **against the direction** of the Z-axis, i.e. -.



Tool number (T)
The dimensions of max. 64 tools can be stored in the tool file (parameter N1001 to N1064).
The tool number is entered under address T.
The tools are called-up in the program under this number.
The values stored under the corresponding parameters (N1001 to N1064) then automatically take effect.

Programming of tool
The tool number under address T may be programmed as a 2-, 4- or as a 6-digit number.

2-digit programming of T
The simplest way to program a tool number is the 2-digit programming. Here, the control system assumes that the position number on the turret disc and the tool file number (N1001 to N1012) are identical; that means the tool number programmed under T signifies the tool position on the turret disc and the tool data are calculated of from the tool file with the corresponding file number.

Example:
T11 means position number on the turret disc and also tool file number 11 that tool data from the parameter N1011 are calculated.

T

Note

1) The turning machine may be equipped with an eightfold, a tenfold or a twelvefold turret.

   An error message is displayed if a turret position number greater than the maximum turret position number is selected.

2) Apart from this way of programming the tool number the user can program the tool number offset with the help of parameter N12 (for more details also see there).

4-digit programming of T
If more than 12 tools are used, tool numbers run from 01 to 64, turret position numbers from 01 to 12.
The first two digits of the 4-digit entry indicate the tool number, the last two the turret position number.

Example:
T3504 means position number 4 on the turret disc and simultaneously tool file number 35 so that the tool data from parameter N1035 are calculated.
Tool compensation D35 will be activated if no other D is programmed.

6-digit programming of T
Additionally the user is allowed to program the tool number in the parts program via a 6-digit reference number.
This number is attached to a tool file or the turret position via parameter.
This number, being in no direct relation with the tool data, can be chosen freely; e.g. allocated to company specific identification numbers of the user for certain tool types.
The control system only accepts the 6-digit programming of tool numbers if the parameter N1600 is switched to 6-digit tool numbers.
If now in parts programs a 6-digit T-number appears, the control system chooses from the tool reference list (N1601 to N1664) the parameter under which the identification number **ID** is to be found.
The parameter address T programmed under this parameter is calculated as tool position on the turret disc.

T

The tool measures are taken from the corresponding tool files **N1001** to **N1064**.
The last two digits of the parameter numbers **N1601** to **N1664** are identical with the last two digits of the parameter numbers **N1001** to **N1064**.

Example
The following example shows a possible section of the tool reference list.

...
...
N1625 25 ID 143567 T 7
N1626 26 ID 123534 T 8 N1026 T26 WT... FC... X... Z... I... K...
...
...

If, e.g., in a parts program **T123534** is programmed, the tool placed on turret position **8** is positioned into processing position.
The tool data are calculated from the tool file **N1026**.

Tool data input
There are 4 possible ways:
- manually using parameters,
- using parameter punched tape (option),
- using NC program with G92 (X, Z, I, K only),
- under setup operation (**MANUAL CONTROL operating mode**, see section 2).

The tool description contains the following data:

WT: The tool type required for the desired machining procedure is programmed under this address.

FC: The color code for graphic simulation (optional) of the tool in question is stored under this address.

X: Compensation value of tool in X

T

Z: Compensation value of tool in Z

I: Position of cutting edge center point in X

K: Position of cutting edge center point in Z

A: Angle, depending on tool type

B: Angle, depending on tool type

C: Number of teeth on the milling tool

D: Diameter (e.g. useable bore diameter of the tool or diameter of the milling tool)

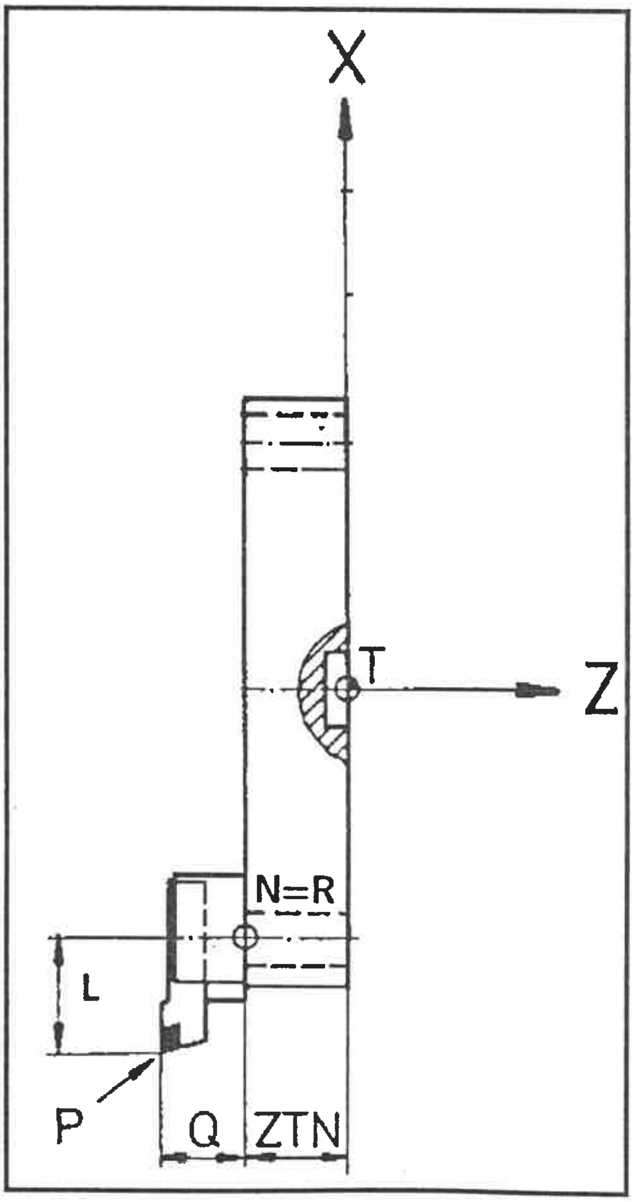
L: Length (e.g. useable boring rod length or length of the milling tool)

Determination of the tool compensation values X and Z
The reference point in located in the pick-up point of the tool in the revolver disc, the "tool reference point N", so that it will only be necessary to enter tool-specific data under the addresses X and Z. The dimension to be entered under addresses X and Z therefore depends on the distance from the tool reference point N (reference dimension **R**) to the tool nose **P** and can be obtained from the tool file sheet.

Note: In the case of revolver discs with an inside and outside circle, the difference between these two circles has to be compensated in the position correction (parameters N401 to N416).

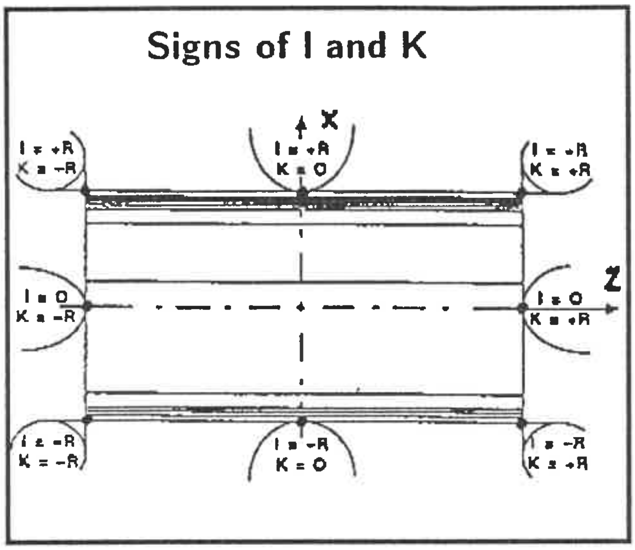
If the reference point in the crosswise direction is located on the inner circle of the revolver disc, the difference in the crosswise direction in the case of parameters N401 to N416 has to be entered with a **positive** prefix under the address X for the tool positions of the outer circle.

If the reference point in the crosswise direction is on the outer circle of the revolver disc, the difference in the crosswise direction in the case of parameters N401 to N416 has to be entered with a **negative** prefix under the address X for the tool positions of the inner circle.

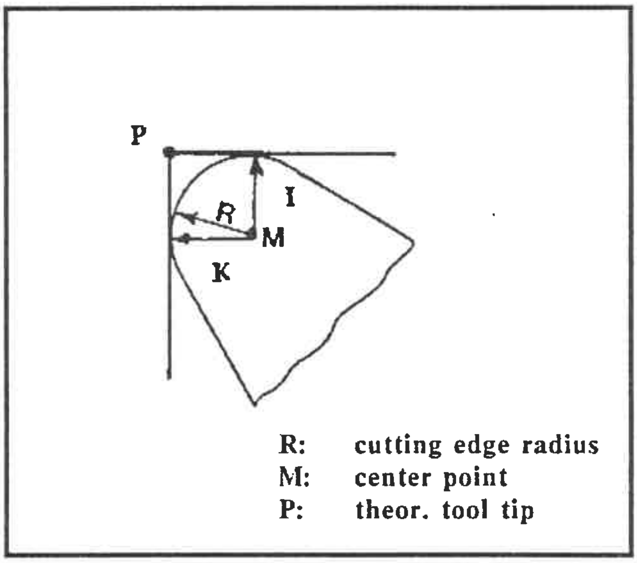


T

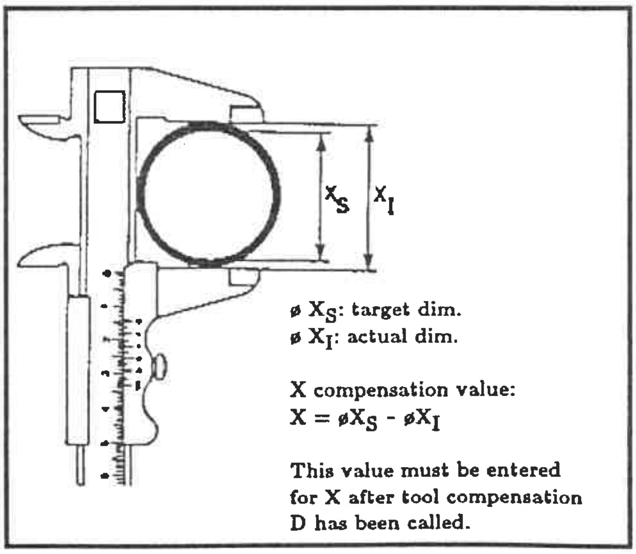
Determining tool position from addresses I and K
By entering values for I and K, the control system is informed about the the tool position. Eight different tool positions are permitted which result, for example, from the directions of scratching (see illustration on the right).



At the same time, the absolute values of I and K also inform the control system about the cutting radius R.
Exception: tool for grooving
Moreover, I and K determine the position of the cutting edge center M relative to the theoretical tool tip **P** (see illustration on the right).
Tool offsets (D)



In addition to the compensation values in tool definition, 80 free offset pairs are available. They are used, e.g., to account for stock allowances or wear compensation of tools which have already been defined.
The values can be altered during program execution.



D

Compensation store
80 different compensation pairs can be entered in the compensation store under parameter addresses N1101 - N1180.

Activating tool offset, D
The compensation pairs are called in the program under address D with following number 1 to 80. The compensation values stored under the number do not take effect until the next axis movement.

If D0 is programmed, no compensation is effective.

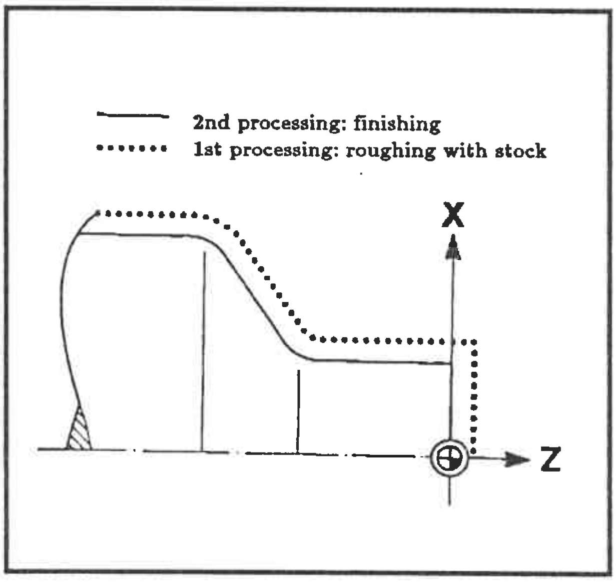
If no D is programmed, the wear compensation will take effect which has the same number as the current tool, i.e., for example, D10 for T10 or D35 for T3510.

3. Programming
3.5 Subprogram Techniques

L

Application
If part of a program is to be used several times within one program or in several different programs, that part can be stored in the form of a subprogram. A subprogram can be called as often as desired within a main program. Protected customer-specific variable programs (begin with the letter "E") can only be called as a subprogram in the parts program.

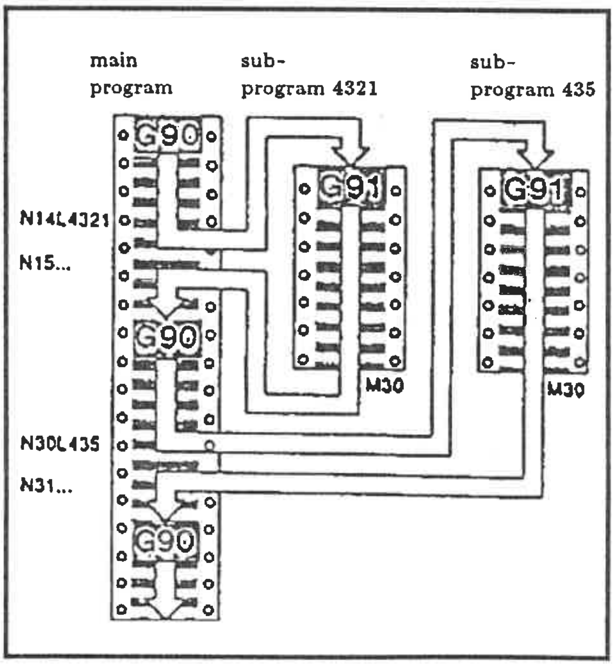
Example
A contour is to be roughed and finished. The contour need only be programmed once. The contour description is stored as a subprogram.
The main program contains feed, speed, tool activation and wear compensation (stock). The subprogram is called at the point in the program at which the contour is to be machined.
The control system executes the subprogram from the first to the last block, then continues in the main program with the block following the subprogram call.



Programming
The subprogram is called from address L. The call must be in a block of its own in the main program.

Example: N14 L 4321

Subprogram number 4321 is called in block 14 of the main program. Apart from address L, this block may contain only information on the deletion levels and the number of repetitions.



Note

In AUTOMATIC and SINGLE BLOCK operating modes, the program cannot start with a subprogram call.

L

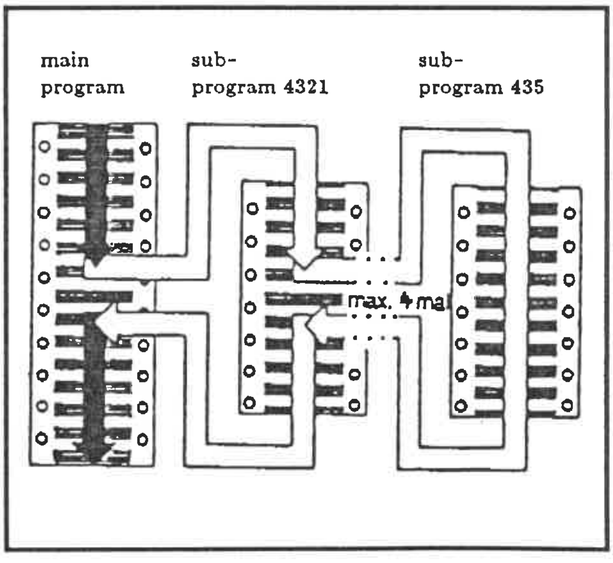
Modal functions
The subprogram only conditionally adopts all modal funcitons from the main program. Abs/Inc function and movement functions are saved via subprograms. For example, the main program contains feed and speed. The subprogram will work using the same data. If a new feed value is programmed in the subprogram, it will apply subsequently in the main program. If you do not want it to (e.g. incutting cycles), the feed which is to be valid in the main program must be programmed in the block with G80 (not necessary for G83).

Repeats of subprograms
It is also possible to repeat a subprogram once or several times.
To do this, the number of repeats is entered under address Q after entering L...

Example:
N10 L888 Q3
N20...
When block N10 is reached, subprogram L888 is repeated 3 times before N20 is executed.

The application is also interesting in conjunction with incremental zero point shift (G56), e.g. for repeating a machining procedure at various points on the workpiece.

Nesting subprograms
Subprograms can be nested to a depth of 4, i.e. a main program calls a subprogram which in turn calls another subprogram etc.
In the program structure, the subprogram blocks are simply inserted in the main program.

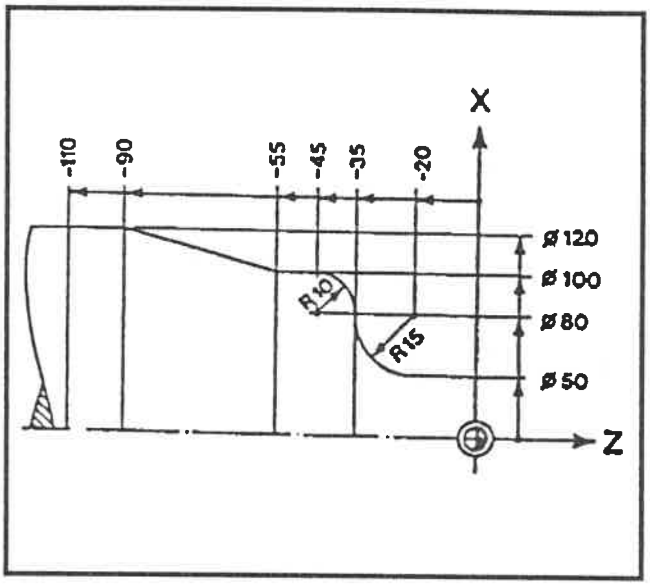


L

Example

Main program

```
N 0  G90  G95  G96  T1  M4  M7  S180  F0.5
N 1  G0  X55 Z0
N 2  L888
N 3  G0  X200  Z100
N 4  G96  S220  F0.15  T2
N 5  G0  X55  Z0
N 6  L888
N 7  G0  X2000  Z100  M30
```



Subprogram

```
L888
N 0  G1  G90  G95  X0
N 1  G0  X50  Z2  G42
N 2  G1  Z-20
N 3  G2  X80  Z-35  R15
N 4  G3  X100  Z-45  R10
N 5  G1  Z-55
N 6  X120  Z-90
N 7  Z-110  G40
N 8  M30
```

Note
Stock allowances in the offset store D1 and D2 are automatically allocated to T1 and T2 respectively:

D1:	X1,0	Z0,2
D2:	X0	Z0

Explanation

N 0	Start conditions.
N 1	Traverse to start point.
N 2	Call subprogram. The contour is machined with stock, as a stock was programmed in the offset store D1 belonging to T1.
N 3	Traverse to tool change point.
N 4	Call tool 2 (no stock or tool compensation).
N 5	Traverse once more to start point.
N 6	Call subprogram. The contour is finished using tool 2.
N 7	Rapid traverse to tool change point, end of program.

L

Input of subprogram, address L

Enter the block number in EDITOR operating mode, block level.

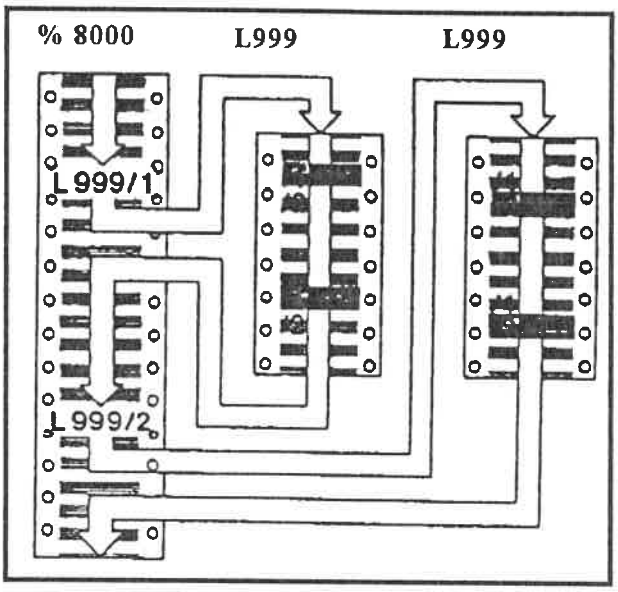
Press the L SUB-PROGRAM softkey.	INPUT SUBPROGRAM NUMBER:
Enter digits.	
Confirm.	NUMBER OF REPEATS Q:
Enter digits.	
Confirm.	DELETION LEVELS:
Enter digits or letters.	
Confirm.	

If no repeats or deletion levels are to be defined, skip the address parameter by pressing the confirm key.
If the block containing the subprogram is to include a **deletion level** or a **deletion step**, proceed as follows:

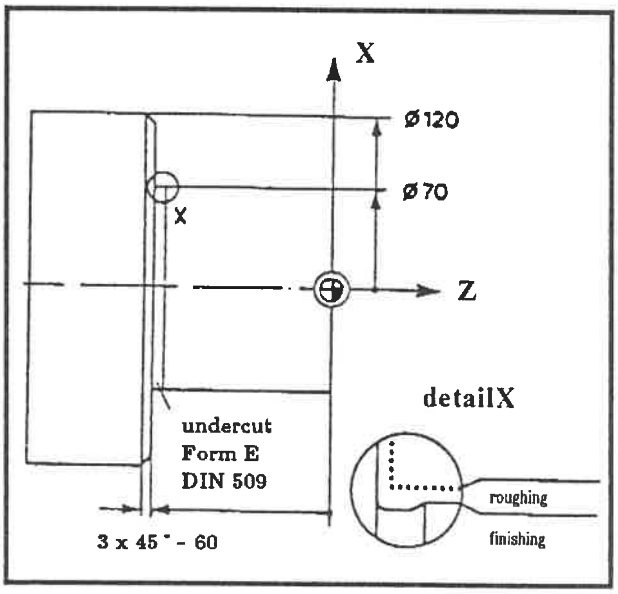
Press the /DELETION LEVEL softkey.	DELETION LEVEL INPUT:
Enter digits.	
Confirm.	

L

Deletion levels
If certain blocks are to be omitted, i.e. not executed, in a subprogram call or within a main program, they can be "deleted". In the illustration on the right the deleted blocks are marked grey.



Example
A subprogram contains the description of a contour - with roughing movements, chamfers and undercuts.
The main program contains feed and speed values, the tool call and the subprogram call.



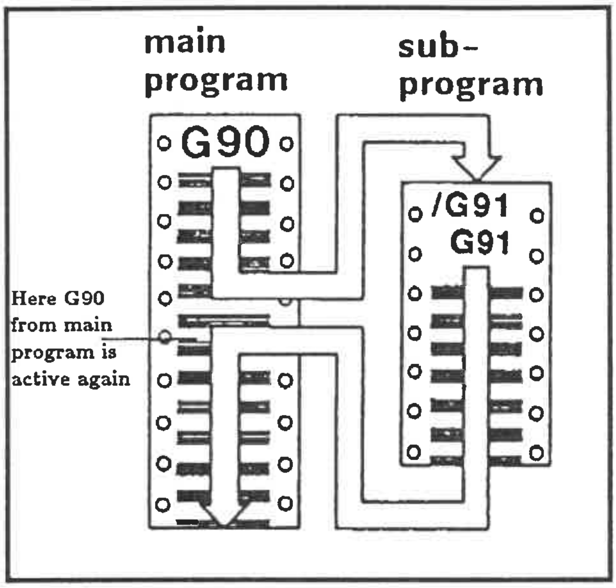
Program prodedure

- Start conditions. Roughing tool.
- Traverse to start point.
- Subprogram with roughing movements, no undercut.
- Finishing tool.
- Subprogram; finish-cutting with undercut.
- End of program.

The first subprogram call omits the undercut, the second subprogram omits the roughing movements. This is achieved by deleting the corresponding blocks when the subprogram is called. Axis-parallel connections between cycles are generated automatically by the control system.

L

Important note
If a block containing modal functions (e.g. G90, G91, G96, G97) subsequently receives a deletion level, the **modal functions** must be **repeated** in the next block which is not deleted.
If you forget to do this, incremental data may be interpreted as absolute data or vice versa.
This can result in nasty **collisions.**
If subprograms contain branching, then programmed deletion levels, absolute and incremental data (G90/G91), the modal path functions and the subprogram repeat factor Q will be saved.
Upon the return from the branch, the original values will apply once more.



Application of the deletion level
The main case of application, as described in the example, is in deleting finishing or roughing movements.

Labelling deletion blocks
Blocks which are to be deleted, i.e. omitted, are indicated by digits 0 to 9. The digit indicates the "deletion level".

E.g.:

/1 N2 G85 Z...
Block 2 with deletion level 1

or

/2 N3 G1 X... Z...
Block 3 with deletion level 2

L

Deletion step
If deletion levels are not always to be deleted, it is possible to enter a deletion step for exactly **one** deletion level.
The deletion step determines how often the block is to run before then being deleted or executed.
For programming the deletion step, see section 6 "parameter description" (N10 and N11). Parameter N11 has the following meaning:

Parameter N11 = 2
The block is executed every **second** program run.

Parameter N11 = 10
The block is executed every **tenth** program run.

It is also possible to program parameter N11 with zero or one.
If the deletion levels are to contain no step, parameters N10 and N11 are of no significance.

Activating the deletion level
The deletion step only takes effect when the corresponding deletion level has been activated in AUTOMATIC mode (see section 5).
If this is **not** the case, the block will always be omitted, i.e. deleted.
The deletion step in this case is of **no** significance to program execution.

L

Example:
Parameter N10 contains a "1",
that is deletion level 1.
Parameter N11 (deletion step) is
set to 4.

% 11111
N 1 M4 G97 S100 G1 X100 Z100 F0.5
N 2 T1 G1 X95 Z0
/1 N3 M5
/1 N4 G4 F5
/1 N5 M4
N 6 G1 X96
N 7 G1 Z100
...
...
...
N... M99

Select AUTOMATIC mode, select program
11111 , starting block N1

Press the SELECT.
DELETION LEVEL softkey.

Enter digits (1).

The dialog line shows:
PROGRAM % 111111 STARTBLOCK N1
DELETION LEVELS: /1

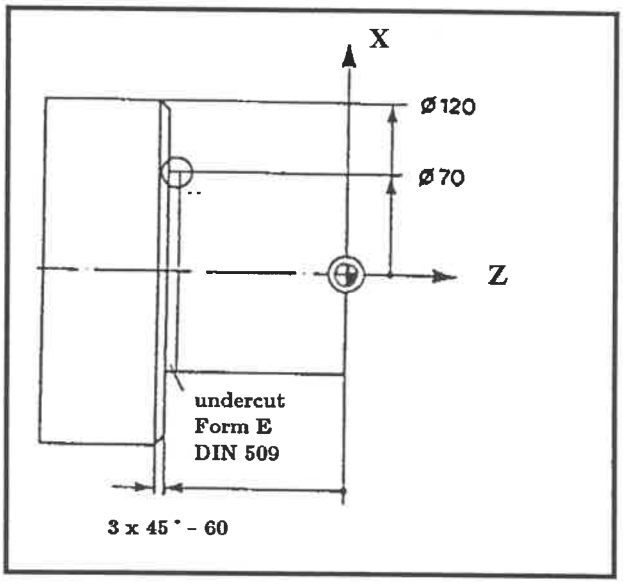
Confirm.

Explanation:
When the program is started and deletion level 1 is active, the spindle will stop in every 4th program run.
5s dwell time expires before the spindle starts once more.
If deletion level 1 has not been selected, the spindle will be stopped in every program run. The same result would be achieved by setting parameter N11 to 1.

The contents of blocks N3...N5 was taken as an example, but any blocks may be deleted.

L

Example
Cycle G817 cuts along the contour described in the following, i.e. roughing without the deleted contour parts (deletion level 1).



Finishing is done after the tool change. i.e. the previously omitted blocks are now executed. As the end or corner points (X, Z) have been entered in the cycles, no straight line need be programmed between the cycle functions. However, if cycles are deleted (/1), the end and corner points (X, Z) will be missing as a result. These are then described by the G1 function of deletion level 2(/2).

Main program %8000

```
N 0  G90  G95  S180  F0.45  T1  M3  M7  M43
N 1  G0  X120  Z3
N 2  G57  X1  Z0.2
N 3  G817  X-1  I5
N 4  L999/1
N 5  G80
N 6  G0  X200  Z100
N 7  G96  G0  X0  Z3  F0.15  S220  T2
N 8  L999/2
N 9  G0  X200  Z100  M30
```

Subprogram L %999

```
N 0  G90  G95  G1  X-1  Z0  F0.15  G42
N 1  G1  X70
/1N 2  G85  Z-60
/2N 3  G1  Z-60
N 4  G88  X120  I3
N 5  G1  Z-65  G40
N 6  M30
```

L

Explanation:

N0/N1	Start conditions with tool 1.
N 2	Stock for the following cycle G817.
N3	Cutting cycle. Longitudinal against contour without "steps".
N4	Call SP 999, all blocks containing deletion level 1 are omitted. N0 Starting position, start concition. N1 Straight line. /1 N2 Cycle undercut is omitted. /2 N3 Roughing movement. N4 Cycle chamfer. N5 Rapid traverse to end position. N6 End of subprogram.
N5	End of cycle.
N6	Traverse to tool change point.
N7	Starting conditions with tool 2.
N8	Call SP 999. All blocks containing deletion level 2 are omitted. N0 Starting position. N1 Straight line. /1 N2 Cycle undercut. /2 N2 Roughing movement is omitted. N4 Cycle chamfer. N5 End position. N6 End of subprogram.
N9	Tool change point, end of program.