SINUMERIK System 800
Cycles, User Memory Submodule 4

User Documentation

SINUMERIK System 800
Cycles, User Memory Submodule 4

Programming Guide

User Documentation

Valid for:

Control	Software version
SINUMERIK 810T/810TE GA1	3 and higher
SINUMERIK 810M/810ME GA1	3 and higher
SINUMERIK 810T/810TE GA2	2 and higher
SINUMERIK 810M/810ME GA2	2 and higher
SINUMERIK 810/820T GA3	1 and higher
SINUMERIK 810/820M GA3	1 and higher
SINUMERIK 820T/820TE	2 and higher
SINUMERIK 820M/820ME	2 and higher
SINUMERIK 850T/850TE	3 and higher
SINUMERIK 850M/850ME	3 and higher
SINUMERIK 880T/880TE	3 and higher
SINUMERIK 880M/880ME	3 and higher
SINUMERIK 880 GA2	1 and higher

January 1994 Edition

SINUMERIK® documentation

Printing history

Brief details of this edition and previous editions are listed below.

The status of each edition is shown by the code in the "Remarks" column.

Status code in "Remarks" *column*:

A . . . New documentation   **B** . . . Unrevised reprint with new Order No.
C . . . Revised edition with new status. If factual changes have been made on the page since the last edition, this is indicated by a new edition coding in the header on that page.

Edition	Order No.	Remarks	UMS
03.89	6ZB5 410-0BQ02-0BA0	A	4
08.89	6ZB5 410-0BQ02-0BA1	C	4
05.90	6ZB5 410-0BQ02-0BA2	C	4
05.91	6ZB5 410-0BQ02-0BA3	C	4
02.93	6ZB5 410-0BQ02-0BA4	C	4
01.94	6ZB5 410-0BQ02-0BA5	**C**	4

Contents

1   Introduction

For standard machining processes which are frequently repeated, machining cycles are available as permanently stored subroutines in the user memory submodule (UMS). Input images of blueprint programmed blocks are also contained in this submodule.

The cycles can be provided with the necessary data by using input screen forms and softkeys or by programming the R parmeters directly into the program.

Machining cycles are called in the part program or subroutine.

In the examples given, the R parameters are assigned values either via the menu display or directly in the part program.
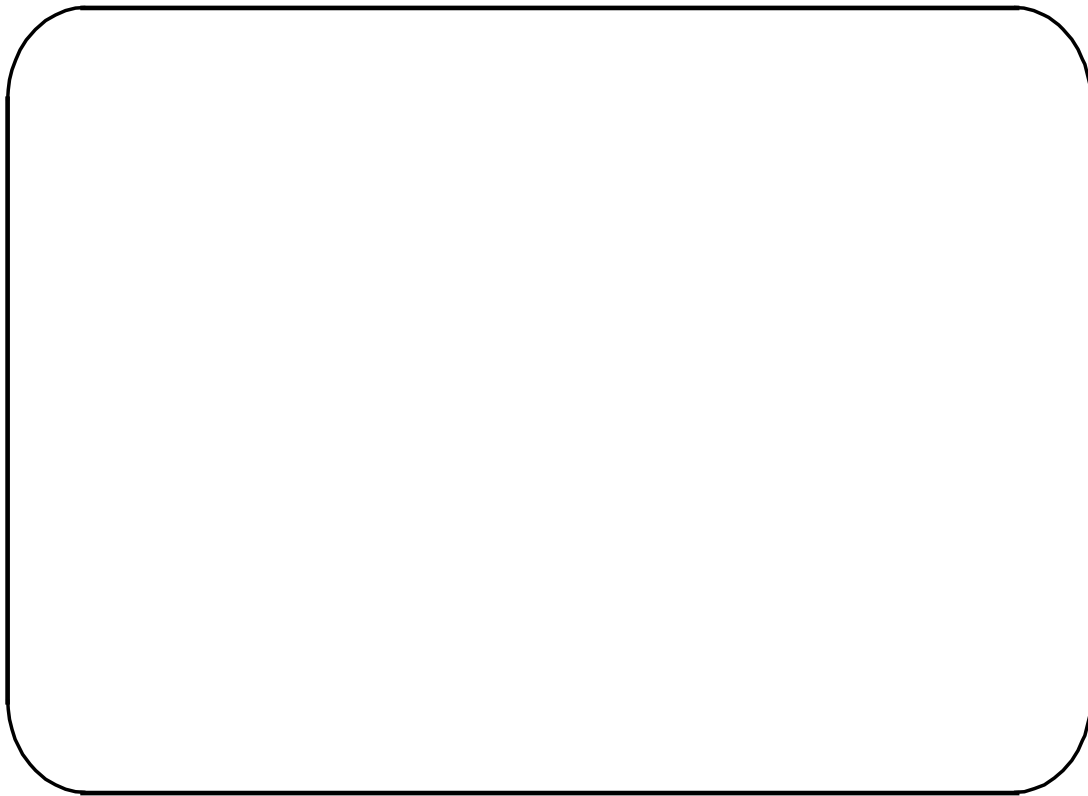
The described cycles can be modified if desired. Ensure compliance with any additional information provided by the machine-tool manufacturer.

All images are enclosed by a frame. This frame contains the menu display and all defining parameters, which are also identified with symbols. The symbols are used in the menu displays as dimensioning mnemonics.

The cycles end uniformly with preparatory functions G00, G60 and G90. Any other G functions required when the program continues must be reprogrammed.

Overview of subroutine numbers:

L No.	Function
L01 . . L05	Free for assignment by user
L06	Siemens
L07 . . L80	Free for assignment by user
L81 . . L99	Siemens
L100 . . L799	Free for assignment by user
L800 . . L999	Siemens

In the Cycles Description the following is presumed:

- The Programming Guide, User's Guide and Operator's Guide have been studied. The Description is valid for all System 800 controls.

- If the R parameters are to be assigned via menu displays, the graphics function must be available.

- In drilling and milling patterns, polar coordinate programming is required.

- The "blueprint programming" option is a prerequisite for L95 Recessing cycle.

- The current plane must be selected before calling the cycle via G16 or G17 to G19. The infeed axis (drilling axis) is always the axis perpendicular to the current axis.

Ordinate
(perpendicular e.g. Y axis)

90°

Applicate
(vertical e.g. Z axis)
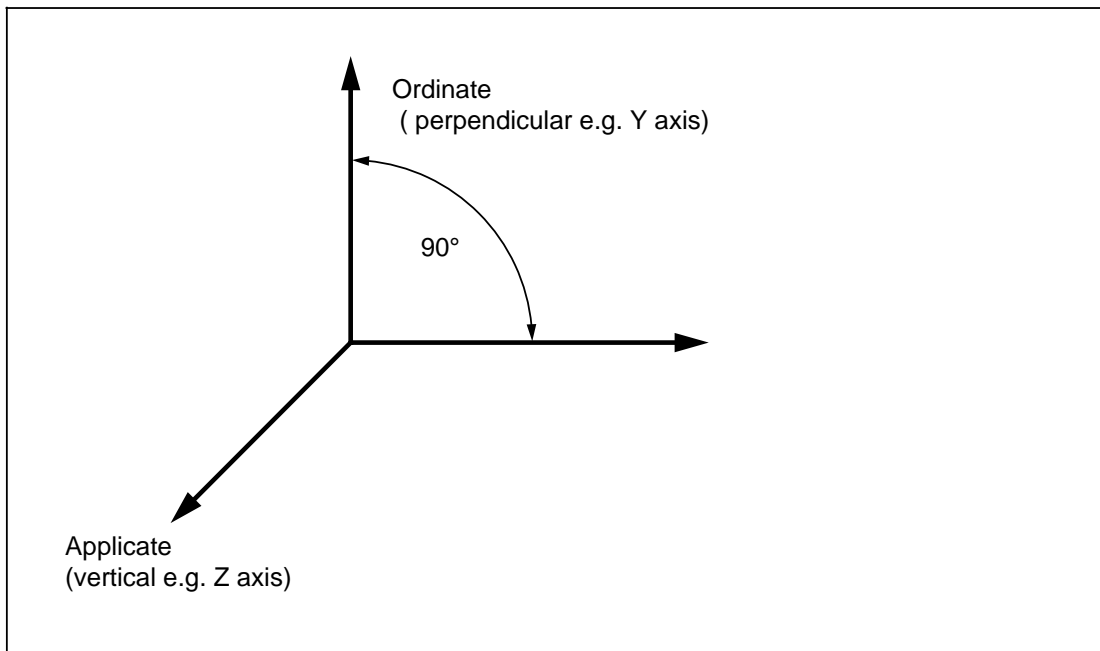
- If the blueprint programmed block is to be supplied with data by means of the input images, the "blueprint programming" option is required.
  If the blueprint programmed blocks are being used on an M control, a plane selection display can be shown by softkey.

  This initiates an axis name adjustment in the coordinate cross according to the selected plane. In addition, the path addresses in the contour definitions are adjusted according to the plane chosen in the selection display.

  Make sure, however, that the selected plane (G16 or G17) is programmed in the part program.

SINUMERIK System 800, Cycles, UMS 4 (PG)

Compatibility

The standard user memory submodule (UMS) has been completely revamped and offers a whole range of new functions and options (UMS 4). The UMS 4 is not compatible with UMS 2. If you wish to make use of the new functions in the UMS 4, part programs created before must be adapted.

Setting data SD 5000 can be used to define whether the expanded functions of the UMS 4 can be used or whether the functions of the user submodule UMS 2 ard to be used (compatible mode).

	T version	M version	
	Turning cycles L95/L93/L98	Drilling/Milling patterns L903/L930	Drilling cycles L81 to L89
SD 5000	Bit 2	Bit 1	Bit 0
	1	1	1
	0	0	0

Bit 1 = The expanded functions of the UMS 4 and UMS 3/60 can be used.
Bit 0 = Functions as UMS 2 (compatible mode)

SD 5000.6= 0 L900 is executed in the normal way (with a safety clearance of 1 mm along the drilling axis)
        = 1 L900 without a safety clearance along the drilling axis

This setting data bit 5000.6 applies to UMS 48 and higher.

SD 5000.7= 0   no calculation of overflow override
        = 1   calculation of overflow override

Machine data (MD) 157

Throughout System 800, machine data MD 157 is used to identify the control type and the software version. This data is used in all control types from SINUMERIK 810 to SINUMERIK 880.

Control type	MD 157
810T	011 xx
810M	012 xx
810G	013 xx
820T	021 xx
820M	022 xx
850T	051 xx
850M	052 xx
880T	081 xx
880M	082 xx
880T GA2	091 xx
880M GA2	092 xx

Example:

SINUMERIK 810M with NC software version 2:

MD 157 = 012 02

SINUMERIK System 800, Cycles, UMS 4 (PG)

1-5

Machine data (MD) 19

For the SINUMERIK 850 the P number of the TO memory in which the address of the following tool stands must be written into machine data MD 19 for tool management.

SINUMERIK 850**/**880 MD 19 = 10   (standard value)
                                                = 5 to 9   (variable P number)

The following alarms are output in the cycles:

Alarms	Description
4100	No D number active (L901-904, L93-L95)
4101	Tool radius = 0 (L903, L930)
4102	Cutter radius too large (L901, L903, L904, L930)
4103	Tool too wide (L93)
4120	No spindle direction programmed (L84)
4121	Spindle not inside tolerance range (L84)
4122 [1]	Calculated feed too great (L841)
4140	Machined part diameter too small (L94)
4153	Thread length too short (L84 only SINUMERIK 850/880)
4180	Option not available
4200	Check definition R (Nxxx)

Example: 4200 1 N 32 Check definition R (Nxxx)
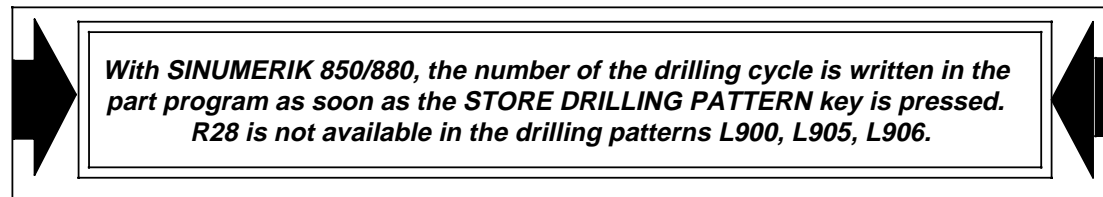
In the cycle being executed in channel 1, the system has determined that parameter R32 has been incorrectly defined.

Apart from L95 and L96 (stock removal cycles), the **scale factor** is taken into account in the cycles. The scale factor is not effective with contour subroutines.

_____

[1]   _This alarm only occurs with SINUMERIK 880M, GA2 ._

SINUMERIK System 800, Cycles, UMS 4 (PG)

If the cycles are supplied with data via menu displays, several storage softkeys are available for selection:

- STORE MENU
  The display branches directly to the selected part program.

- STORE CHOICE
  The display branches to the selection menu (overall selection of drilling and milling patterns). The parameters are already stored in the part program.

- STORE
  The display branches to the selection menu. Storage takes place with blueprint programmed blocks. The selected display is retained.

- STORE DRILLING PATTERN
  The display branches to the drilling patterns (L900, L905, L906); R28 is automatically supplied with the selected drilling cycle (L81 to L89) in the input image.

> **With SINUMERIK 850/880, the number of the drilling cycle is written in the part program as soon as the STORE DRILLING PATTERN key is pressed. R28 is not available in the drilling patterns L900, L905, L906.**

2   Cycles

2.1   Machining cycles for turning
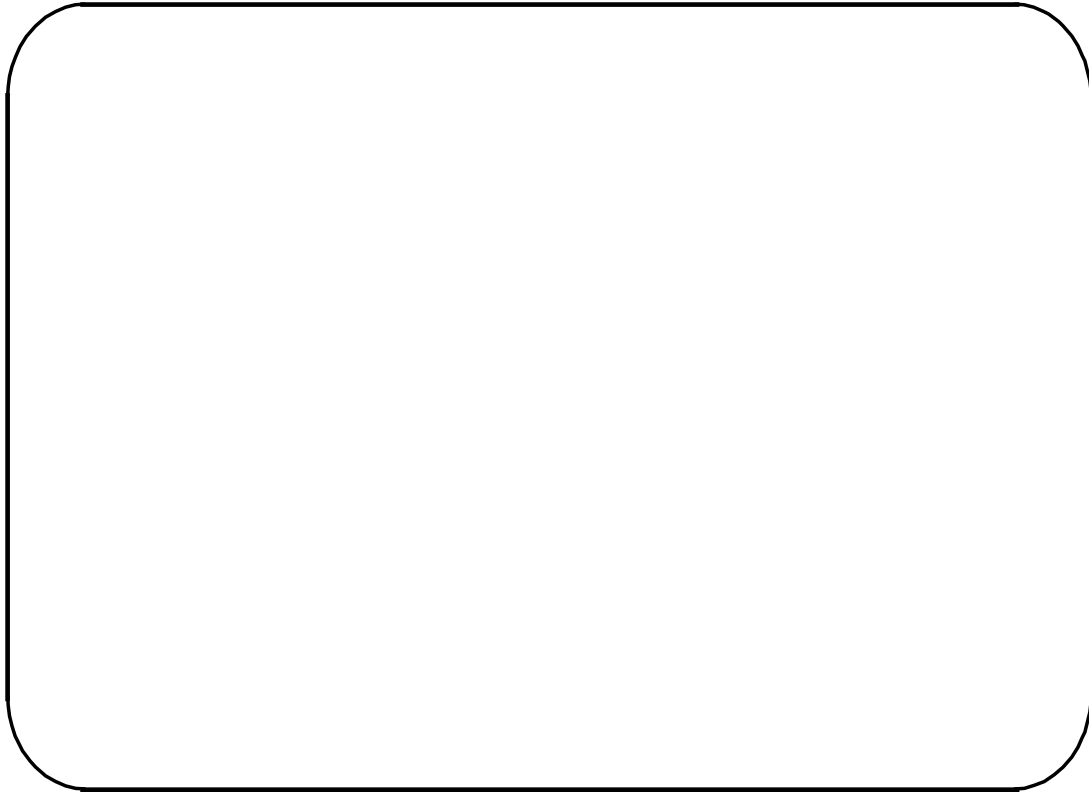
2.1.1 L93 Grooving cycle (prerequisite: blueprint programming)

The grooving cycle L93 allows symmetrical and asymmetrical outside and inside grooves to be made; these may be longitudinal or facing cuts.

In the case of two-edged tools (recessing tools), the tool offset for *one* cutting edge of the recessing tool must be selected and the desired offset value programmed before the grooving cycle is called in a machining program.

The tool offset for the second cutting edge of the recessing tool must then be stored in the tool offset memory under the next higher offset number. If the tool offset for the first cutting edge is D = "n", the tool offset for the second cutting edge will have the offset number D = "n" + 1.

In connection with tool management, the P No. of the TO memory in which the address of the following tool stands must be written in machine data MD 19 for SINUMERIK 850.

When using the tool management system in the 840/880 controls and L93, the cycle machine data (MDZ) 7000 bit 4 must be set to 1.

The following values are entered in the menu display or they are programmed directly in the part program as parameter assignments:

Symbol	Parameter	Description	
	R10	Type of machining: longitudinal R10 = 0; face R10 = 1	
D1/L1	R21	Outside/inside diameter or starting length	(face) (absolute)
Ap	R22	Starting point: longitudinal Z; face X	(absolute)
	R23	Control parameter: starting point left or right	
S1	R24	Finishing cut depth at groove base	(incremental)
S2	R25	Finishing cut depth of flanks	(incremental)
Zt	R26	Infeed depth; enter without ± sign	(incremental)
B	R27	Width of groove	(incremental)
t	R28	Dwell at groove depth	
W1	R29	Angle (0° to 89°)	
R1	R30	Radius or chamfer at groove base	
D2/L2	R31	Groove diameter or length of groove depth	(face) (absolute)
R2	R32	Radius or chamfer at groove edge	
R3	R33	Radius or chamfer at groove edge	
R4	R34	Radius or chamfer at groove edge	
W2	R35	Angle (0° to 89°)	

SINUMERIK System 800, Cycles, UMS 4 (PG)
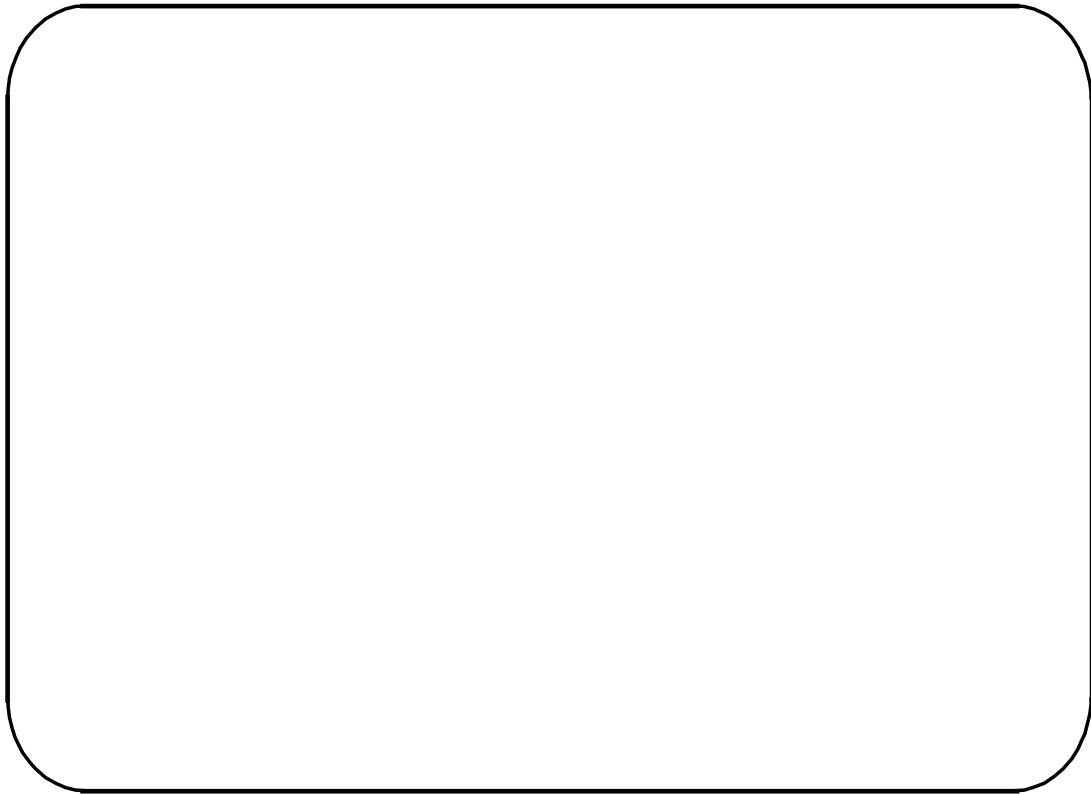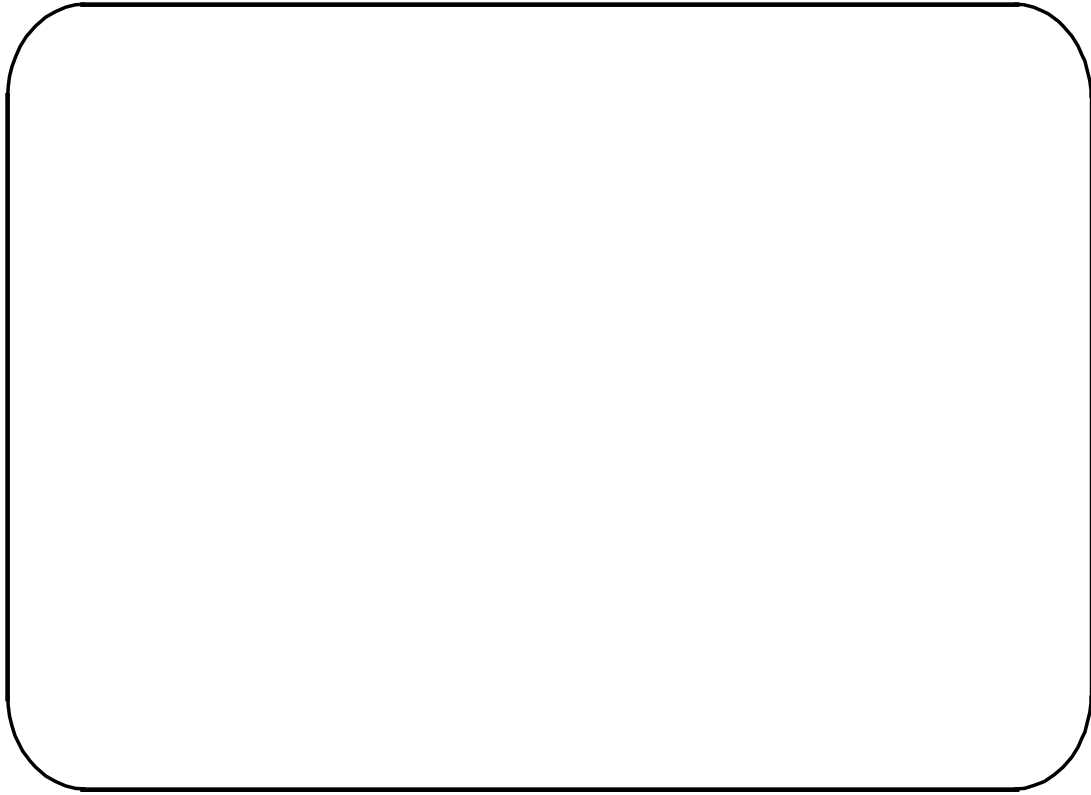
R10: Type of machining

R10 defines the type of groove:

Longitudinal groove: R10 = 0
Facing groove: R10 = 1

D1/L1 R21: External or internal dimension or starting length (face) (absolute)
Ap   R22: Starting point: longitudinal Z; face X (absolute)
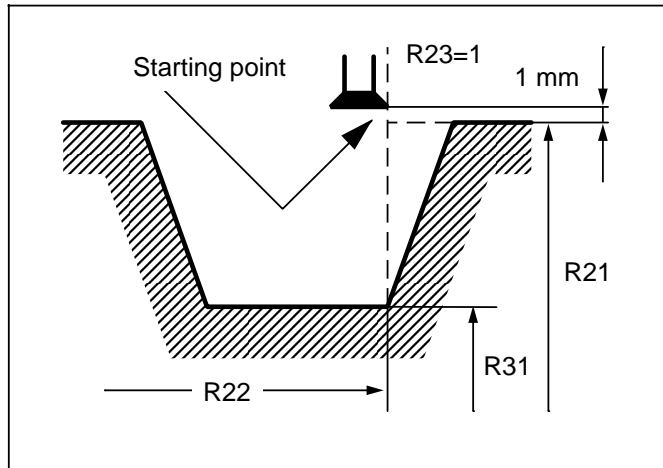
Parameters R21 and R22 define the starting point.

The control automatically approaches the point programmed with R21 and R22. For an external groove traversing is first in the Z direction and for an internal groove in the X direction.

R21 is approached with a safety clearance of 1 mm with both longitudinal and facing cuts.

R23: Control parameter

The control parameter defines the starting point:

Longitudinal groove :	23 =	1	Outside right	Facing groove:	R23 =	1	Right inside
		1	Inside right			1	Left inside
		−1	Outside left			−1	Right outside
		−1	Inside left			−1	Left outside

S1 R24: Final machining allowance at recess groove base (incremental)
S2 R25: Final machining allowance of flanks (incremental)

The final machining allowances R24/R25 can be input with different values.

Stock removal (roughing) is effected down to the finishing allowance. Then the finishing increment is removed parallel to contour with the same tool.

If radii or chamfers are to be inserted at the groove base, a check is made to determine whether these would be damaged when plunge cutting.

If R24/R25 = 0, stock removal parallel to the contour does not occur.

Zt R26: Infeed depth (incremental)

By programming the infeed depth, it is possible to determine whether the groove depth is to be reached in one or more cuts. If several cuts are required, the tool is retracted by 1 mm for chip breaking after each infeed (cf. 2 step).

**B R 27: Width of groove (incremental)**

If the width of groove is wider than the cutting tool, the infeed is divided into equal parts. The maximum infeed depends on the tool width. It is 95 % of the tool width *after* deduction of the cutter radii. This guarantees overlapping cuts.

t R28: Dwell at groove depth

Dwell must be large enough to permit at least one spindle revolution.

W1 R29: Angle
W2 R35: Angle

The flank angle can be between 0° and 89°. For longitudinal grooves, enter the angle from the perpendicular axis, and with face cuts the angle from the horizontal axis.
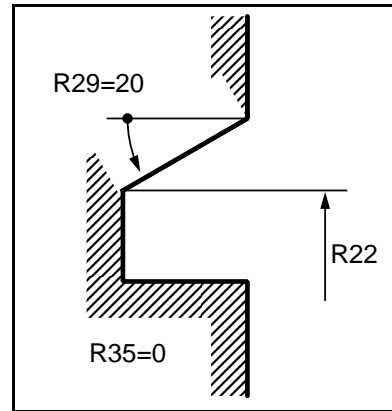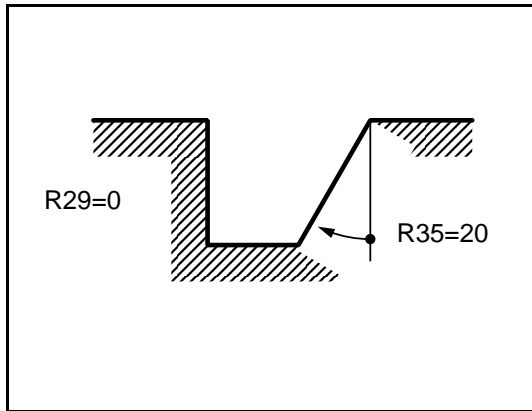
D2/L2 R31:   Groove diameter or length of groove depth (face) (absolute)

R31 determines the groove depth:

R1 R30: Radius or chamfer at groove base
R2 R32: Radius or chamfer at groove edge
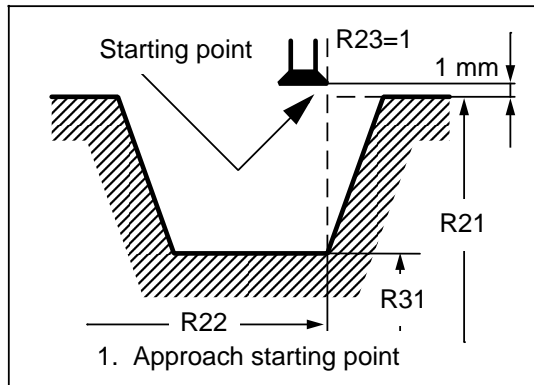**R3 R33: Radius or chamfer at groove base**
R4 R34: Radius or chamfer at groove edge

Radii or chamfers can be inserted at the base and/or edge of the groove by means of parameters R30, R32, R33 and R34.

Signs   + = Radius
               − = Chamfer

Machining sequence

1. Approach starting point

1st step:

Automatic approach to programmed starting point.

2. Radial grooves
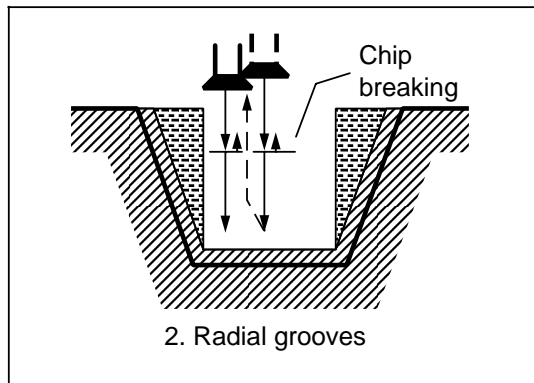
2nd step:

Groove perpendicular to the turning axis in one or more cuts. Before withdrawing from the groove, the tool is retracted by 1 mm in the Z direction from the second step on onwards.

3. Stock removal of flanks
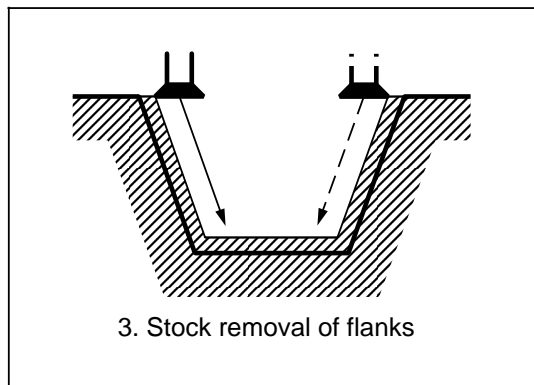
3rd step:

Stock removal of the flanks in one cut, provided that an angle has been programmed with R29 or R35.

Infeed in Z direction is in several steps if the tool width is less than the flank width.

4. Finish cut
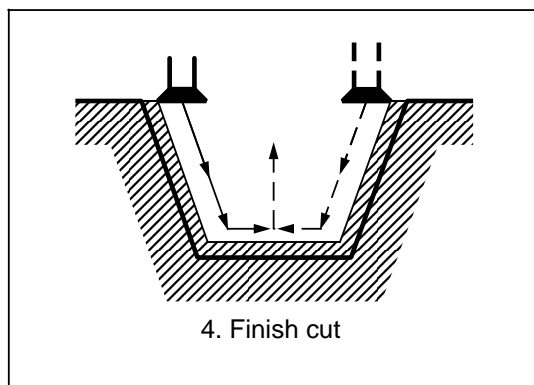
4th step:

Cutting the finishing increment parallel to contour up to the centre of the groove.

Example 1:   "Outside left" longitudinal groove selected by softkey

```
%1
N05  G95  G0  X65  Z105  D03  T03  S500  M04  LF        Select groove position
N10  G01  F0.2  LF
N15  R10=0  R21=60  R22=100  R23=-1  LF
N20  R24=1  R25=1  R26=5  R27=20  LF
N25  R28=0  R29=10  R30=-2  R31=40  LF
N30  R32=2  R33=-2  R34=2  R35=15  LF
N35  L93  P1  LF                                        Call grooving cycle
N40  G0  X100  Z200  LF
N45  M30  LF
```
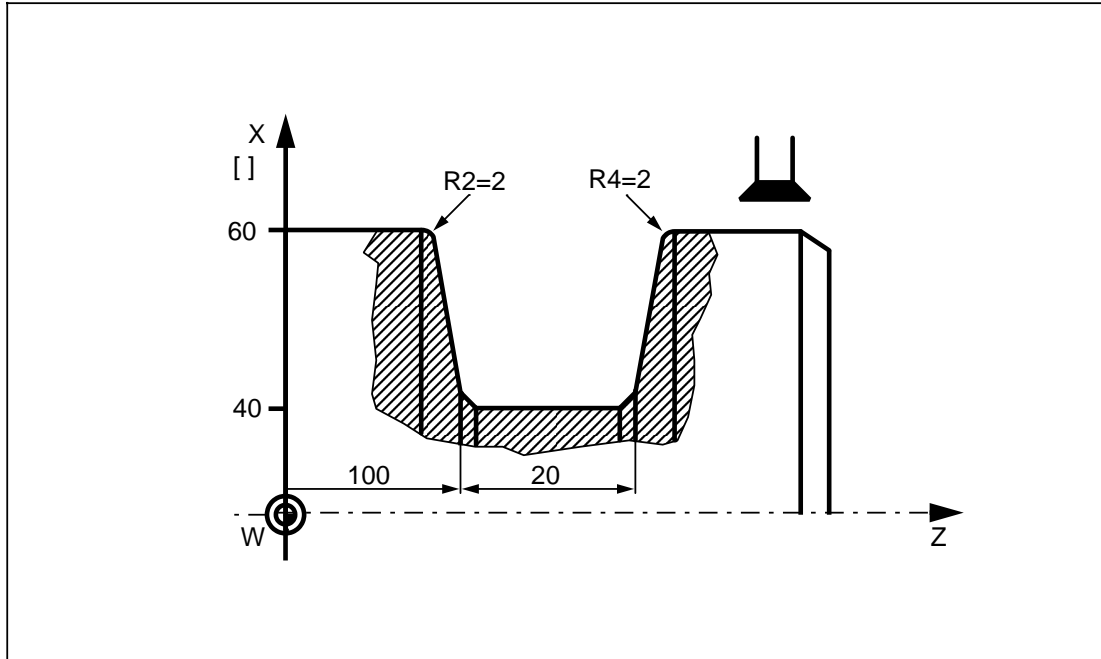
Example 2:   "RIGHT OUTSIDE" facing groove selected by softkey

```
%2
N05  G95  G0  X65  Z10  D03  T03  S500  M04  LF              Select groove position
N10  G01  F0.2  LF
N15  R10=1  R21=0  R22=60  R23=-1  LF
N20  R24=1  R25=1  R26=5  R27=20  LF
R25  R28=0  R29=10  R30=2  R31=-15  LF
N30  R32=-2  R33=2  R34=-2 R35=15  LF
N35  L93  P1  LF                                             Call grooving cycle
N40  G0  X100  Z200  LF
N45  M30  LF
```

SINUMERIK System 800, Cycles, UMS 4 (PG)
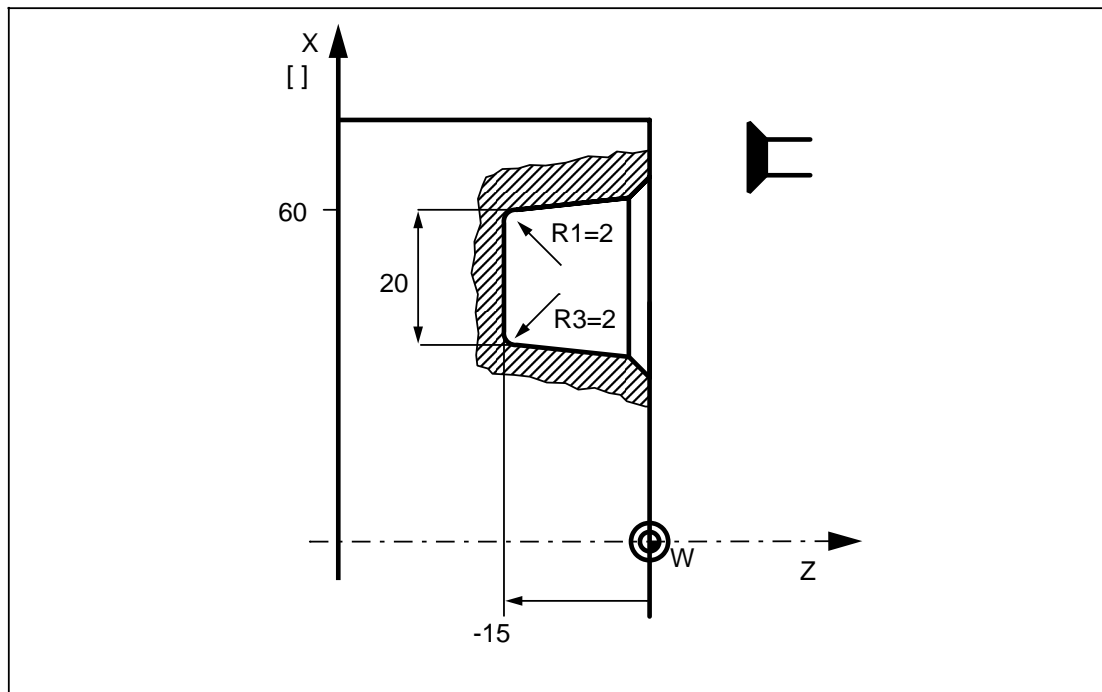
2.1.2   L94 Undercut cycle

L94 cycle permits form E and F undercuts under normal conditions according to DIN 509 with a machined part diameter of > 18 mm. The TNRC is automatically selected in the cycle. A feed value must be programmed before L94 is called.

The following values are input in the menu display or programmed directly as parameter assignments:

Symbol	Parameter	Description
S1	R01	Definition of the tool nose position (1 to 4)
D	R02	Starting point of the contour in X
Ap	R03	Starting point of the contour in Z
	R04	Identifier for form E or F: R04 = 4 Form E for workpieces with one machining surface R04 = 5 Form F for workpieces with two machining surfaces positioned perpendicular to each other

S1 R01: Definition o   the tool nose position (1 to 4)

The tool nose position is defined in R01. Four tool nose positions can be selected: Sl=1, Sl=2, Sl=3, Sl=4.

D R02: Starting point of the contour in X

R02 is supplied with the diameter of the machined part. The cycle automatically adds 2 mm in diameter to this dimension, which represents the starting point in X.

Ap R03: Starting point of the contour in Z

In R03, the machined part dimension in Z is entered. The cycle automatically adds 10 mm to this dimension, which represents the starting point in Z.

2.1.3   L95 Stock removal cycle with relief cut elements
L96 Stock removal cycle without relief cut elements

Stock removal cycles L95/L96 permit paraxial machining of a blank with the contour programmed in a subroutine. The machining cycles can be called from any position where there is no danger of collision. The control calculates the starting point automatically using the final contour description.

- L95
  With L95, relief cut elements (max. 10) are also permitted in the contour. Removal of remaining corners and finishing are always carried out in the same direction as roughing, even if the contour is programmed in the opposite direction.

- L96
  L96 permits quicker starting of the first traversing movement. With L96, removal of remaining corners and finishing are always carried out in the direction of the programmed contour. Before calling L96, a feedrate must be programmed in the part program since the R parameters R28 and R30 are omitted.

If the parameters are assigned via the menu display, the desired cycle number L95/L96 must be entered.

The following values are entered in the menu display or programmed directly in the part program as parameter assignments:

Symbol	Parameter	Description
	R20	Subroutine number under which the contour is stored
Ap1	R21	Starting point of the contour in X (absolute)
Ap2	R22	Starting point of the contour in Z (absolute)
S1	R24	Finishing cut depth in X (incremental)
S2	R25	Finishing cut depth in Z (incremental)
Zt	R26	Depth of roughing cut in X or Z (incremental) (not required for roughing where R29 = 21 to 24)
	R27	Tool nose radius compensation (G41, G42)
F	R28	Feedrate (with cycle L95 only)
	R29	Type determination for roughing and finishing
	R30	Feed factor for infeed with relief cutting
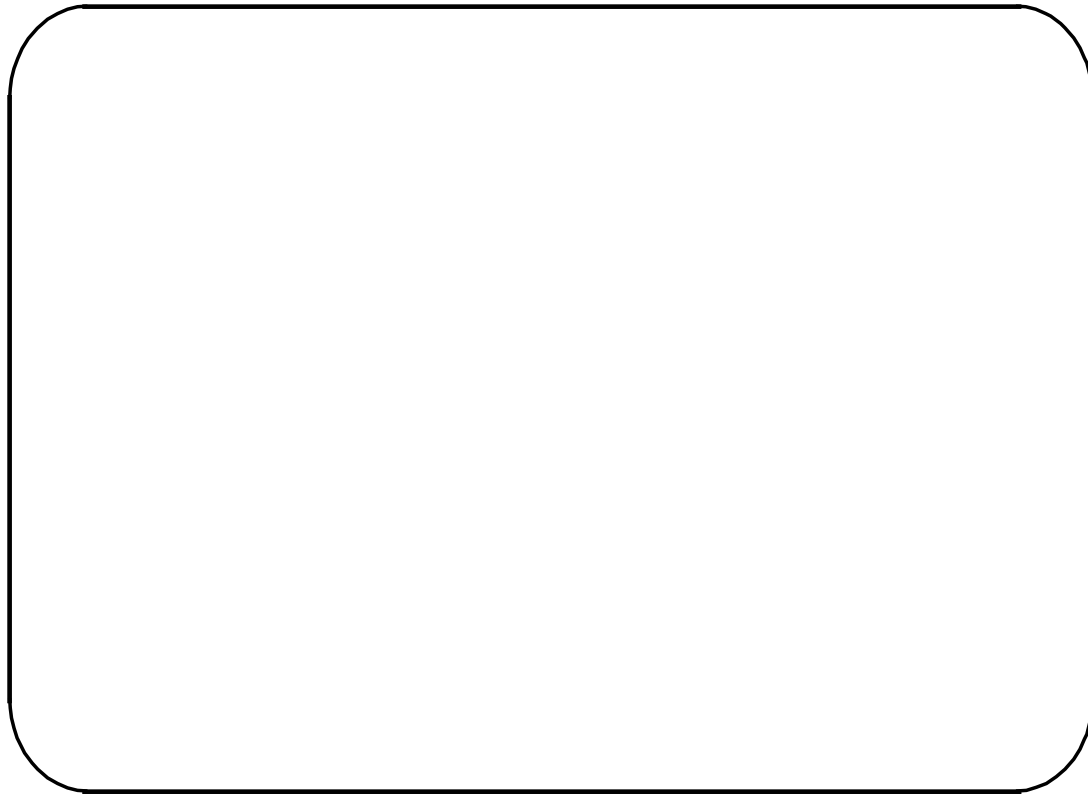
R20: Subroutine number

The subroutine number intended for programming the final contour must be specified in R20.

The subroutine can comprise any number of blocks, but at least two. A traverse path must be programmed in each block. The final contour can be defined with blueprint programming (see Programming Guide). Skippable blocks are permitted in the contour.

A contour element with the **maximum** diameter must be provided at the end of the contour.

The starting point of the contour must not be programmed in the contour subroutine. This is defined with parameters R21 and R22 (contour starting points).

Ap1 R21: Starting point of the contour in X
Ap2 R22: Starting point of the contour in Z

The parameters R21 and R22 must be supplied with the contour starting points. With roughing, the points are automatically approached by the finishing increment R24/R25 plus a safety margin of 1 mm. If this margin is insufficient, the contour starting points R21 and R22 must be shifted accordingly.
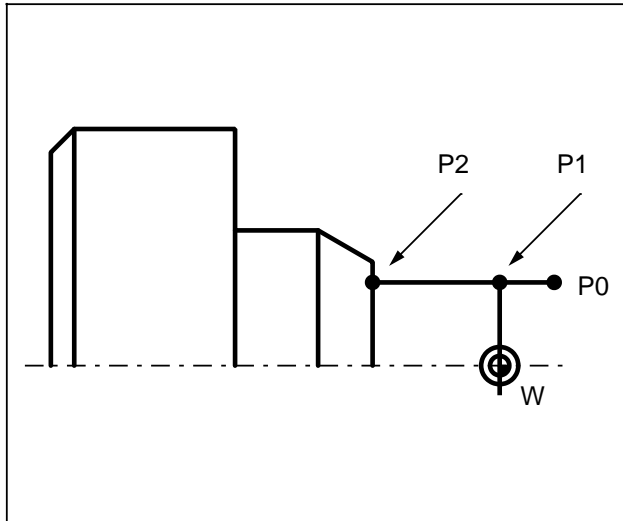
Example:

Point P1 corresponds to starting points R21 and R22, e.g. R21 = 20, R22 = 0, P0 = displaced starting point

Point P2 must be programmed as the first point in the contour subroutine, e.

```
L102
N05 X20 Z-15  LF
N10 X25
N15 ...
N20 ...
```

Cycle L95 approaches this starting point as follows:

In the case of longitudinal machining, the starting point in cycle L95 is approached with both axes simultaneously. This applies to the values 11 **/** 21 **/** 31 **/** 41 **/** 13 **/** 23 **/** 33 **/** 43 of parameter R29.

In the case of face machining, the starting point is always approached with both axes simultaneously, except for finishing (R 29=22 or 24), with a final machining allowance of at least a value not equal to zero (R24 or R25 or both values not equal to zero). In such cases, first the Z axis and then the X axis is traversed.

Explanation of parameters R24, R25 and R29 for starting point approach with face machining:

R29	R24 (final machining allowance in X)	R25 (final machining allowance in Z)	Approach strategy
12 and 14	any	any	X **/** Z together
22 and 24	R24=0	R25=0	X **/** Z together
	R24=0	R25 0	first Z, then X
	R24 0	R25=0	first Z, then X
	R24 0	R25 0	first Z, then X
32 and 34	any	any	X **/** Z together
42 and 44	any	any	X **/** Z together

S1 R24: Final machining allowance in X (incremental)
S2 R25: Final machining allowance in Z (incremental)
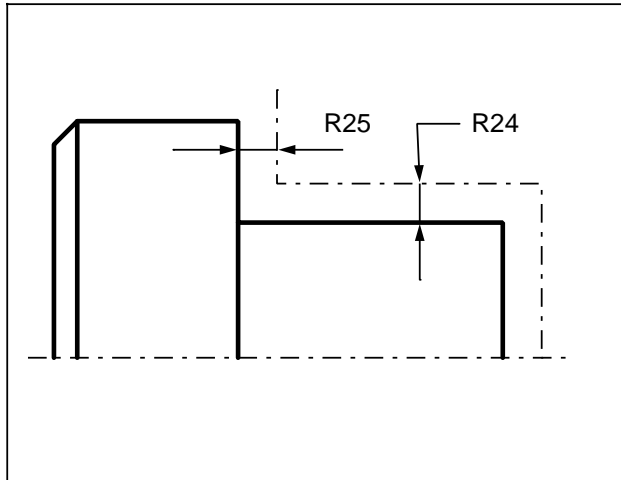
The contour is shifted by the amount of finishing increment entered (R24, R25). In the "roughing" machining mode, roughing is carried out down to this finishing increment.

In the "finishing" machining mode, stock removal is effected parallel to contour down to the finishing increment. Roughing depth R26 need not be programmed.

If R24/R25 are supplied with 0, the tool travels along the contour final dimensions.

Zt R26: Roughing chip depth with X or Z (incremental)

During roughing the cycle checks whether the current chip depth is less than double the roughing chip depth R26. If the current chip depth is less than double, then the following applies to the last two cuts:

R26 roughing chip depth = current chip depth/2.

R27: Tool nose radius compensation (G41, G42)

The cycle selects and deselects the tool nose radius compensation independently when selected by R27 = (G41, G42). (See R29 for further details.)

F R28: Feedrate

R28 must be supplied with the desired feedrate. To execute the cycle at a constant cutting rate, the "constant cutting rate G96" function must be selected before calling the cycle.

R30: Feedrate factor for rate of infeed with undone cutting

The plunge-cutting rate can be influenced by the feed factor R30. The feedrate factor must then be supplied with a value less than or equal to 1. The plunge-cutting rate is the product of R28 and R30.

R29: Type of machining for roughing and finishing

If parameters are assigned without the menu display, the type of machining (R29) is defined according to table 2.1:

The type of machining selected (R29) gives information on the type of cutting. Roughing or finishing, external or internal machining, and whether longitudinal or face.
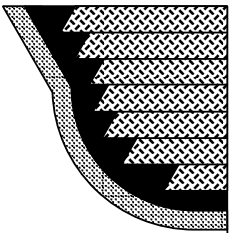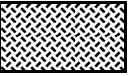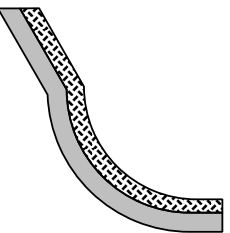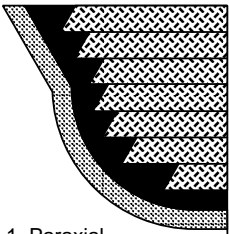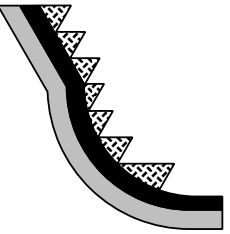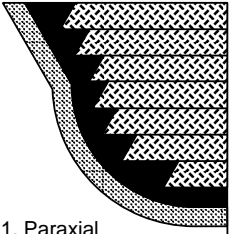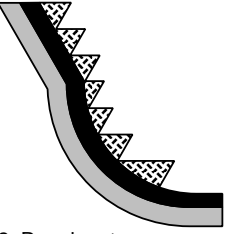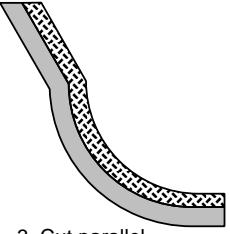
1. Paraxial roughing	Removed material		R29 = 11 longit. (Z) external R29 = 12 face (X) external R29 = 13 longit. (Z) internal R29 = 14 face (X) internal
	Finish cut parallel to contour		R29 = 21 longit. (Z) external R29 = 22 face (X) external R29 = 23 longit. (Z) internal R29 = 24 face (X) internal *)
1. Paraxial roughing	2. Rough cut parallel to contour		R29 = 31 longit. (Z) external R29 = 32 face (X) external R29 = 33 longit. (Z) internal R29 = 34 face (X) internal *)
1. Paraxial roughing	2. Rough cut parallel to contour	3. Cut parallel to contour	R29 = 41 longit. (Z) external R29 = 42 face (X) external R29 = 43 longit. (Z) internal R29 = 44 face (X) internal *)

Table 2.1   Explanation of machining types R29

---

*) In these cases, the cycle activates the tool nose radius compensation (TNRC) automatically in the correct direction if beforehand a selection via R27 = (G41 or G42) has been made. Moreover, it controls the timely selection and cancellation of the TNRC itself. TNRC is internally suppressed in the case of paraxial roughing. It is cancelled at the end of the cycle and must be programmed again, if required.

Notes on contour definition

• With cycles L95 and/or L96 it is possible to machine ascending contours. A contour element containing the maximum contour diameter must be programmed at the end of the contour definition, where the final diameter must be larger than the initial diameter. "Ascending" means that all the contour sections programmed in the contour form an angle W_i of between 90° and 180° with the positive horizontal axis, i.e. 90° W_i 180°.
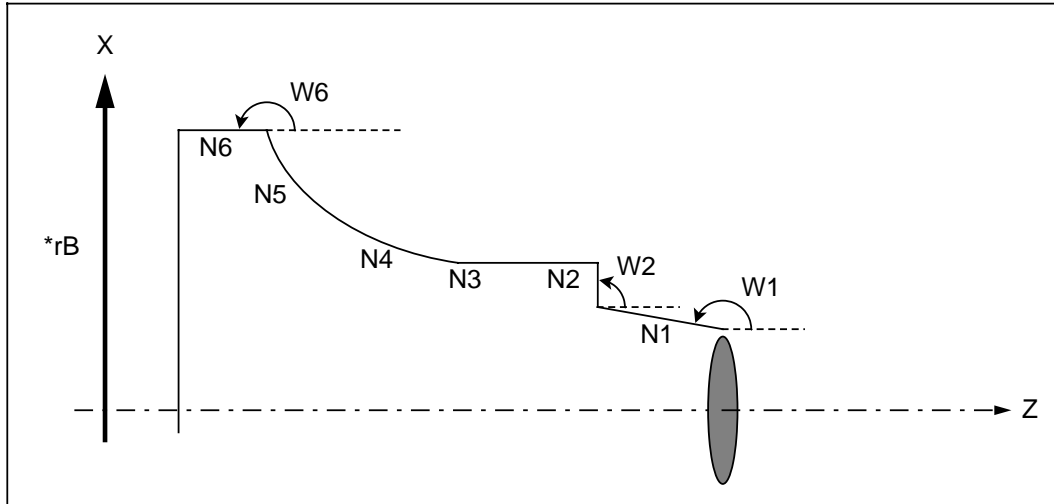
The figure above shows an example of an ascending contour. The contour section N1 embraces an angle of between 90° and 180° with the positive Z axis, the path section N2 an angle of exactly 90° and contour sections N3 and N6 an angle of exactly 180°.

• If cycle L95 is used, the programmed contour can also include relief cut elements. A contour can include a maximum of 10 relief cut elements. Relief cutting when roughing is allowed.

1 Relief cut element

SINUMERIK System 800, Cycles, UMS 4 (PG)
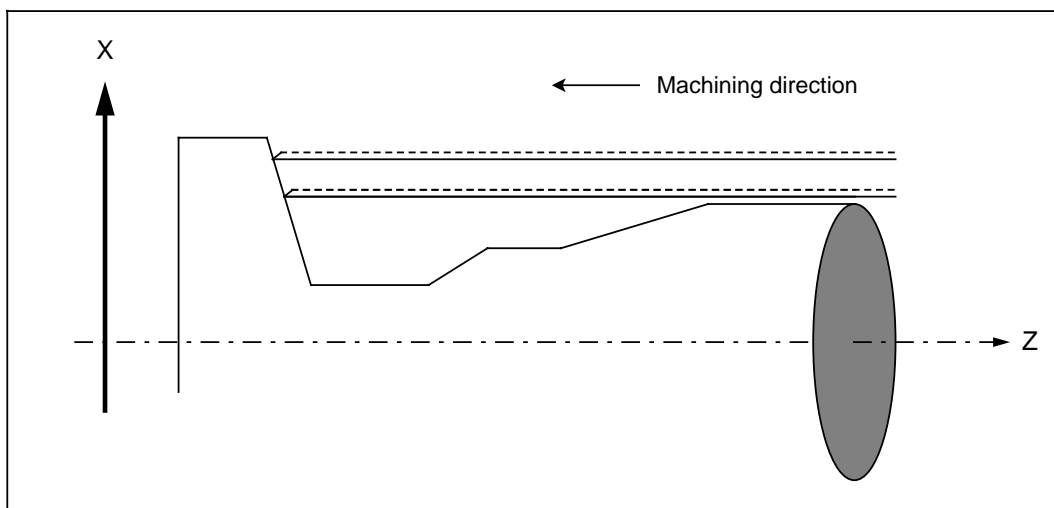
Several relief cut elements can follow each other in succession.

Relief cut elements contain "descending" contour sections, which, analogous with "ascending" contour sections, produce an angle between 0 and 90 degrees, $0 < W_i < 90°$, to the positive horizontal axis.

2 relief cut elements
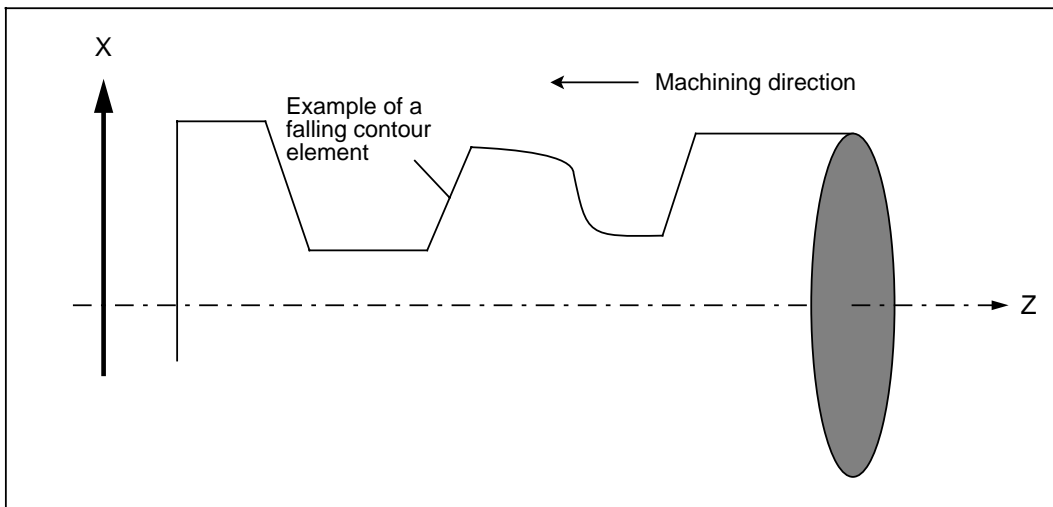
Circular path sections with relief cutting (contour sections programmed with G2 or G3) must be programmed in such a way that the starting point and finishing point lie within one quadrant of the coordinate system. Larger radii must be programmed in several blocks of the contour subroutine (see programming example 1).

Examples showing illegal relief cut elements
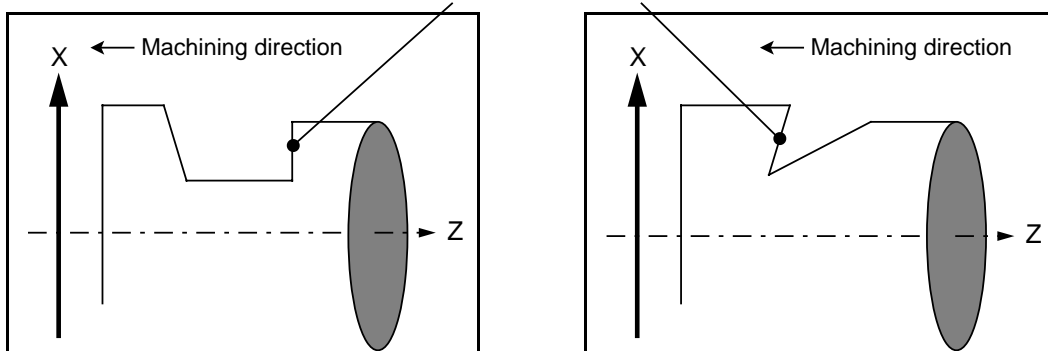
Paraxial relief cut elements cannot be machined when infeeding into a relief cut (see example). Nor is it possible to produce relief cuts only from descending contour elements (see example), as this would cause a collision when the tool is withdrawn after roughing.

SINUMERIK System 800, Cycles, UMS 4 (PG)

- A defined final contour can be roughed paraxially either parallel to the horizontal axis (longitudinal machining) or to the vertical axis (face machining). This is defined in parameter R29.

  Not every contour with relief cut elements can be both machined longitudinally and on the face with stock removal cycle L95. This depends on the geometry of the contour and the turning tool.

  To ensure that the correct type of machining is used for a workpiece for a defined contour, the following explains which contour types should be machined longitudinally, which contour types should be face machined and which types can be machined in both ways. The contours are therefore either described as longitudinal or face contours.

  A longitudinal contour is a contour on which all intersection points in longitudinal machining lie on ascending contour elements and not on contour elements which are parallel to the horizontal axis.

Example of longitudinal contour
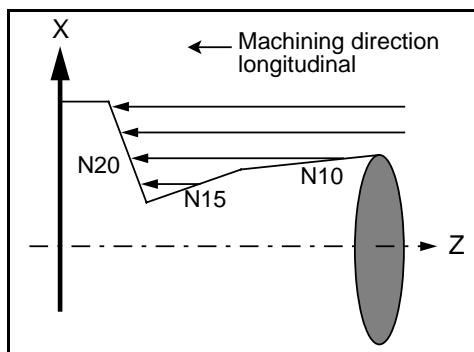
The determined intersection points for longitudinal machining are only located in N20, i.e. in a rising contour element
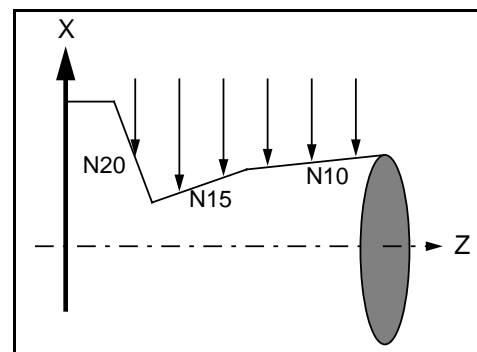
To compare:
Face machining also produces points of intersection on "descending" contour elements in N10 and N15

All intersection points on a face contour are located either on descending contour elements or contour elements which are parallel to the horizontal axis.

Example of a face machined contour
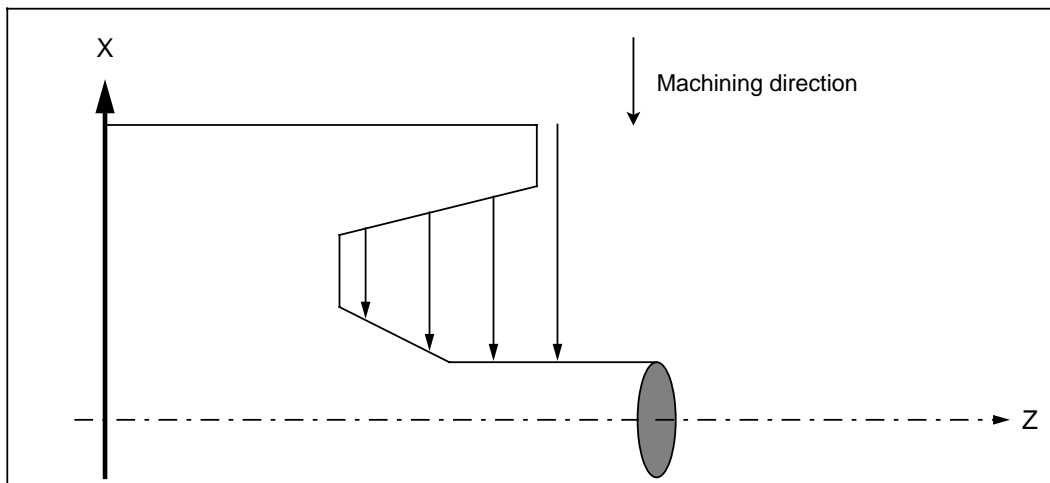
By the same token, contours without relief cut elements are both longitudinal and face contours and can be parameterized with any R29 value. It is however better to use cycle L96 (stock removal cycle without relief cut) to machine these contours.

For each contour the machining type (in R29) must be defined according to the above definition, i.e. longitudinal contours must be machined longitudinally (R29 is always uneven) and face contours therefore face machined (R29 is always an even number). It is however also possible to machine contours which fulfill certain conditions using the other machining method if the correct tool is used.

Longitudinal contours can be face machined (and face contours can be machined longitudinally) if:

– For longitudinal contours:
No angle of W_i 45° to the horizontal axis is contained in any "descending" path sections.

For face contours:
No angle of $W_i < 45°$ to the horizontal axis is contained in any "descending" path sections.

– The following interrelationship is maintained between the final machining allowances in X and Z:

Final machining allowance in X $c \cdot$ final machining allowance in Z

with

$c =$ amount (Diff $X_{RE_{max}}$ **/** Diff $Z_{RE_{max}}$)

and by Diff $X_{RE_{max}}$ (or Diff $Z_{RE_{max}}$) is meant the coordinate difference between the starting point and end point in X (or Z) of the descending contour section which together with the horizontal axis produces the largest angle. Accordingly, c is the tangent of this contour section.

Example:

The following contour (which is actually a longitudinal contour) is to be face machined.
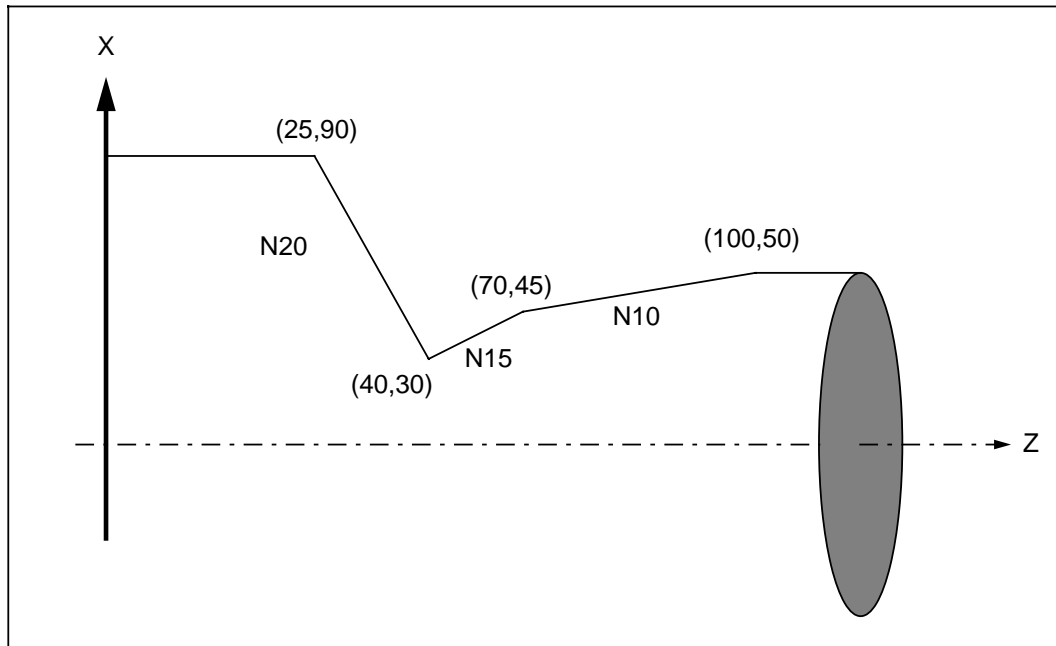
When selecting the final machining allowance the following applies:

The contour contains two descending contour elements, N10 and N15. These form the angles

$W1 = 9.463°$ (N10)
$W2 = 26.565°$ (N15)

with the positive Z axis. This produces the value C (which refers to N15) where

c = amount $((45–30) / (70–40)) = 0.5$.

Therefore, if a final machining allowance in Z of 3.5 mm is selected the final machining allowance in X must be at least 1.75 mm or conversely, with a final machining allowance in X of 0.5 mm, the final machining allowance in Z must not be greater than 1 mm.

This relationship between the final machining allowances is not required if the contour is to be machined longitudinally. Then any values can be entered in R24 and R25.

If the contour is only to be finish cut, the longitudinal contours must be parameterized with the R29 values 21 and 23 and the face contours with R29=22 and/or 24.

Example 1: "Complete machining with external longitudinal stock removal" machining type selected via softkey

```
%1
N05  G95  G0  X120  Z10  D01  T01  S1000  M04  LF          Select stock removal
N10  R20=105  R21=28  R22=0  R24=1  LF                            position
     R25=1  R26=5  R27=42 R28=.2  LF
     R29=41  R30=.5  L95  P1  LF                          Call stock removal cycle
N20  G0  X200  Z200  LF                                   L95 with recess cut element
N25  M30  LF
```

For automatic stock removal, the final contour of the finished part must be described. This is stored as a subroutine and called within the stock removal cycle. In this example, the contour is programmed in subroutine L105 and stored in the program memory.

```
L105
N50  G01  X32  Z-2  F.05  LF
N55  Z-22  LF
N60  X74  LF
N65  G03  X94  Z-32  B10  LF                             Radius programming quadrant 1
N70  X88.1  Z-39.1  B10  LF                              Radius programming quadrant 2
N75  G1  A225  A180  X50  Z-78  B12  B3  LF             3 points contour + radius + radius
N80  X106  Z-78  LF
N85  X112  A135  LF
N90  M17  LF
```

Note:

Relief cut element: this radius extends over more than one quadrant and must therefore be divided into two steps (compare N65 and N70).
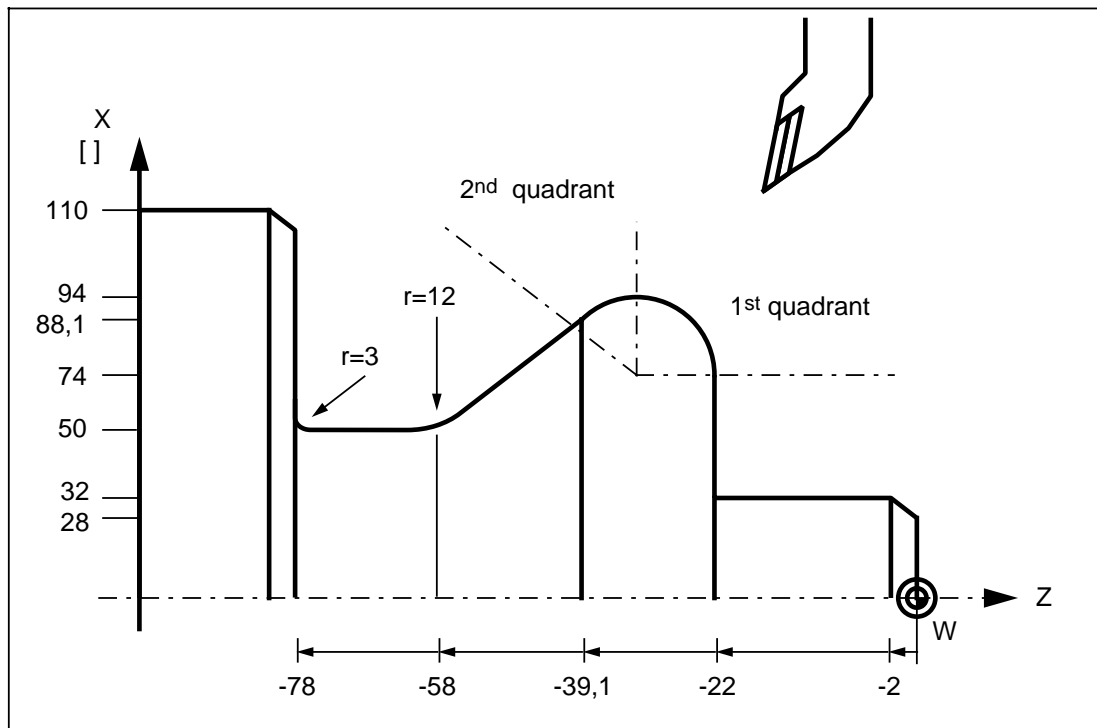
Example 2:   **"Roughing with facing external stock removal" machining type selected via softkey**

```
%2
N05  G96  G0  X80  Z40  D01  T01  S2000  M04  LF
N10  R20=106  R21=140  R22=25  R24=2  LF
     R25=2  R26=0  R27=41 R28=.05  LF
     R29=22  R30=1  L95  P1  LF
N20  G0  X200  Z200  LF
N25  M30  LF


L106
N50  X110  Z25  LF
N55  X90  Z15  LF
N60  X60  LF
N65      Z30  LF
N70  M17  LF
```

Select stock removal position

Call stock removal cycle L95
with recess cut element
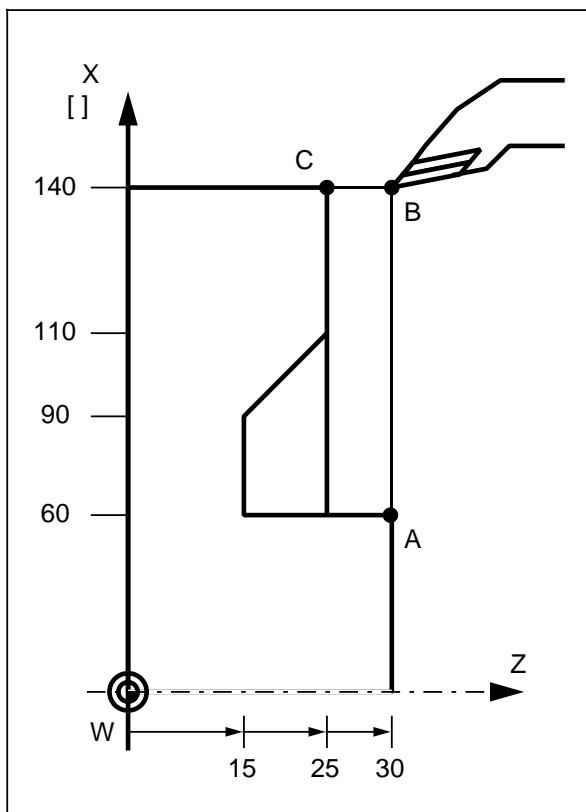
Corner point B also represents the change-over point to the finishing cycle.

The cycle calculates this point from points A and C.

2.1.4 L97 Thread cutting cycle

Using this cycle, external threads, internal threads, taper threads and transversal threads can be cut. Infeed is automatic and is degressively quadratic, the cut cross-section thus remains constant.

The following values are entered in the menu display or programmed directly into the part program as parameter assignments.

Symbol	Parameter	Description
P	R20	Thread pitch
Ap1	R21	Starting point of thread in X (absolute)
Ap2	R22	Starting point of thread in Z (absolute)
	R23	Number of idle passes
T	R24	Thread depth (incremental), sign required to define inside or outside thread: + = inside thread / − = outside thread, transversal thread
S	R25	Finishing increment (incremental)
se	R26	Approach path (incremental)
sa	R27	Run-out path (incremental)
	R28	Number of roughing cuts
W	R29	Infeed angle
Ep1	R31	End point of thread in X (absolute)
Ep2	R32	End point of thread in Z (absolute)
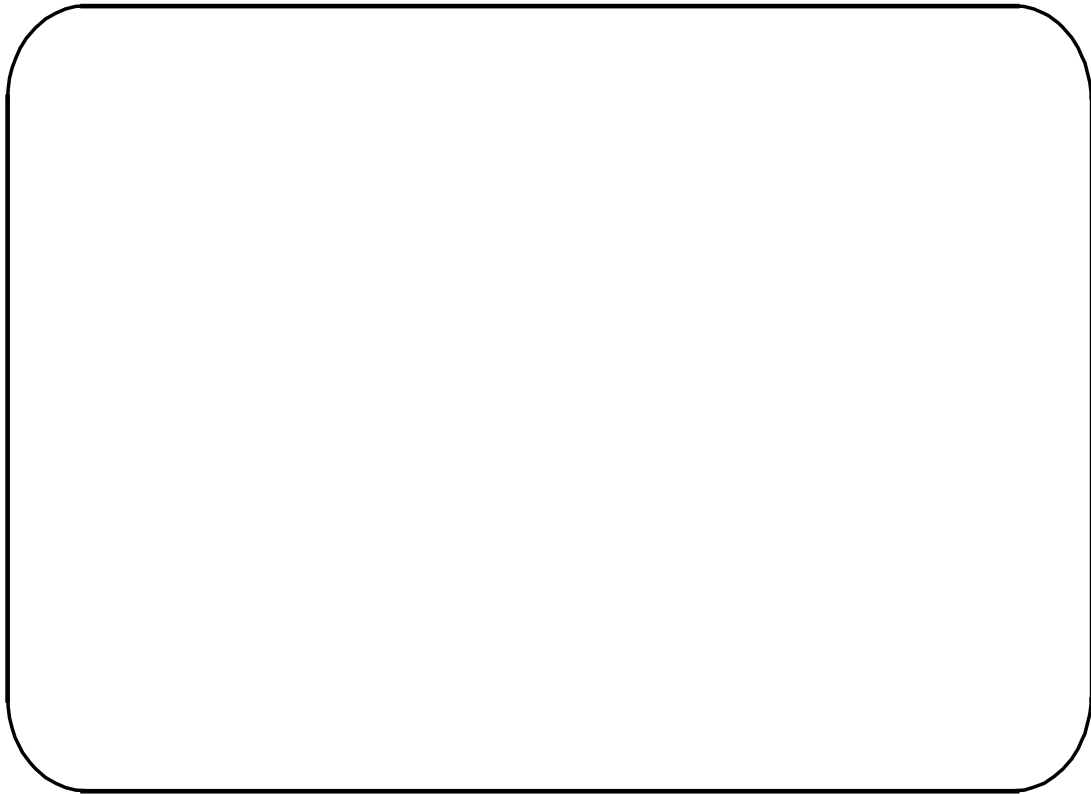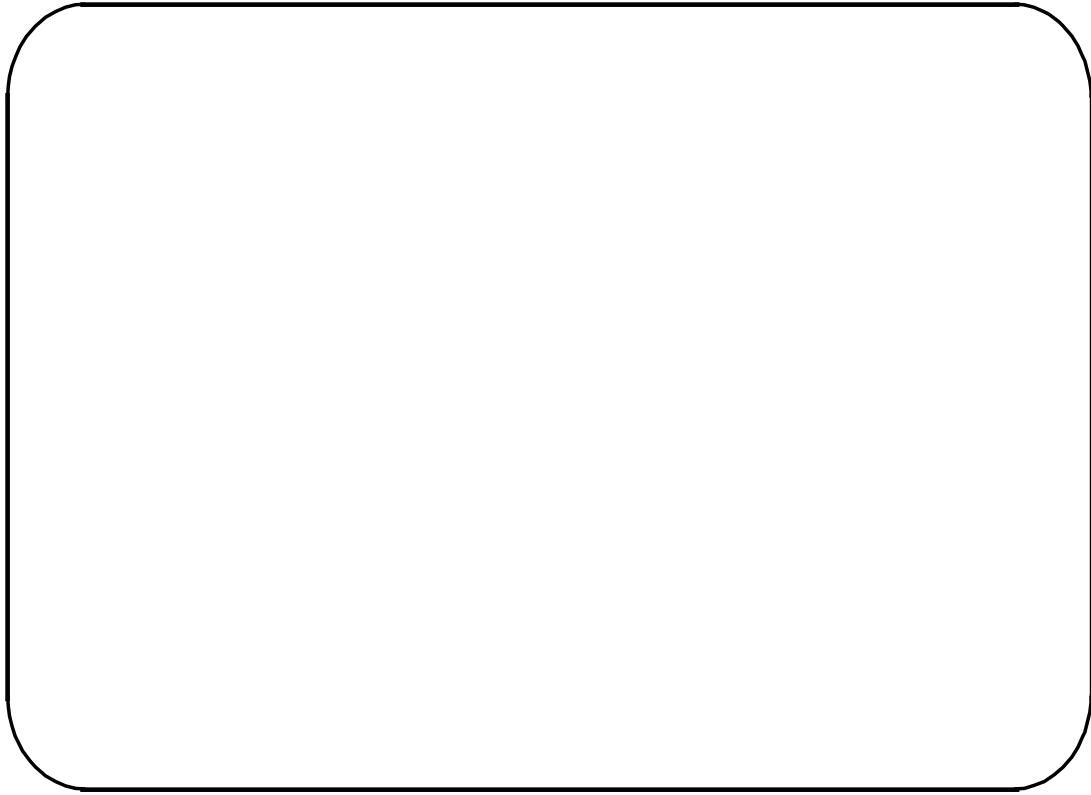
Thread cutting cycles: Distinction between transversal and longitudinal threads

Both longitudinal and transversal threads are possible with L97 and L98. The distinction
depends on the angle resulting from the thread starting point, P1, and the first
intermediate/end point of the thread, P2. If an angle greater than 45° results, the thread is
machined transversally (see example).

- Longitudinal thread
  The | Delta Z1 | amount must be greater than or equal to the | Delta X1 | amount, i.e. the
  angle is less than or equal to 45°.

L99			L97		
Ap1	R11=30	Starting point in X	Ap1	R21=30	Starting point in X
Zp1	R12=42	First intermediate point in X	Ap2	R22=0	Starting point in Z
Ap2	R21=0	Starting point in Z	Ep1	R31=42	Thread end point in X
Zp3	R22=-52	First intermediate point in Z	Ep2	R32=-52	Thread end point in Z

6ZB5 410-0BQ02
SINUMERIK System 800, Cycles, UMS 4 (PG)
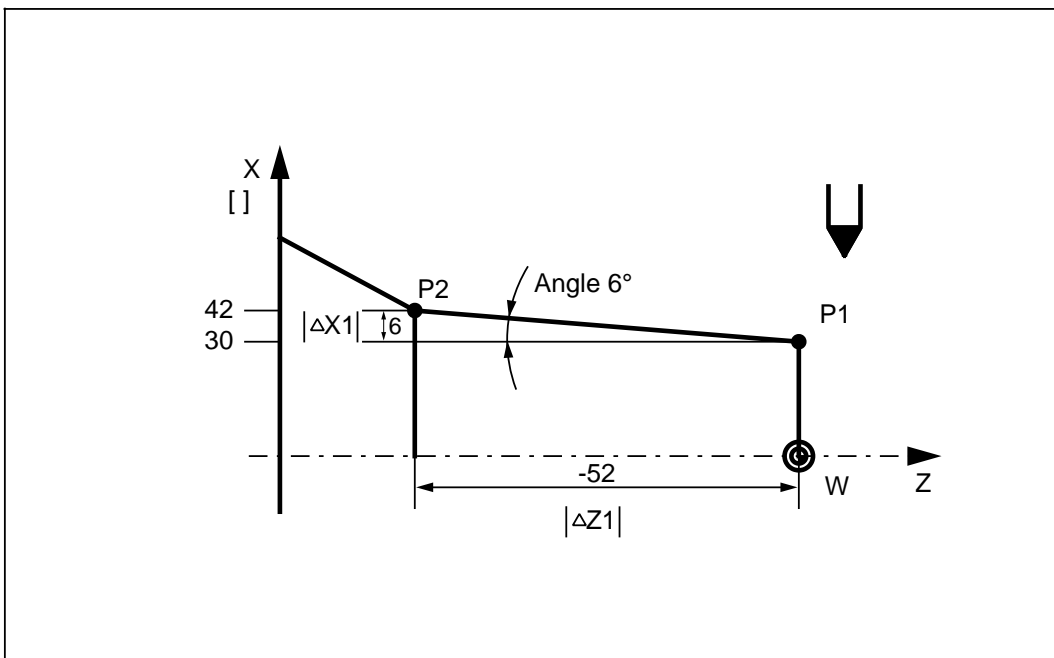
• Transversal thread
The | Delta Z1 | amount must be less than the | Delta X1 | amount, i.e. the angle is greater than 45°.

L99			L97		
Ap1	R11=20	Starting point in X	Ap1	R21=20	Starting point in X
Zp1	R12=120	First intermediate point in X	Ap2	R22=0	Starting point in Z
Ap2	R21=0	Starting point in Z	Ep1	R31=120	Thread end position in X
Zp3	R22=-30	First intermediate point in Z	Ep2	R32=-30	Thread end position in Z

SINUMERIK System 800, Cycles, UMS 4 (PG)
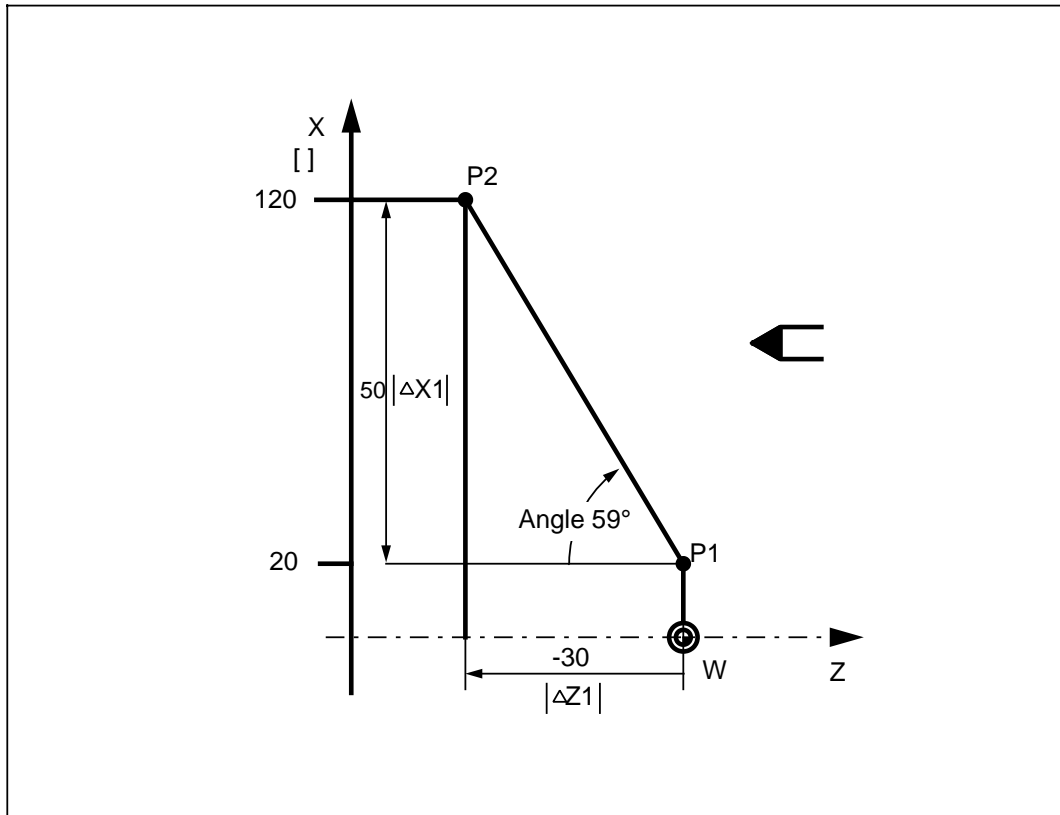
P R20: Thread pitch

The thread pitch must be entered as a paraxial value without sign.

Ap1 R21: Starting point of thread in X (absolute)
Ap2 R22: Starting point of thread in Z (absolute)

The parameters R21 and R22 represent the original starting points of the thread (A). The starting point of the thread cycle is at point B, which is located at parameter R26 (approach path) in front of the thread output point.

With a longitudinal thread, the starting point B is 1 mm above value R21, and 1 mm ahead of value R22 in the case of transversal threads. This retracted plane is generated automatically by the control.

The thread cycle can be called from any slide position, the approach to point B is effected in rapid traverse.
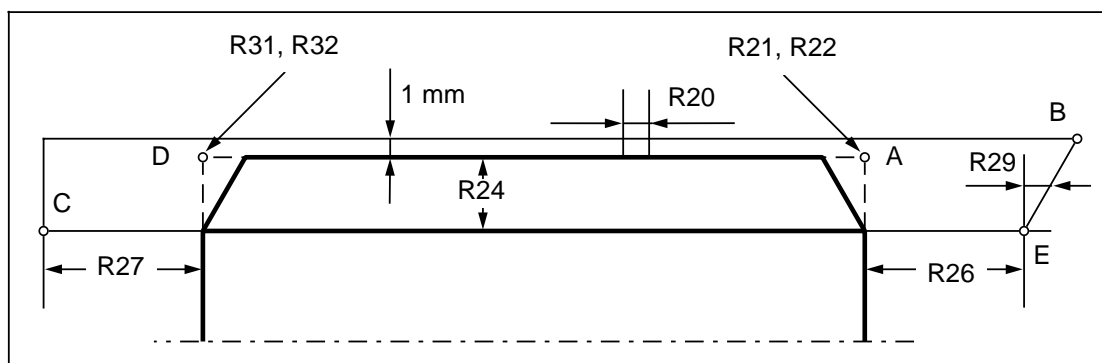
R23: Idle passes

Any number of idle passes can be selected.

T R24: Thread depth (incremental)

The depth of the thread is entered using parameter R24. The sign determines the infeed direction, i.e. whether it is an outside, inside or transversal thread. (+ inside thread, -outside thread, transversal thread).

S R25: Finishing cut depth (incremental)
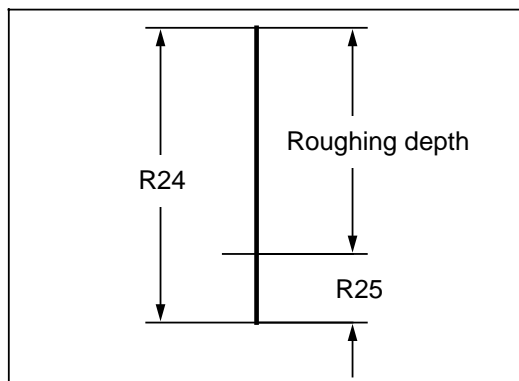
If a finishing cut depth is programmed under R25, this depth is subtracted from the thread depth and the remaining value is divided into roughing cuts.

After the roughing cuts have been completed, a finishing cut is made and then the idle passes programmed under R23 are executed.

The roughing depth is automatically calculated and divided into roughing cuts.

se R26: Approach path (incremental)
sa R27: Run-out path (incremental)

The approach and run-out paths are entered as paraxial, incremental values without signs.

In the case of taper threads, the control calculates the approach and run-out path distances in relationship to the taper and determines the corner points B and C.
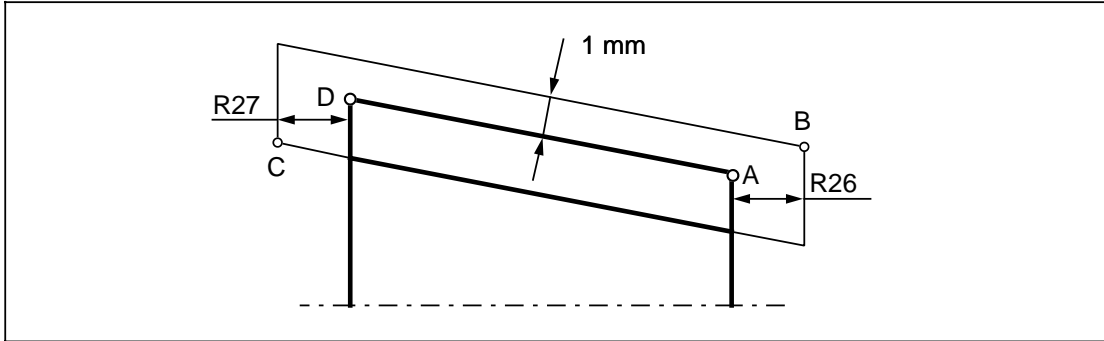
R28: Number of roughing cuts

The parameter value determines the number of thread roughing cuts. The control automatically calculates the individual infeed depths keeping the cross-sectional area of the cut constant. Thereby, it is guaranteed that the cut pressure from the first to the last roughing cut remains the same.

The depth of the current cut t is calculated with the following equation:

$$t= \frac{t}{R28} \cdot i \qquad \begin{array}{l} t=R24 - R25 \\ i=\text{current cut} \end{array}$$

Example:
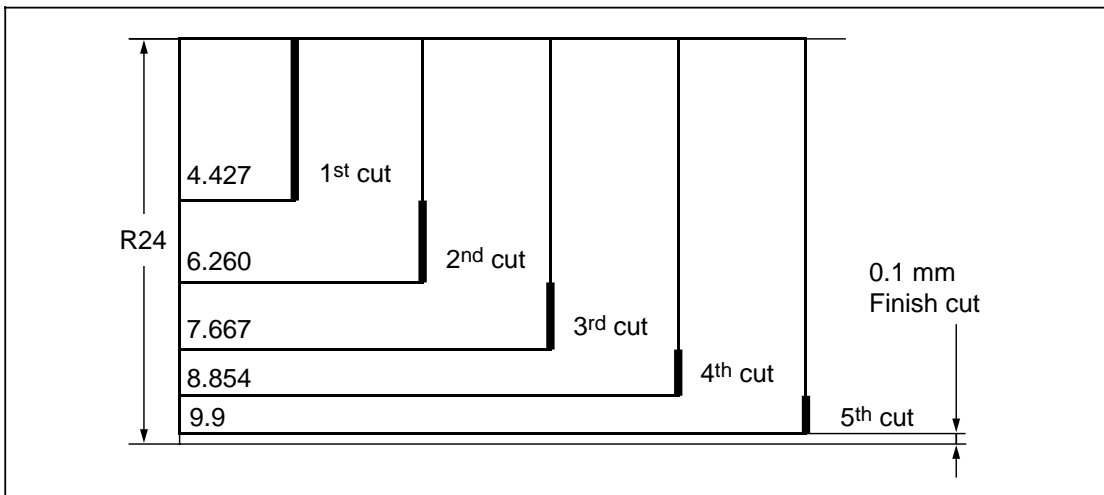
Thread depth:	R24=10 mm
Number of roughing cuts:	R28=5
Finishing cut depth:	R25=0.1 mm

W R29: Infeed angle for longitudinal or transversal threads

The tool can be infed perpendicular to the direction of cutting or along the flank. The angle is input without sign and must not exceed half the value of the flank angle.

If the tool is to be infed perpendicular to the axis, R29 must be assigned 0.
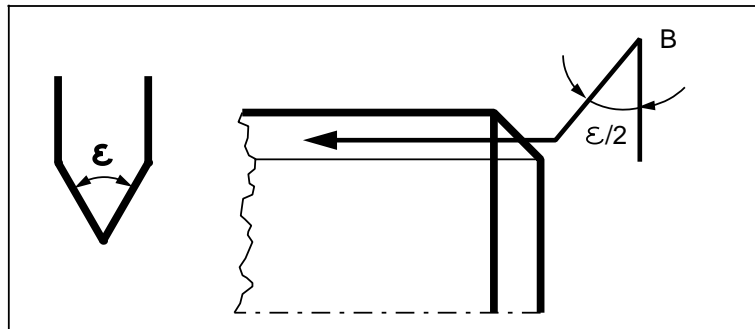
Metric thread 60°
/2=30°
R29=30

Ep1 R32: Thread end point in X (absolute)
Ep2 R32: Thread end point in Z (absolute)

The parameters R31 and R32 represent the original end points of the thread (D). The reversal point of the thread cycle is at point C, which is located after the thread end point by the length of the run-out path in parameter R27.

Example: "External" thread type selected via softkey

```
%97
N05  G95  G0  X50  Z10  D01  T01  S1000  M04  LF          Select thread cutting position
N10  R20=2  R21=42  R22=0  R23=0  LF
     R24=-1.23  R25=0  R26=10 R27=3  LF
     R28=5  R29=30  R31=42  R32=-35  LF
     L97  P1  LF                                          Call thread cutting cycle
N15  G0  X200   Z200  LF
N20  M30  LF
```

SINUMERIK System 800, Cycles, UMS 4 (PG)
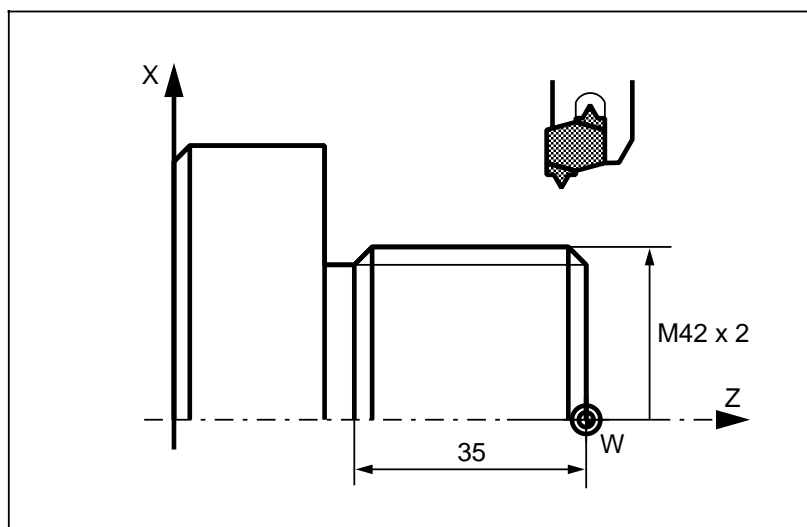
2.1.5   L99 Chaining of threads (four-point thread cutting cycle)

Cycle L99 permits several contiguous threads with different pitches to be cut. These contiguous threads may be either longitudinal or transversal threads.

The following values are entered in the menu display or are programmed directly in the part program as parameter assignments:

Symbol	Parameter	Description	
Ap1	R11	Starting point of thread in X	(absolute)
Zp1	R12	First intermediate point of thread in X	(absolute)
Zp2	R13	Second intermediate point of the thread in X	(absolute)
Ep1	R14	Thread end position in X	(absolute)
Ap2	R21	Starting point of thread in Z	(absolute)
Zp3	R22	First intermediate point of thread in Z	(absolute)
Zp4	R23	Second intermediate point of the thread in Z	(absolute)
Ep2	R24	Thread end position in Z	(absolute)
S	R25	Finishing increment	(incremental)
se	R26	Approach path	(incremental)
sa	R27	Run-out	(incremental)
	R28	Number of roughing cuts	
W	R29	Infeed angle	
	R35	Number of idle passes	
T	R36	Thread depth (incremental) with sign, depending on whether the thread is internal or ext.: + = internal thread - = external thread, transversal thread	
P1	R41	Thread pitch 1	
P2	R42	Thread pitch 2	
P3	R43	Thread pitch 3	

SINUMERIK System 800, Cycles, UMS 4 (PG)

Ap1 R11: Starting point of thread in X (absolute)
Ap2 R21: Starting point of thread in Z (absolute)

Parameters R11 and R21 represent the original starting points of the thread (A). The starting point of the thread cycle is at point B, which is positioned in front of the thread starting point by the length of the approach path in parameter R26:

In the diameter (X axis), starting point B is 1 mm above the parameter value R11; in the case of a transversal thread, it is 1 mm in front of value R21. This retraction plane is automatically created by the control. The thread cycle can be called from any slide position; the approach to point B is at rapid traverse.

Zp1 R12: First intermediate point of thread in X (absolute)
Zp3 R22: First intermediate point of thread in Z (absolute)

Parameters R12 and R22 represent the first intermediate point of the thread.

Zp2 R13: Second intermediate point of thread in X (absolute)
Zp4 R23: Second intermediate point of thread in Z (absolute)

Parameters R13 and R23 represent the second intermediate point of the thread. If only one intermediate point is to be entered, parameters R12 or R22, R13 or R23 and R14 or R24 must be given the same values.

Ep1 R14: Thread end position in X (absolute)
Ep2 R24: Thread end position in Z (absolute)

Parameters R14 and R24 represent the original end point of the thread (D). If no intermediate point is to be given, parameters R12 or R22, R13 or R23 and R14 or R24 and pitches R41, R42 and R43 must be given the same values.

SINUMERIK System 800, Cycles, UMS 4 (PG)

S R25: Finishing increment (incremental)

Roughing dimension

R36

R25
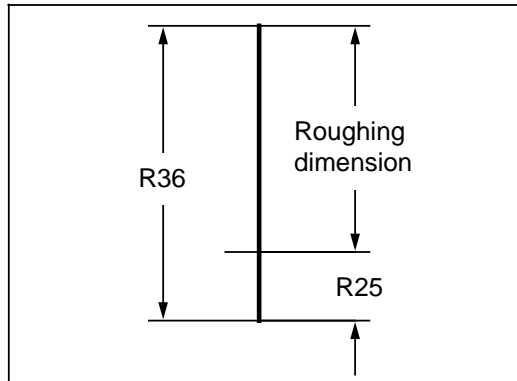
The finishing increment is entered in R25. If a finishing increment is programmed, it is subtracted from the thread depth R36 and the remaining value is divided into roughing cuts.

When the roughing cuts have been completed a finishing cut is made followed by the number of idle passes programmed in R35.

se R26: Approach path (incremental)
sa R27: Run-out path (incremental)

The approach and run-out paths are entered as incremental paraxial values without sign.

In the case of a tapering thread, the control converts the approach and run-out paths into the taper ratio and defines corner points B and C.
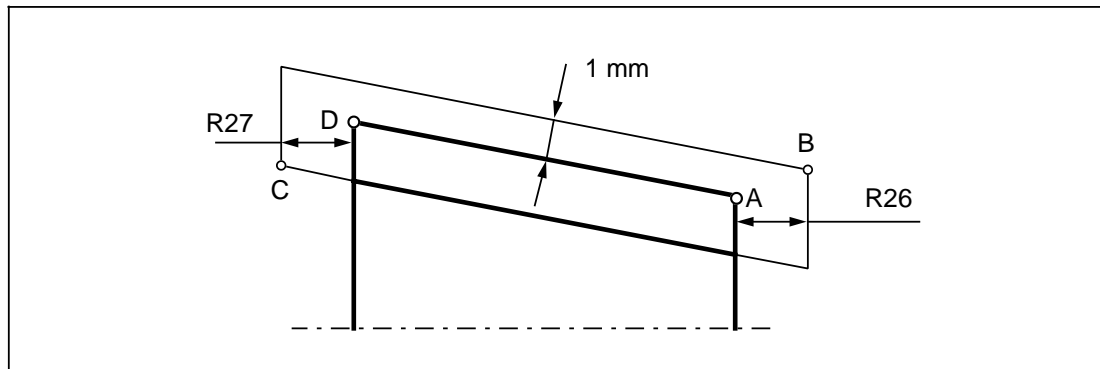
R28: Number of roughing cuts

The parameter value defines the number of thread roughing cuts. The control automatically calculates the individual infeed depths at constant chip cross-section. This ensures that the cut pressure remains the same from the first to the last roughing cut.

The current cut depth t is calculated according to the following equation:

$$t = \frac{t}{R28} \cdot i \qquad \begin{array}{l} t = R36 - R25 \\ i = \text{current cut} \end{array}$$

Example:
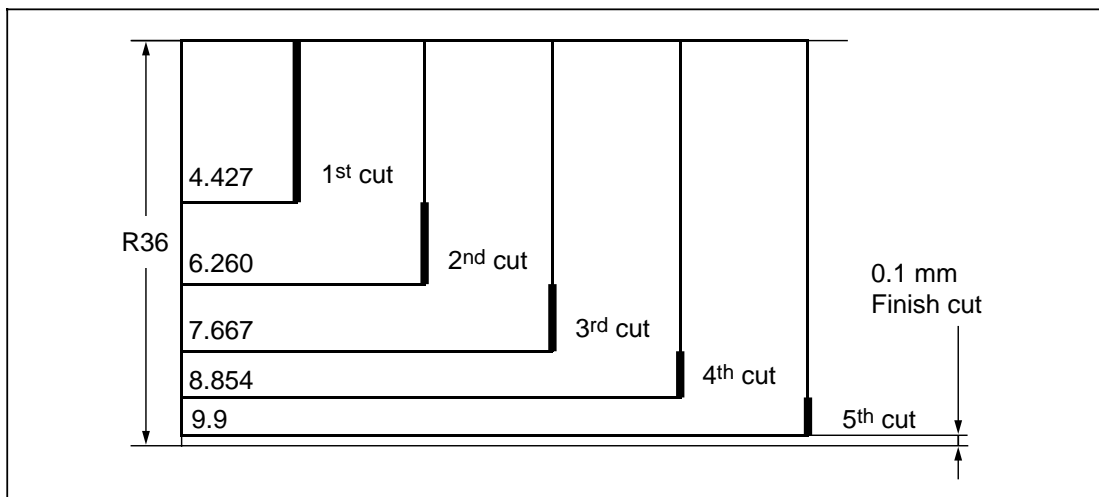
Thread: R36=10 mm
Number of roughing cuts: R28=5
Finishing increment: R25=0.1 mm

W R29: Infeed angle with longitudinal or transversal cuts

The tool can be infed perpendicular to the direction of cutting or along the flank. The angle is input without sign and must not exceed half the value of the flank angle.

If the tool is to be infed perpendicular to the axis, R29 must be assigned 0.
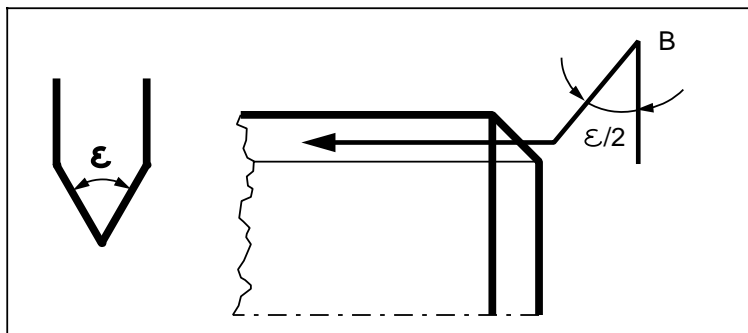
Metric thread 60°
/2=30°
R29=30

R35: Number of idle passes

The number of idle passes can be selected as required. It is entered in parameter R35.

T R36: Thread depth (incremental)

The thread depth is entered in parameter R36. The sign determines the infeed direction, i.e. whether it is an external or internal thread. (+ = internal thread, - = external thread, transversal thread).

P1 R41: Thread pitch 1
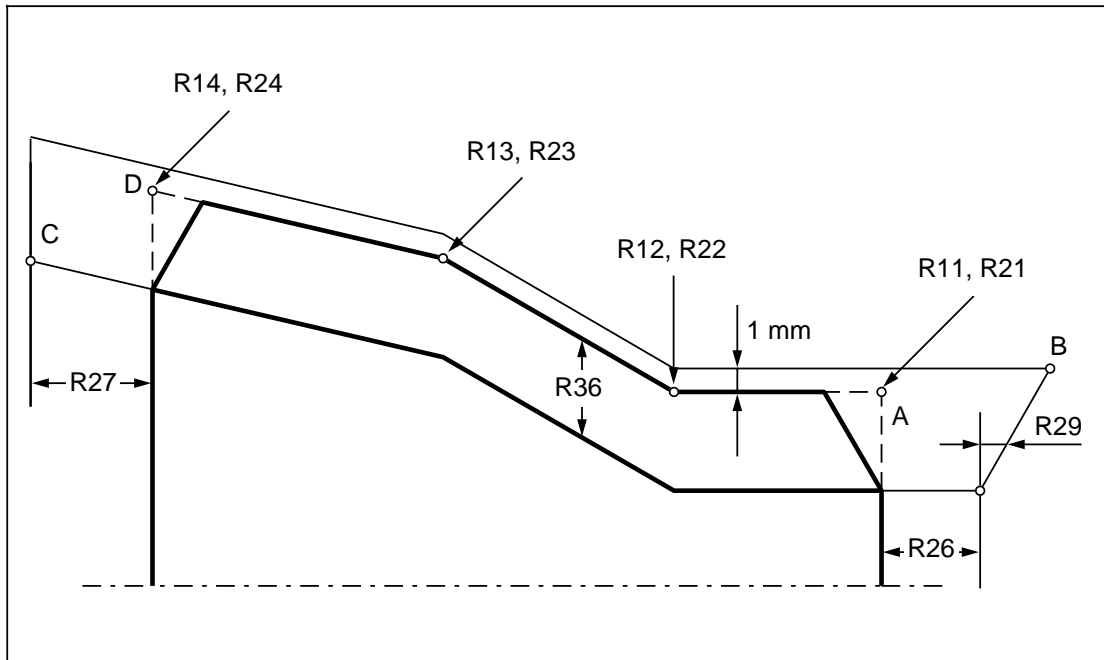### P2 R42: Thread pitch 2
P3 R43: Thread pitch 3

The parameters represent the values of the pitches for each element. The paraxial value is always entered without sign.

Example: "External thread cutting" machining type selected via softkey

```
%99
N05  G95  G0  X40  Z10  D01  T01  S1000  M04  LF          Select thread cutting position
N10  R11=30  R12=30  R13=36  LF
     R14=50  R21=0  R22=-30  LF
     R23=-60  R24=-80  R25=0  LF
     R26=10  R27=10  R28=5  R29=0  LF
     R35=1  R36=-0.92  R41=1.5  R42=2  LF
     R43=2  L99  P1
                                                         Call thread cutting cycle
N15  G0     X200  Z200  LF
N20  M30  LF
```

SINUMERIK System 800, Cycles, UMS 4 (PG)
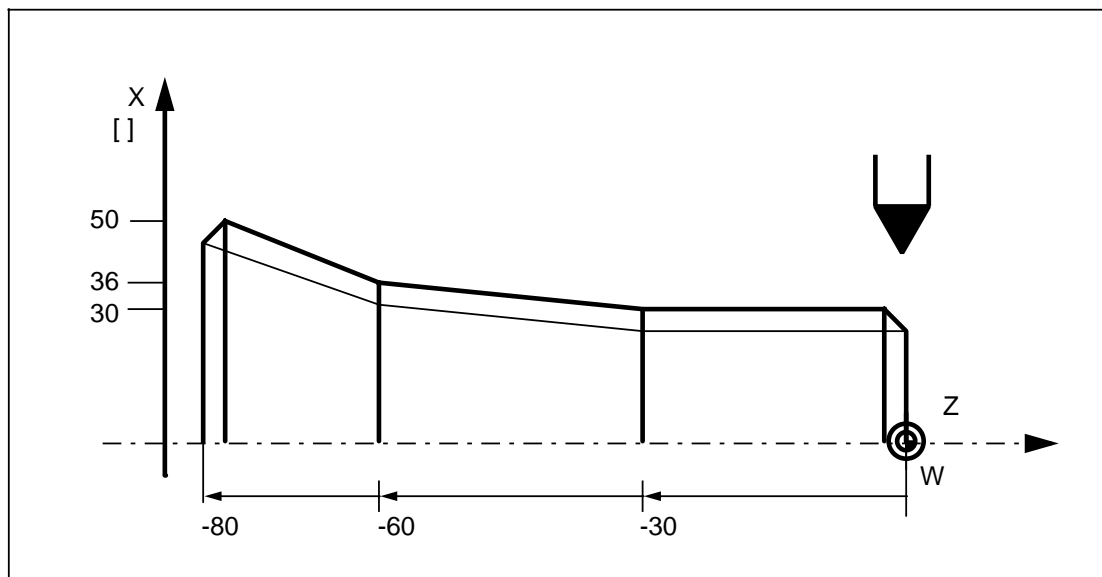
2.1.6   L98 Deep hole drilling cycle

This cycle permits deep holes to be drilled. For chip removal purposes, the drill can be moved to the starting point from each infeed depth.

The following values are entered in the menu display or programmed directly in the part program as parameter assignments:

Symbol	Parameter	Description
	R11	0=With chip breaking, 1=With chip removal
Ap	R22	Starting point in Z (absolute)
Db	R24	Enter amount of degression (incremental) without sign
T1	R25	Enter the first drilling depth (incremental) without sign
T2	R26	Final drilling depth (absolute)
t1	R27	Dwell time at the starting point (for chip removal)
t2	R28	Dwell time at the bottom of drilling hole (chip breaking)
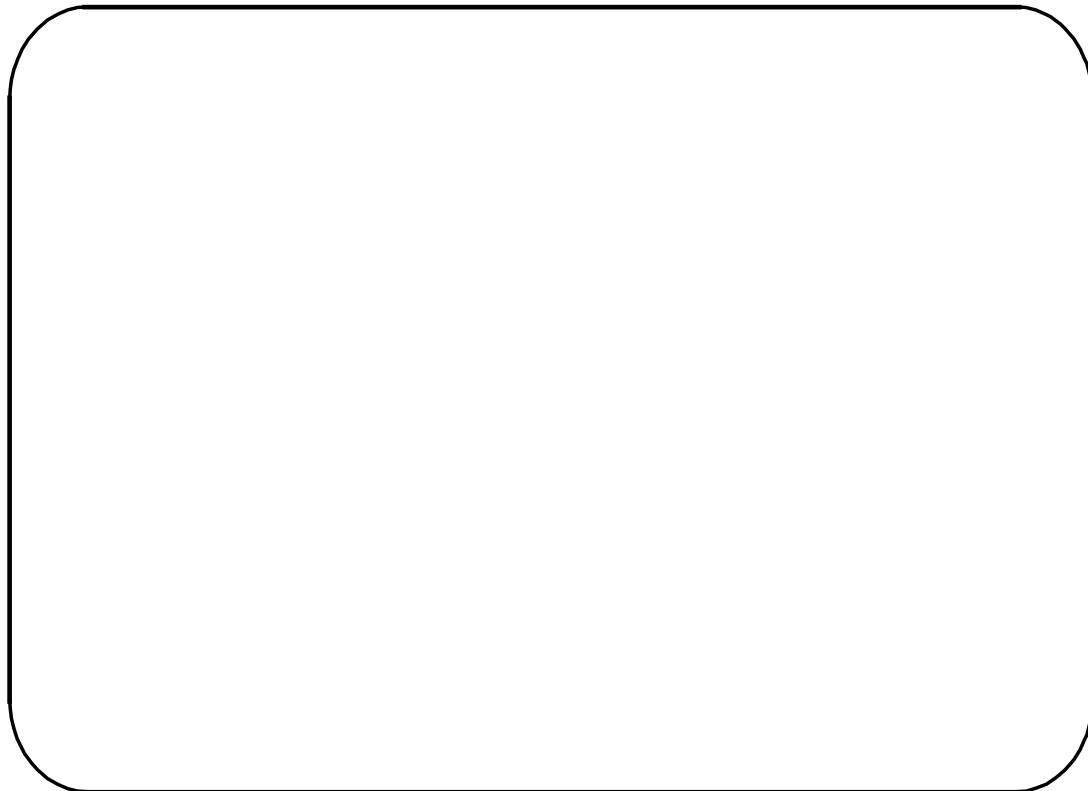
R11: Chip breaking / chip removal

If R11 is assigned with 0, the drill bit is retracted by 1 mm for chip breaking each time that the drill depth has been reached.

If R11 is assigned with 1, the drill bit travels to the reference plane for chip removal each time that the drill depth has been reached.

SINUMERIK System 800, Cycles, UMS 4 (PG)

Ap R22: Starting point in Z (absolute)

The starting point should be selected to allow sufficient room for drilling with chip removal. The final drilling depth is calculated from the starting point.

Example: "Deep hole drilling" machining type selected via softkey

```
%98
N05  G95  G1  X0  Z20  D01  T04  F0.1  S500  M03  LF          Select drilling position
N10  R11=1  R22=10  R24=20  R25=50
     R26=-141  R27=2  R28=0  LF
     L98      P1  LF                                          Call deep drilling cycle
N15  M30  LF
```
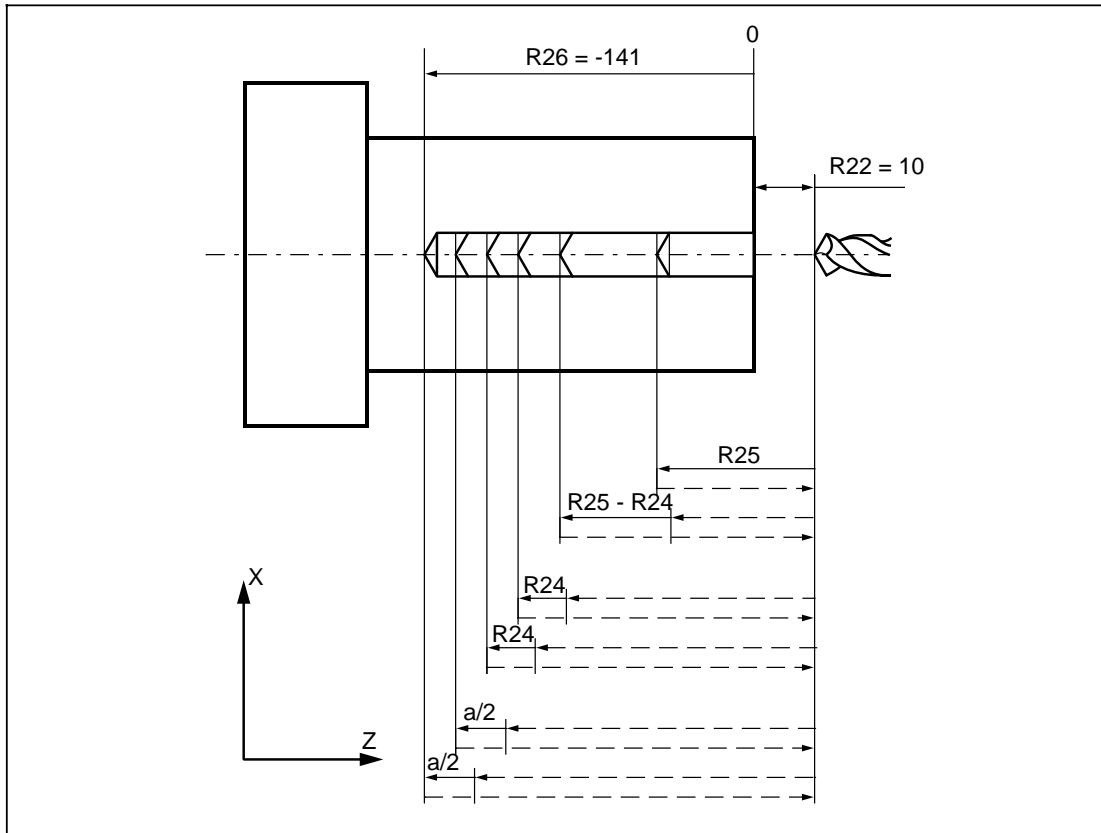

T2 R26: Final drilling depth

The drilling depth goes on reducing by a constant amount of degression until the end point R26 is reached in each case.

However, if a particular drilling depth is theoretically less than the amount of degression, it is maintained constant at this magnitude.

If the remaining infeed depth is less than twice the amount of degression, the remaining amount is halved. The two final infeeds are executed with this halved value. This ensures that the final infeed is not executed with a value that is not high enough. This calculation always results in a minimum infeed of half the degression amount.

2.2   Machining cycles for drilling and milling
(prerequisite: polar coordinate programming)

The drilling cycles, drilling patterns, milling cycles and milling patterns are available as machining cycles for drilling and milling.

L81	Drilling, centering	L900	Hole circle drilling pattern
L82	Drilling, countersinking	L905	Single hole drilling pattern
L83	Deep hole drilling	L906	Row of holes drilling pattern
L84	Tapping (with or without encoder)	L901	Slot milling pattern
L85	Boring 1	L902	Elongated hole milling pattern
L86	Boring 2	L903	Mill rectangular pocket
L87	Boring 3	L904	Circular slot milling pattern
L88	Boring 4	L930	Mill circular pocket
L89	Boring 5		
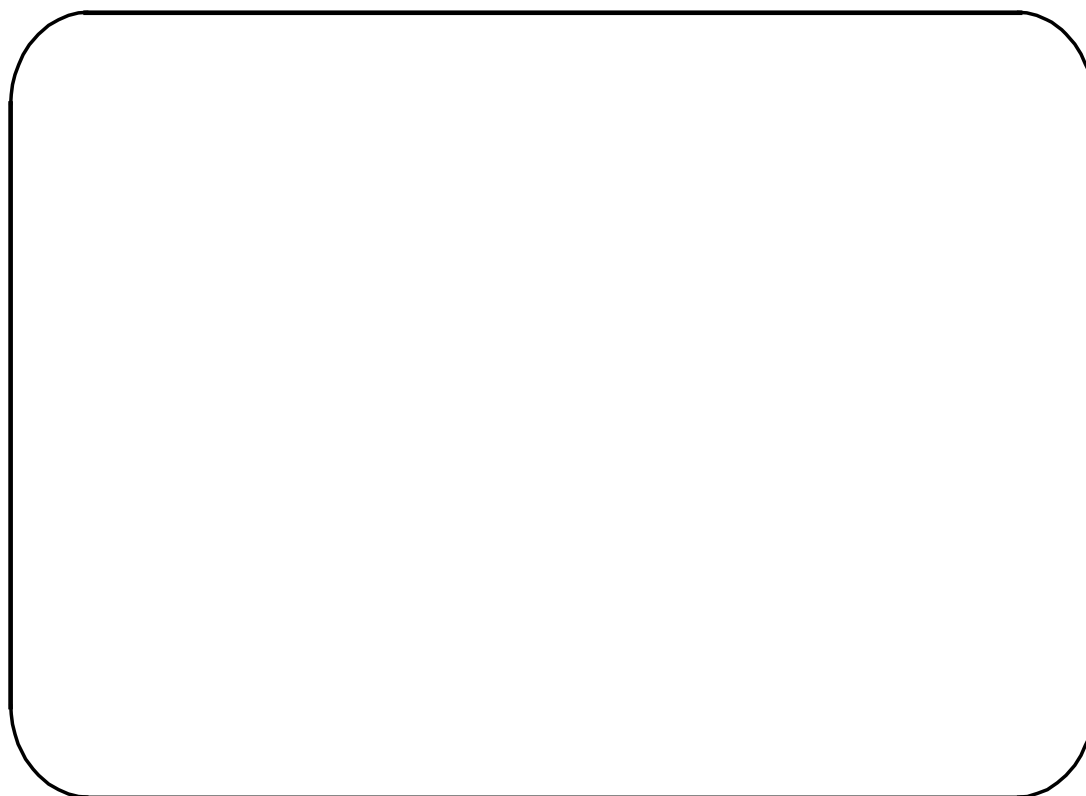
The drilling and milling cycles L900 to L930 are programmed as absolute values. The axis name, radius and angle can be selected with variable addresses by means of machine data.

The current plane must be selected via G16 or G17 to G19 before calling the cycles. The infeed axis (drilling axis) is always the axis positioned perpendicular to the current plane. This permits utilization of drilling and milling cycles in all axes.

Before calling the cycles, the length compensation must be selected. The length compensation of the tool (milling cutter, drill) is always effective perpendicular to the selected plane and remains active also after the cycle has been completed.

The appropriate feedrate, spindle speed and spindle direction of rotation must be programmed in the part program (apart from the cycles in which the values can be programmed as input parameters).

The centre point coordinates R22 and R23 are programmed in a right-handed system, for example:

G17 plane   R22=X, R23=Y, infeed axis=Z
G18 plane   R22=Z, R23=X, infeed axis=Y
G19 plane   R22=Y, R23=Z, infeed axis=X

2.2.1   Drilling cycles G81 to G89

A drilling cycle (working cycle) defines a series of machine motions for drilling, boring, tapping etc. in accordance with DIN 66025. The drilling cycles G81 to G89 are executed as subroutines L81 to L89. These subroutines are stored in the control.

The user can deviate from a standard fixed cycle and redefine it, if it meets his specific machine or workpiece requirements in a better way. The parameters R00 to R17 are used by the subroutines to define the variable values (reference planes, final depth, drilling feedrate, dwell time etc.) and their values are defined in the higher-level program.

The subroutines L81 to L89 can be called via G81 to G89 by assigning the parameters in the program. They are modal and are cancelled with G80. The selection and cancellation of G81 - G89 should be done only within one program level (compare example).

Example: Call G81 (drilling, centering)

```
%81
N8101  G90  F130  S710  M03  LF
N8102  G00  D01  Z50  T03  LF
N8103  X10  Y15  LF                              Approach 1st drilling position
N8104  G81  R2=2  R3=-15  R10=10  LF             Call L81, parameter assignment
N8105  X30 Y40  LF                               Approach 2nd drilling position
N8110  G80  Z50  LF                                         Deselect L81
N8115  M30  LF
```

The drilling hole position must be approached in the current plane by the calling program. The drilling cycle called with G81-G89 is executed in every NC block until it is cancelled with G80. It should therefore be noted that the drilling cycle becomes active even after NC blocks containing no position data.

SINUMERIK System 800, Cycles, UMS 4 (PG)

The following parameters are used in cycles L81 - L89:

Symbol	Parameter	Description
t1	R00	Dwell time at the starting point (for chip removal)
T1	R01	Enter first drilling depth (incremental) without sign
E1	R02	Reference plane (absolute)
T_	R03	Final depth of hole (absolute)
t_	R04	Dwell time at the bottom of drilling hole (chip breaking)
Db	R05	Amount of degression (incremental)
Me	R06	Direction of spindle rotation for retraction (M03/M04)
M_	R07	Direction of spindle rotation (M03/M04)
	R08	Thread drilling with and without encoder
P	R09	Thread pitch (only for thread drilling with encoder)
E2	R10	Retraction plane (absolute)
	R11	Deep hole drilling with chip breaking or removal
	R11	Number of drilling axis (not contained in L84 with SINUMERIK 850/SINUMERIK 880
sa	R12	Retraction path (horizontal with sign) (incremental)
so	R13	Retraction path (vertical with sign) (incremental)
Ft	R16	Feedrate
Fr	R17	Retraction feedrate

Subroutine L81: Drilling, centering

The following values are entered in the menu display or programmed directly as parameter assignments:

Symbol	Parameter	Description	
E1	R02	Reference plane	(absolute)
T	R03	Final depth of hole	(absolute)
E2	R10	Retraction plane	(absolute)
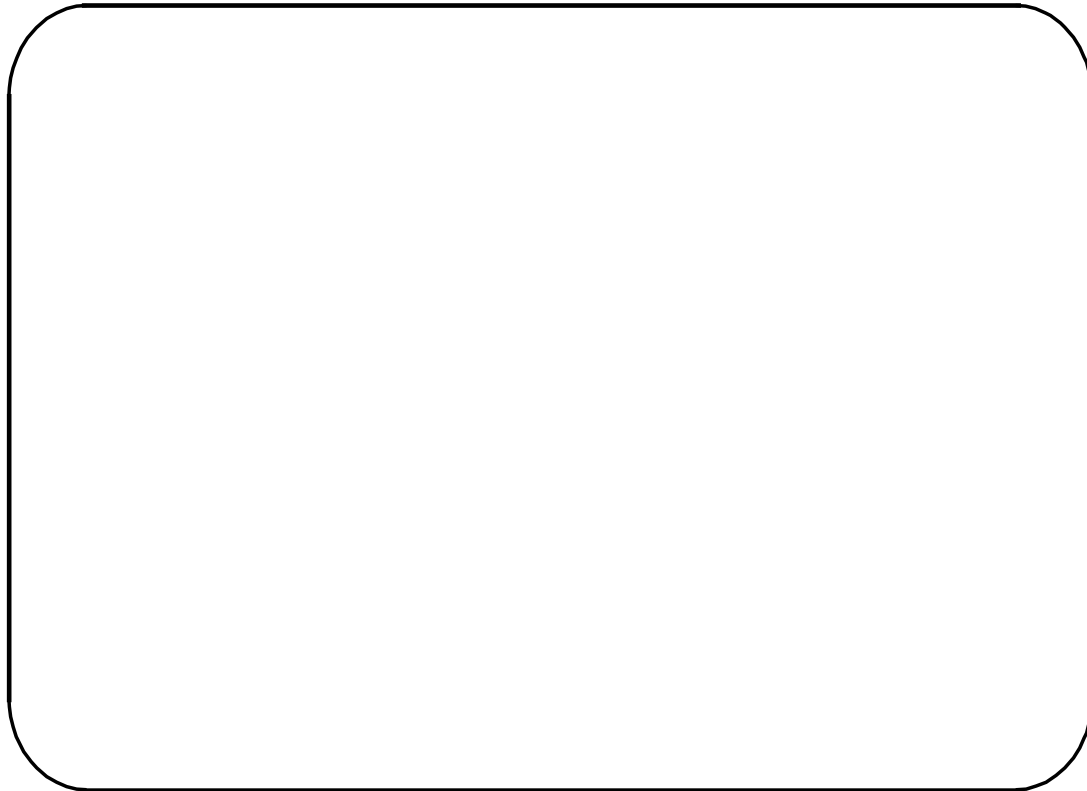
Example: "Drilling, centering" machining menu selected via softkey

```
%81
N8101  G90  F130  S710  M03  LF
N8102  G00  D01  Z50  T03  LF
N8103  X10  Y15  LF                          Select 1st drilling position
N8104  G81  R2=2  R3=-15  R10=10  LF         Call up drilling cycle, 1st hole
N8105  X25  Y60  LF                          Select 2nd drilling position and
                                             automatic call drilling cycle, 2nd hole
N8106  G80  Z50  LF                          Deselect L81
N8107  M30  LF
```
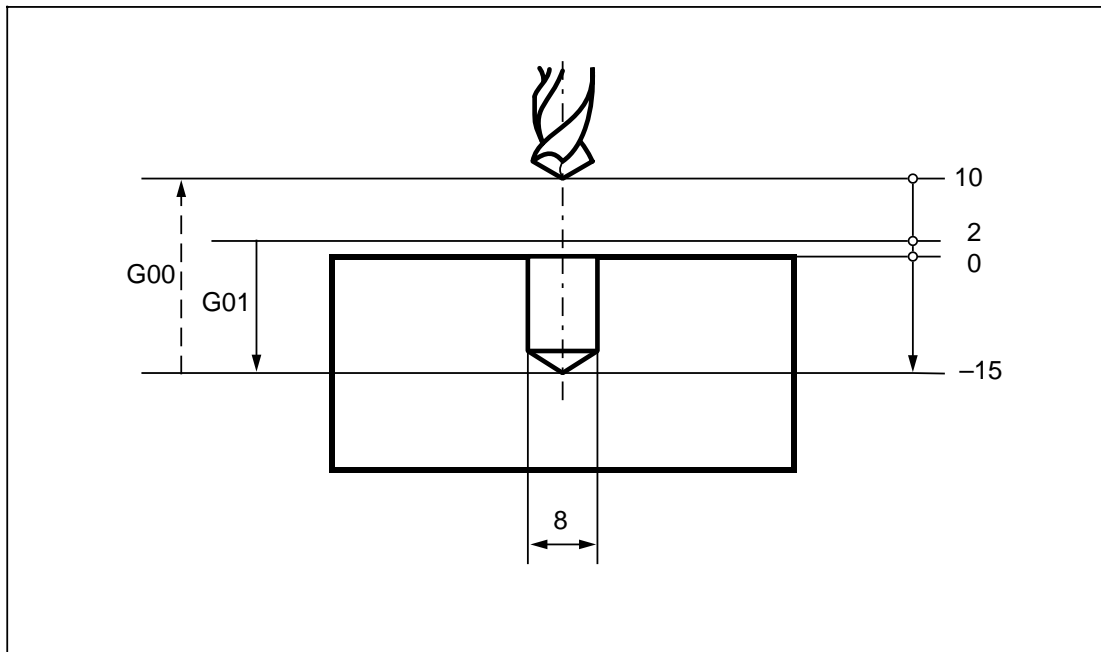
Subroutine L82: Drilling, counterboring

The following values are entered in the menu display or programmed directly as parameter
assignments:

Symbol	Parameter	Description	
E1	R02	Reference plane	(absolute)
T	R03	Final drilling depth	(absolute)
t	R04	Dwell time at bottom of hole (chip breaking)	
E2	R10	Retraction plane	(absolute)

Example: ”Drilling, counterboring” machining menu selected via softkey

```
%82
N8201  G90  F130  S710  M03  LF
N8202  G00  D01  Z50  T03  LF
N8203  X10  Y15  LF                                    Select 1st drilling position
N8204  R2=2  R3=-8  R4=1  R10=10  L82  P1  LF          Call drilling cycle, 1st hole
N8205  X25  Y60  LF                                    Select 2nd drilling position
N8206  R2=2  R3=-8  R4=1  R10=10  L82  P1  LF          Call drilling cycle, 2nd hole
N8207  Z50  LF
N8208  M30  LF
```
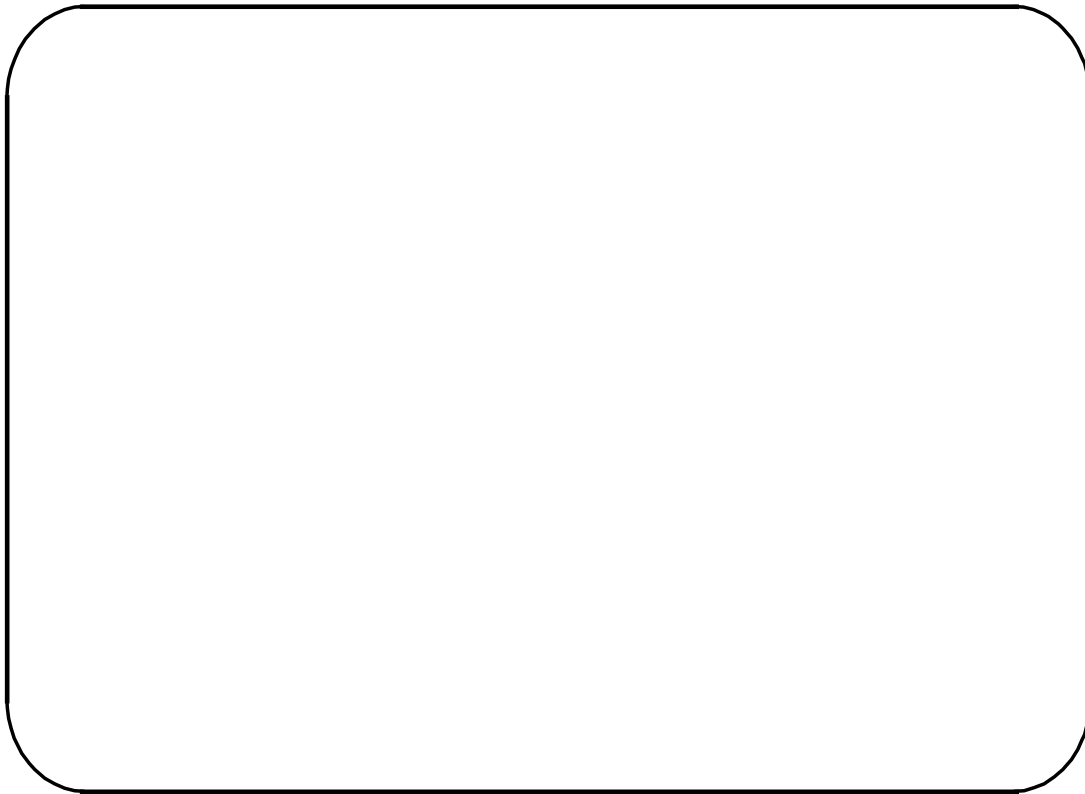
Subroutine L83: Deep hole drilling

The cycle permits deep holes to be drilled. For chip removal purposes, the drill can be moved to the reference point from each infeed depth.

The following values are entered in the menu display or programmed directly as parameter assignments:

Symbol	Parameter	Description
t1	R00	Dwell time at the starting point (for chip removal)
T1	R01	Enter first drilling depth (incremental) without sign
E1	R02	Reference plane (absolute)
T2	R03	Final drilling depth (absolute)
t2	R04	Dwell time at the bottom of hole (chip breaking)
Db	R05	Enter amount of degression without sign (incremental)
E2	R10	Retraction plane (absolute)
	R11	0=with chip breaking, 1 = with chip removal
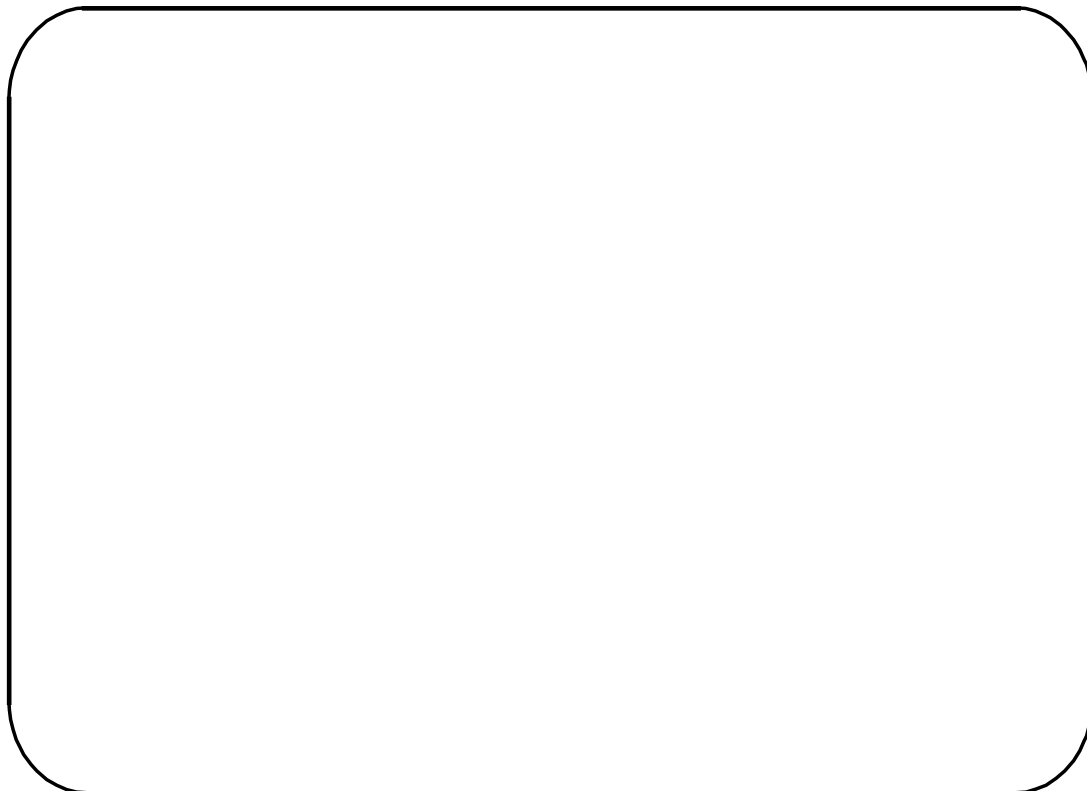
T1 R01: First drilling depth

Enter R01 as an incremental value without sign.

T2 R03: Final drilling depth (absolute)

1. The first drilling stroke is processed according to the programmed R01 (first drilling depth).

2. The second drilling stroke is processed: this is arrived at by subtracting R05 (amount of degression) from R01 (first drilling depth). If the drilling stroke calculated internally in the cycle is less than the degression amount, the remaining drilling strokes will be executed with the R05 value (amount of degression) until the remaining drilling depth is reached.

3. If the remaining drilling depth is greater than R05 (amount of degression) and less than double R05, it is divided into two drilling strokes.

    $R05 < a < 2 \cdot R05$ (a = remaining drilling depth)

    Once the drilling strokes concerned have been executed, the retraction movement depends on the programming of R11 (chip breaking/chip removal).

R11: Chip breaking, chip removal

If R11 is assigned with 0, the drill is retracted by 1 mm for chip breaking once each drilling depth has been reached.

If R11 is assigned with 1, the drill travels to the reference plane for chip removal once each drilling depth has been reached.

Example: "Deep-hole drilling" machining menu selected via softkey

```
%83
N8310  G90  F30  S500  M03  LF
N8320  G00  D01  Z50  T03  LF
N8330  X40  Y40  LF                                    Select drilling position
N8340  R0=1  R1=50  R2=4  R3=-141
       R4=1  R5=20  R10=10  R11=1  L83 P1  LF           Call drilling cycle
N8350  Z50  LF
N8360  M30  LF
```
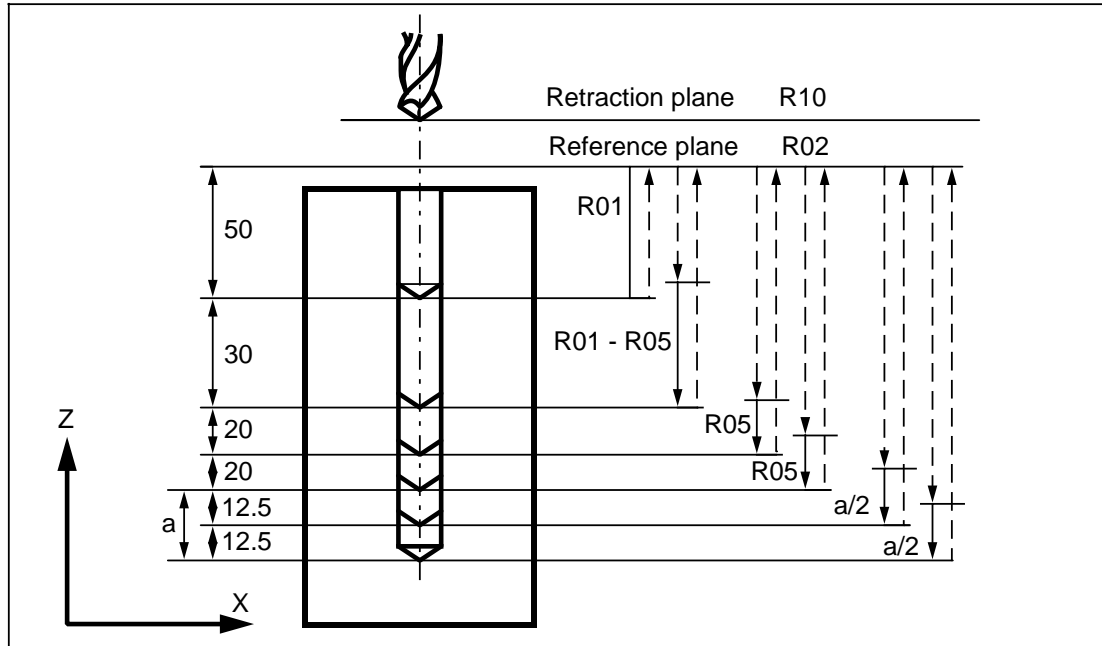
Subroutine L84: Tapping for machines with and without encoder

Cycle L84 permits tapping **with** and **without** encoder. In both cases, a compensating chuck must be used.

However, SINUMERIK 810M GA1 (from NC SW 3 onwards) and SINUMERIK 810M GA2/SINUMERIK 820 (from NC SW 2 onwards), tapping can be executed without using a compensating chuck subject to the use of the "dynamic spindle compensation" (Option) and an encoder.

Depending on machine data 5013.1, the cycle recognizes whether tapping is to take place with or without encoder.
MD 5013.1 = 1: tapping without encoder, MD 5013.1 = 0: tapping with encoder.

If cycle L84 is used on T controls, the compensating chuck used must be longer than on M controls.

Spindle override and feedrate override must be permanently set to 100 %.

The following values are entered in the menu display or programmed directly as parameter assignments:

Symbol	Parameter	Description
E1	R02	Reference plane (absolute)
T	R03	Final drilling depth (absolute)
t	R04	Dwell at thread depth
Me	R06	Direction of rotation for retraction (M03/M04)
Ma	R07	Direction of rotation after cycle (M03/M04)
	R08	Tapping 1=with encoder, 0=without encoder
P	R09	Thread pitch
E2	R10	Retraction plane (absolute)
	R11	Number of drilling axis (not included in L84 SINUMERIK 850/880)

t R04: Dwell at thread depth

The dwell time is effective only with tapping *without* encoder.

Me R06: Direction of rotation for retraction

If the direction of spindle rotation is to be reversed automatically, R06 must be set to 0. If MD 5013.1 without encoder is set, R06 *must* be programmed, otherwise error message 4120 "No spindle direction programmed" appears.

Ma R07: Direction of rotation after cycle

If the tapping cycle is called with modal function G84, the cycle requires a direction of rotation in order to be able to make further thread holes. This information is programmed in R07.

To make the first thread hole, the direction of rotation must be entered in the part program with M03 or M04 before the cycle is called. This also applies to a single call.

If R06 = 0 is programmed (automatic reversal of spindle direction of rotation), R07 must *not* be assigned.
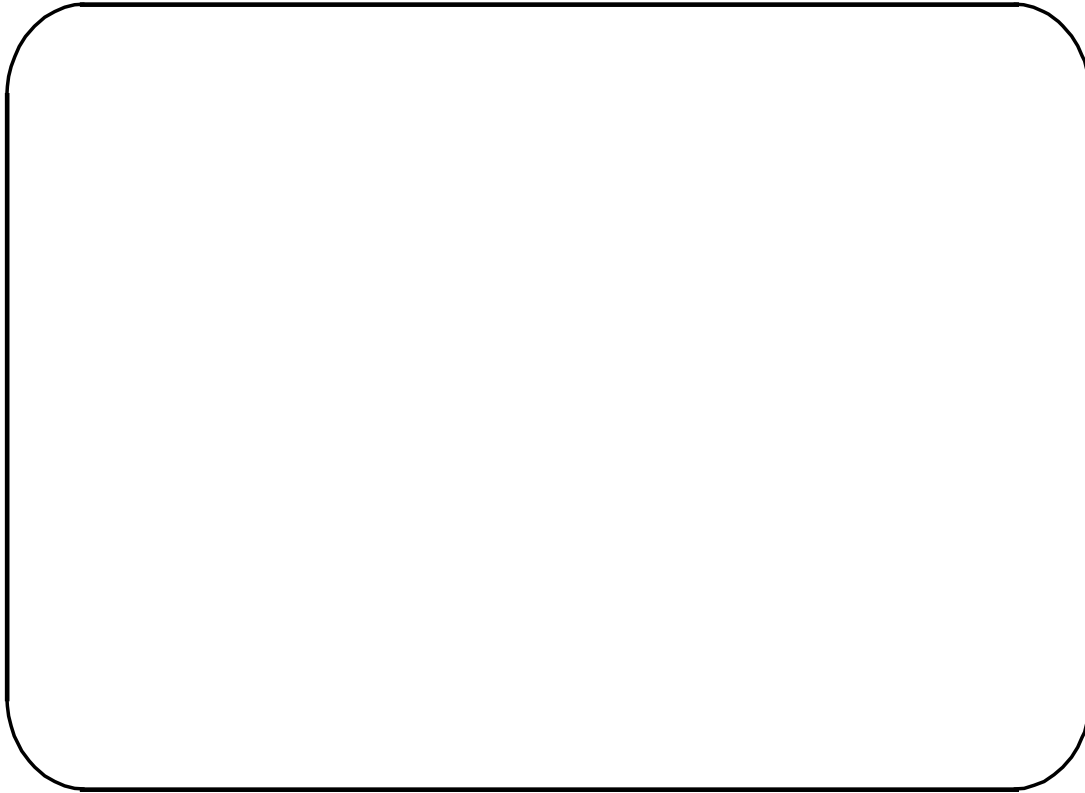
R08: Tapping with/without encoder

If tapping is to be executed without an encoder although one is available, (MD 5013.1 = 0), R08 must be assigned with 0.

If MD 5013.1 = 1 (without encoder) and R08 = 1 (with encoder) are selected, R08 is disregarded.

P R09: Pitch

Thread pitch is effective only in conjunction with tapping *with* encoder. The required feedrate value is calculated by means of the input spindle speed and thread pitch.

When tapping *without* encoder, a feedrate value must be entered in the part program.

R11: Number of drilling axis

SINUMERIK 850M/SINUMERIK 880M:

If cycle L84 is used with these control types, R11 is invalid. The drilling axis is obtained from the selected plane.

SINUMERIK 810M, GA1 und SINUMERIK 810M GA2/SINUMERIK 820:

If cycle L84 is used with these control types, R11 **can** be supplied with the number of the drilling axis or, alternatively (R11 programmed with 0), the drilling axis is obtained from the selected plane.

However, if R11 is supplied with the number of the drilling axis, no STOP DECODING is output in the cycle. L84 is processed more quickly in this case.

L84 Function extension

For SINUMERIK 850/880, cycle L843 has been extended to include the brake start point calculation function enabling the tap overtravel to be compensated for depending on spindle speed, actual gear speed and spindle acceleration time constant when tapping (G33).

The brake start point calculation function is activated with setting data 5000.7.
SD 5000.7 = 0   No calculation of overtravel compensation
SD 5000.7 = 1   Calculation of overtravel compensation

Note

Cycle L84 has a new cycle alarm: 4153 Thread length too short
This alarm occurs if the calculated overtravel compensation path is greater than the thread length.

Remedy:

*   Program lower spindle speed
*   Place reference plane point (R2) higher.
    Make sure the compensating chuck releases properly.
*   Set SD 5000 bit 7= 0 (deselect overtravel compensation)

Points to note:

*   The cycle runs with the first spindle only. The first spindle must be processed in the 1st PLC (signals from/to spindle must be set for the 1st PLC).
*   The cycle functions only if the calculated overtravel path is shorter than the thread length.
*   When tapping with a drilling pattern make sure that the spindle will not reach its fall speed immediately at cycle start but only after some time has elapsed. Otherwise the cycle would calculate an incorrect overtravel compensation
*   The spindle override must be at 100%.
*   The spindle run-up time must be set in the NC via MD and not at the drive, since otherwise calculation would not be correct.
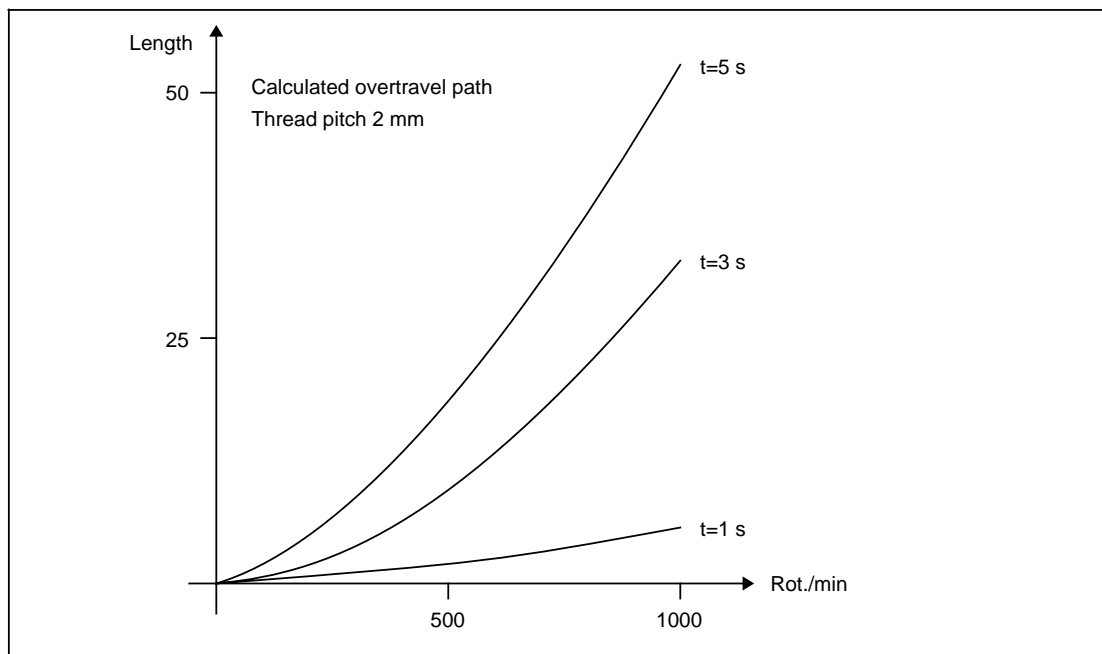
Diagram showing the minimum thread length with various speeds and run-up time constants

Example 1: "Tapping with encoder" machining menu selected via softkey

MD 5013.bit 1=0

```
%1
N05  G90  D01  T03  S500  M03  LF
N10  G0  X20  Y20  Z15  LF                                    Select drilling position
N15  R2=2  R3=-25  R4=0  R6=0
     R7=4  R8=1  R9=1.25  R10=10  LF
N20  L84  P1  LF                                              Call drilling cycle
N25  G0  X200  Y200  Z100  LF
N30  M30  LF
```

Example 2: "Tapping without encoder" machining menu selected via softkey

MD 5013.bit 1=1

```
%2
N05  G90  D01  T03  S500  M03  LF
N15  G0  X20  Y20  Z15  LF                                    Select drilling position
N20  G1  F1.25  LF                                            Feedrate value
N25  R2=2  R3=-25  R4=1  R6=4
     R7=3  R8=0  R9=0  R10=10  LF
N30  L84  P1                                                  Call drilling cycle
N35  G0  X200  Y200  Z100  LF
N40  M30  LF
```

SINUMERIK System 800, Cycles, UMS 4 (PG)
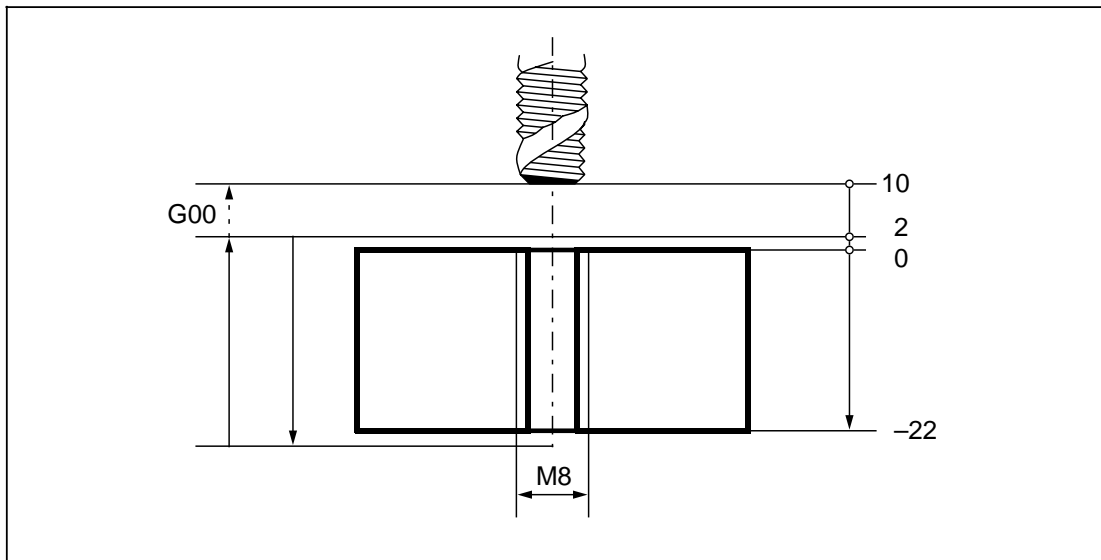
Subroutine L841: Tapping without compensating chuck (option), as from UMS 47

With SINUMERIK 880M GA2, Software Version 1 and higher, it is possible to machine tapped holes without a compensating chuck. In contrast to the previous tapping cycle, this subroutine has special features which are described below.

Therefore the selection display for tapping cycles L81 to L84 has been extended to include the selection between tapping with and without compensating chuck. If the option "Tapping without compensating chuck" has been implemented, the operator can either select the input display for the conventional cycle L84 (key "with chuck") or the input display for the new cycle L841 (key "without compensating chuck") via the softkeys. However, the cycle can also be used if a compensating chuck is available.

The following values are entered in the menu display or directly in the part program to set the parameters:

Symbol	Parameter	Description	
E1	R02	Reference plane	(absolute)
T	R03	Final drilling depth	(absolute)
t	R04	Dwell in depth	
P	R09	Pitch	
E2	R10	Retraction plane	(absolute)
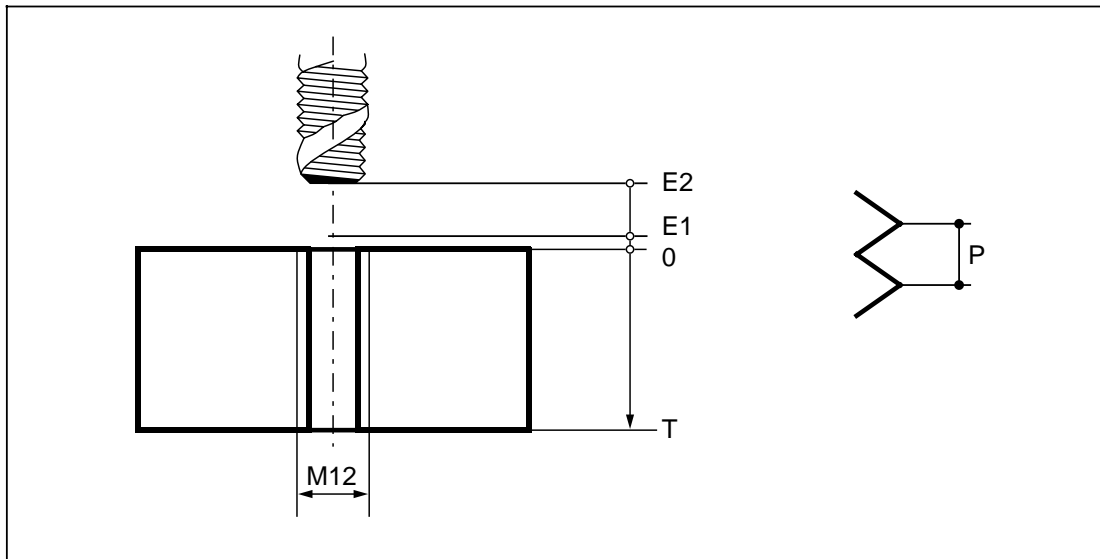
t   R04: Dwell in thread

The dwell time is always active and is programmed to match an F word.

Note: A dwell time should not be programmed if an exceptionally good dynamic response of the tapping function is required, i.e. R4=0 should be set.

P   R09: Pitch

As the cycle is based on the interrelationship between the spindle as a rotary axis and a longitudinal axis, the pitch has to be programmed in the cycle in order to calculate the position of the rotary axis which corresponds to the programmed final drilling depth of the thread as well as the required feedrate value which corresponds to the programmed speed.

The following conditions have to be fulfilled when cycle L841 is used:

If cycle L841 is activated without the option "Tapping without compensating chuck", alarm 4180 "Option not available" appears and the cycle is not executed as a programming error has occurred.

The feedrate override is of no significance as the spindle interpolates as an axis. The feedrate override can be altered but, depending on the desired thread size, should not be less than a certain minimum value in order to obtain optimum cutting conditions.

The feedrate calculated for the tapping is compared with the maximum permissible feedrate of the rotary axis and should the latter be smaller, alarm 4122 "Calculated feed too great" is output. In order to eliminate the error, either the speed programmed in the S word in the G203/G204 block must be reduced or the maximum permissible feedrate in the machine data must be checked.

As the spindle operates as a rotary axis in the cycle a special check must be made of the axis-specific machine data for rotary axes, MD560* bit 3 "Rounding for rotary axes". It must not be set as alarm 2064, "Rounding for rotary axis incorrectly programmed" will be output if the setpoint position for the rotary axis calculated by the cycle does not involve a movement to 0.5 or 1 degree.

If cycle L841 is called from drilling pattern L900, please note setting data bit SD 5000.6 with UMS 48 and higher (see L900 description).

Programming

Cycle L841 is a part of the function "Tapping without compensating chuck" and is programmed as follows:
First the drilling position is selected in the part program. Then the spindle is switched to rotary axis mode with G203 (direction of rotation clockwise) or G204 (direction of rotation counter-clockwise). For this a separate program block must be written in the following format:

G203 or G204 [name of rotary axis] [angle] [name of infeed axis] S [speed]

Switchover with G203/G204 can only be executed from initial setting (G205).

Then the cycle is parameterized and is called with L84 or G84. If the control recognizes the call with L84 or G84 and G203/G204 is active the cycle for which the program number in NC machine data 274 is set (standard value 841) is activated. If this machine data is set to zero, alarm 3008, "Subroutine error" is output.
If additional holes are to be drilled a new position must be programmed and the cycle must be called again if necessary.

After the last hole has been machined the function "Tapping without compensating chuck" is deselected with G205. If G84 has been programmed G80 must be programmed before G205.

Once the cycle has been completed the feedrate F must be reprogrammed. G functions G00, G60 and G90 are still active when cycle L841 has been completed.

Example 1: Single call of function "Tapping without compensating chuck" with L84

```
%86
N8601  G1  F1000   LF                    Progr. feedrate before tapping is selected
N8602  G00   LF                                    Select tapping position
N8603  G203  C180  S100  LF                      Switch over to tapping without
                                          compensating chuck (direction of rotation
                                            clockwise, starting angle 180 degrees,
                                                setpoint speed 100 rev/min)


N8604  R2=5  R3=-30  R4=0  LF                         Parameterize cycle
N8605  R9=1.25  R10=10    LF
N8606  L84  LF                                            Call cycle
N8607  G205  LF                                       Deselect function
N8608  G1  F1000      LF                              Program feedrate
N8609  ...
```

Example 2: Function called 5 times with G84 with the same input parameters

```
%90
N9010  G0  LF                                  Select first tapping position
N9020  G204  C0  Z  S300  LF                     Switch over to tapping without
                                                      compensating chuck
N9030  R2=5  R3=-30  R4=0  LF                         Parameterize cycle
N9040  R9=1.25  R10=10   LF
N9050  G84  LF                                            Call cycle
N9060  G91  X10  LF                            Select 2nd tapping position
N9070  G91 X10   LF                            Select 3rd tapping position
N9080  G91 X15  LF                             Select 4th tapping position
N9090  G91 X20  LF                             Select 5th tapping position
N9100  G80  LF
N9110  G205  LF                                       Deselect function
N9120  ...
```

Subroutine L85: Boring 1

The following values are entered in the menu display or programmed directly as parameter assignments:

Symbol	Parameter	Description
E1	R02	Reference plane (absolute)
T	R03	Final drilling depth (absolute)
t	R04	Dwell time at bottom of hole (chip breaking)
E2	R10	Retraction plane (absolute)
Ft	R16	Feedrate
Fr	R17	Retraction feedrate

With setting data 5000, bit 0 = 0, a mode compatible with UMS 2 is possible, i.e. programs generated with UMS 2 can be run.

If programs created with UMS 3/60 are used, parameters R04, R16 and R17 must be added subsequently, and setting data 5000 bit 0 must be set to 1.

SINUMERIK System 800, Cycles, UMS 4 (PG)

Example: "Boring 1" machining menu selected via softkey

```
%85
N8501  G90  S150  M03  LF
N8502  G00  D01  Z50  T03  LF
N8503  X40  Y40  LF                          Select drilling position
N8504  R2=2  R3=-25  R4=0  R10=10  LF
N8505  R16=60  R17=1000  L85  P1  LF          Call drilling cycle
N8506  G00  Z50  LF
N8507  M30  LF
```
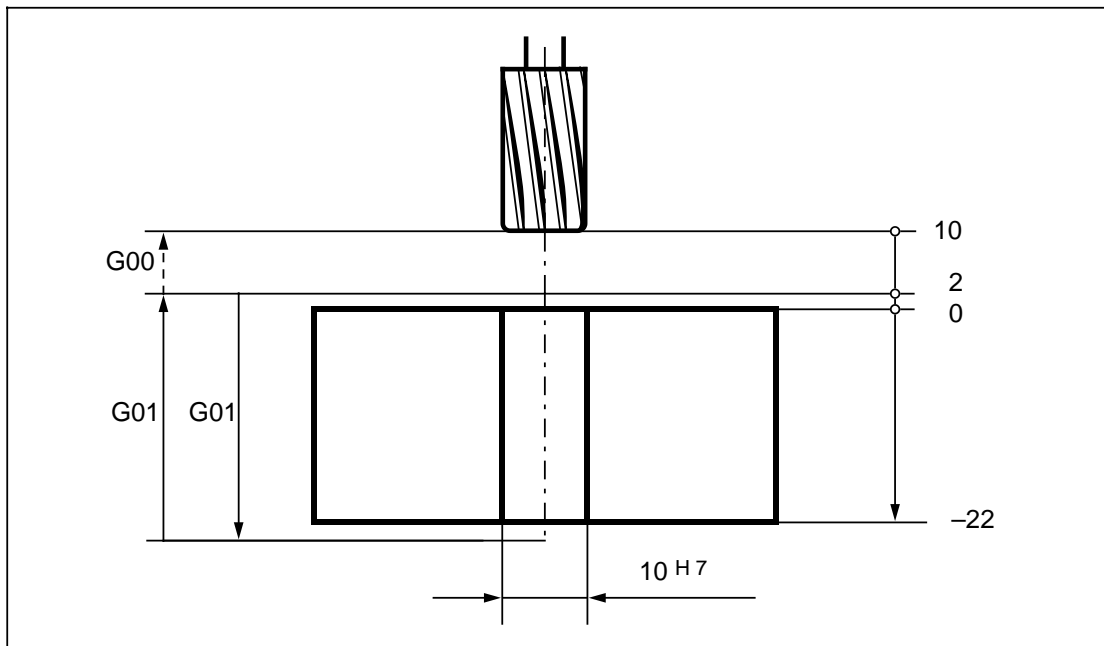

With L85, the inward and outward movements are executed with the feedrate programmed in R parameters R16 and R17.

SINUMERIK System 800, Cycles, UMS 4 (PG)

Subroutine L86: Boring 2

The following values are entered in the menu display or programmed directly as parameter assignments:

Symbol	Parameter	Description	
E1	R02	Reference plane	(absolute)
T	R03	Final drilling depth	(absolute)
t	R04	Dwell time at the bottom of hole (chip breaking)	
M	R07	Direction of spindle rotation (M03/M04)	
E2	R10	Retraction plane	(absolute)
sa	R12	Retraction path (horizontal with sign)	(incremental)
so	R13	Retraction path (vertical with sign)	(incremental)

With setting data 5000, bit 0 = 0, a mode compatible with UMS 2 is possible, i.e. programs generated with UMS 2 can be run.

If programs created with UMS 3/60 are used, parameters R04, R16 and R17 must be added subsequently, and setting data 5000 bit 0 must be set to 1.

Example: "Boring 2" machining menu selected via softkey

```
%86
N8601  G90  F100  S500  LF
N8602  G00  D01  Z50  T03  LF
N8603  X40  Y40  LF                              Select 1st drilling position
N8604  R2=2  R3=-22  R4=1  R7=3
       R10=10  R12=-2  R13=2  L86  P1  LF        Call drilling cycle, 1st hole
N8605  Z50  LF
N8606  M30  LF
```


If setting data 5000 bit 0 is set to 1, once the final drilling depth has been reached, an M19 oriented spindle stop occurs. After this, rapid traverse to the programmed retraction positions R12 and R13 is executed as far as the retraction plane.

With M19, an oriented stop can be executed with the main spindle. The corresponding angle value is programmed and entered via the operator's panel under "Setting data spindle".

If setting data 5000 bit 0 is set to 0, a non-oriented M05 spindle stop follows at the final drilling depth.

SINUMERIK System 800, Cycles, UMS 4 (PG)

Subroutine L87: Boring 3

The following values are entered in the menu display or programmed directly as parameter assignments:

Symbol	Parameter	Description	
E1	R02	Reference plane	(absolute)
T	R03	Final drilling depth	(absolute)
M	R07	Direction of spindle rotation (M03/M04)	
E2	R10	Retraction plane	(absolute)
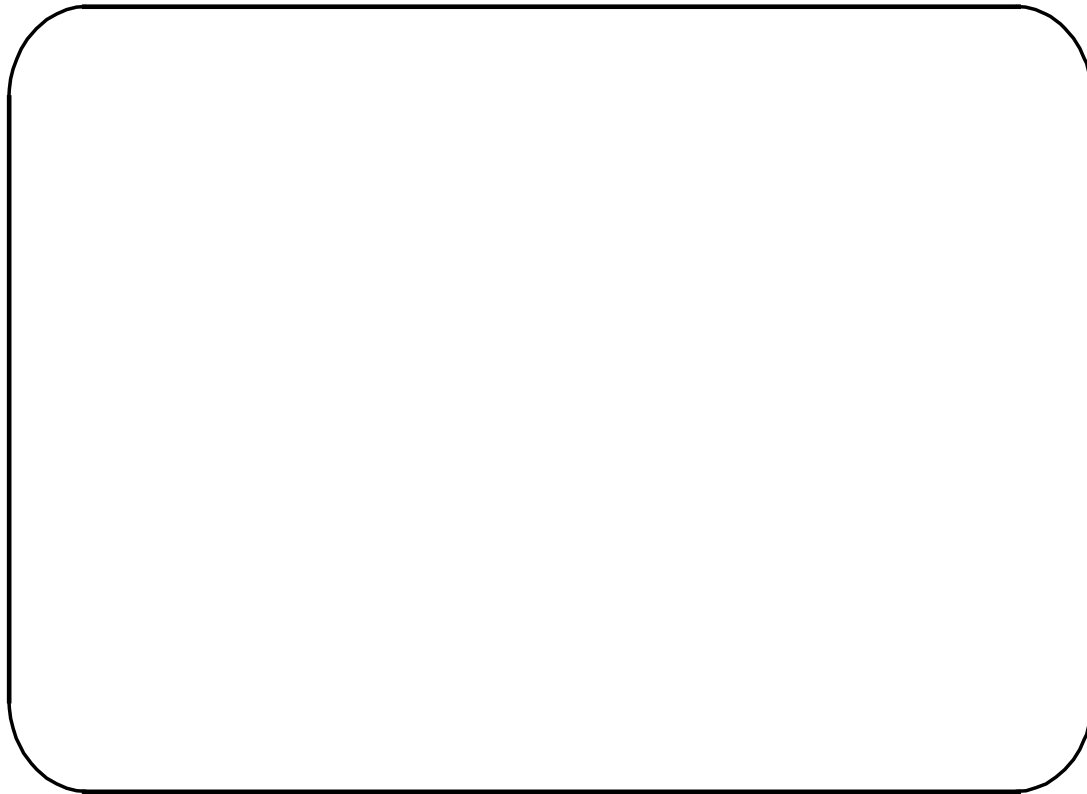
Example: "Boring 3" machining menu selected via softkey

```
%87
N8701  G90  F100  S500  LF
N8702  G00  D01  Z50  T03  LF
N8703  X40  Y40  LF                          Select 1st drilling position
N8704  R2=2  R3=-24  R4=1  R7=3
       R10=10  L87  P1  LF                   Call drilling cycle, 1st hole
N8705  X80  Y70  LF                          Select 2nd drilling position
N8706  R2=2  R3=-24  R7=3
       R10=10  L87  P1  LF                   Call drilling cycle, 2nd hole
N8707  Z50  LF
N8708  M30  LF
```
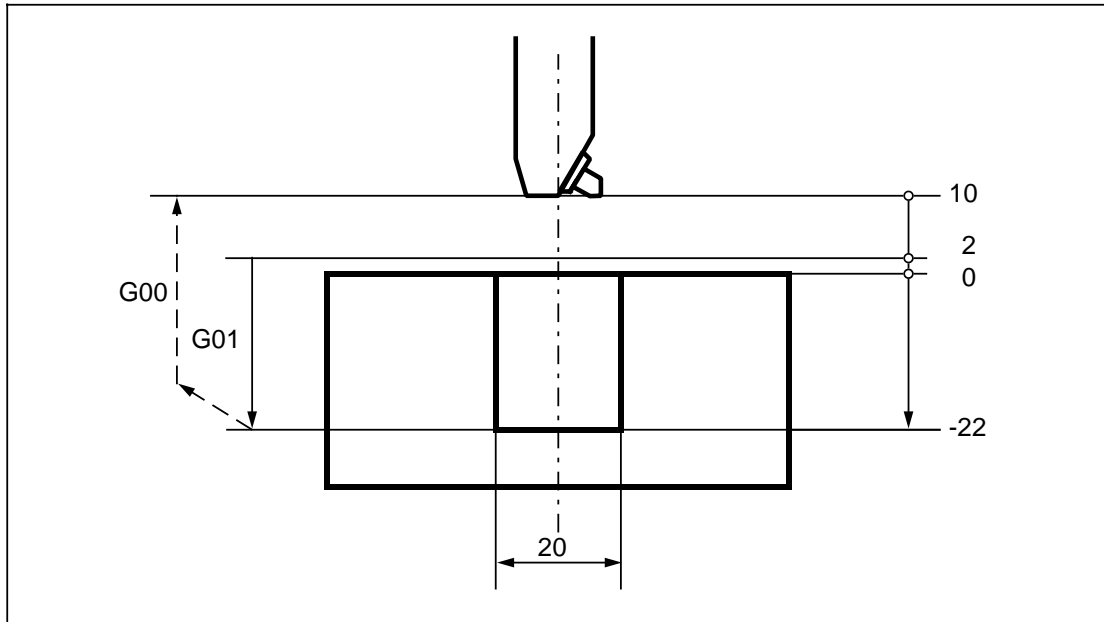

Once the final drilling depth has been reached, an M05 non-oriented spindle stop and an M00 progam stop are executed. By pressing the NC start key, the outward movement is continued in rapid traverse as far as the retraction plane.

SINUMERIK System 800, Cycles, UMS 4 (PG)

Subroutine L88: Boring 4

The following values are entered in the menu display or programmed directly as parameter assignments:

Symbol	Parameter	Description	
E1	R02	Reference plane	(absolute)
T	R03	Final drilling depth	(absolute)
t	R04	Dwell time at bottom of hole (chip breaking)	
M	R07	Direction of spindle rotation (M03/M04)	
E2	R10	Retraction plane	(absolute)

Example: "Boring 4" machining menu selected via softkey

```
%88
N8801  G90  F100  S500  LF
N8802  G00  D01  Z50  T03  LF
N8803  X40  Y40  LF                          Select 1st drilling position
N8804  R2=2  R3=-18  R4=1  R7=3
       R10=10  L88  P1  LF                    Call drilling cycle, 1st hole
N8805  X80  Y70  LF                          Select 2nd drilling position
N8806  R2=2  R3=-18  R4=1  R7=3
       R10=10  L88  P1  LF                    Call drilling cycle, 2nd hole
N8807  Z50  LF
N8808  M30  LF
```
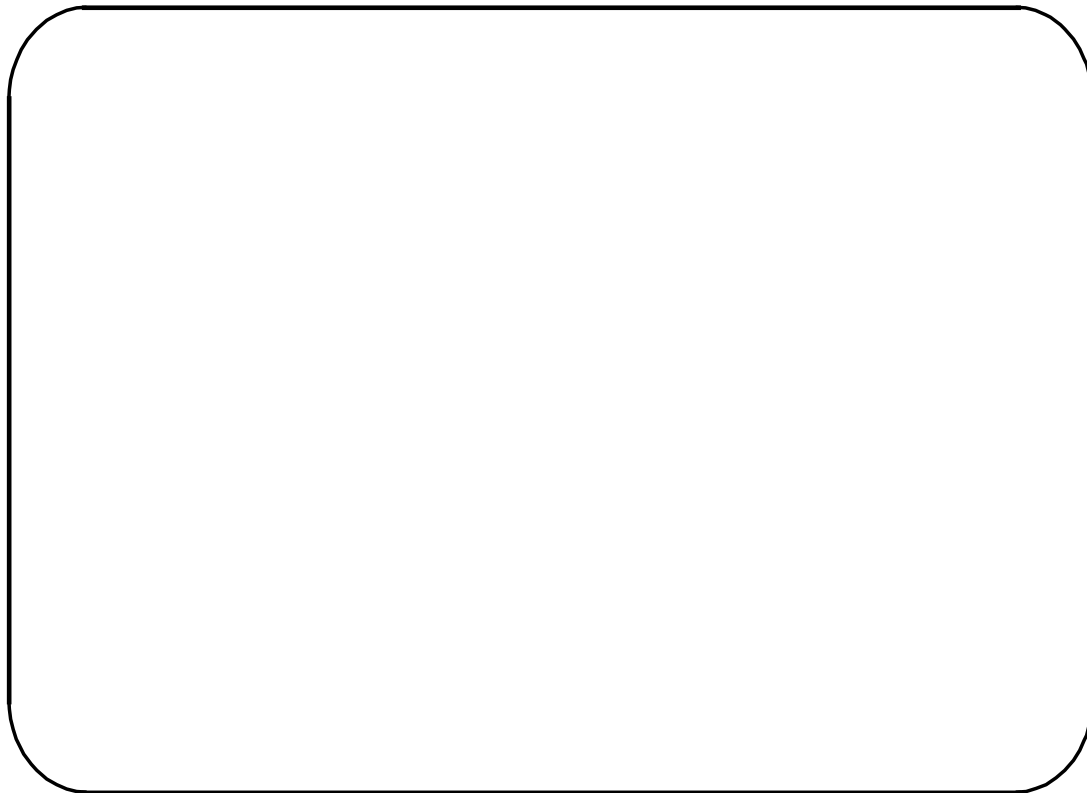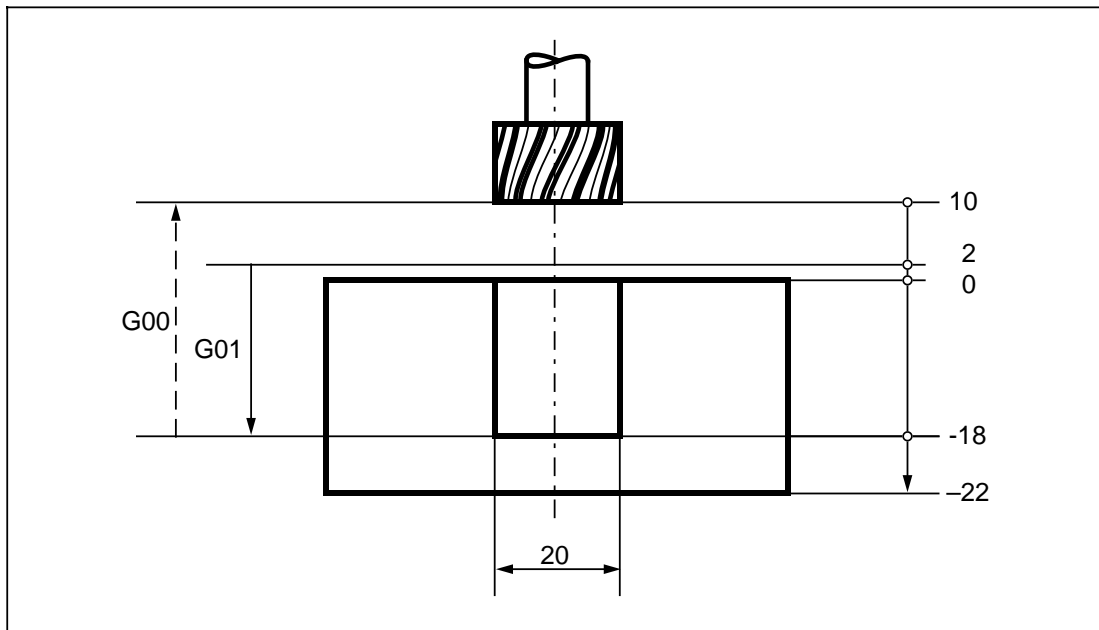

Once the final drilling depth has been reached, an M05 non-oriented spindle stop and an M00 program stop are executed. By pressing the NC start key, the outward movement is continued in rapid traverse as far as the retraction plane. A dwell time can be programmed at the final drilling depth.

SINUMERIK System 800, Cycles, UMS 4 (PG)

Subroutine L89: Boring 5

The following values are entered in the menu display or programmed directly as parameter
assignments:

Symbol	Parameter	Description	
E1	R02	Reference plane	(absolute)
T	R03	Final drilling depth	(absolute)
t	R04	Dwell time at bottom of hole (chip breaking)	
E2	R10	Retraction plane	(absolute)
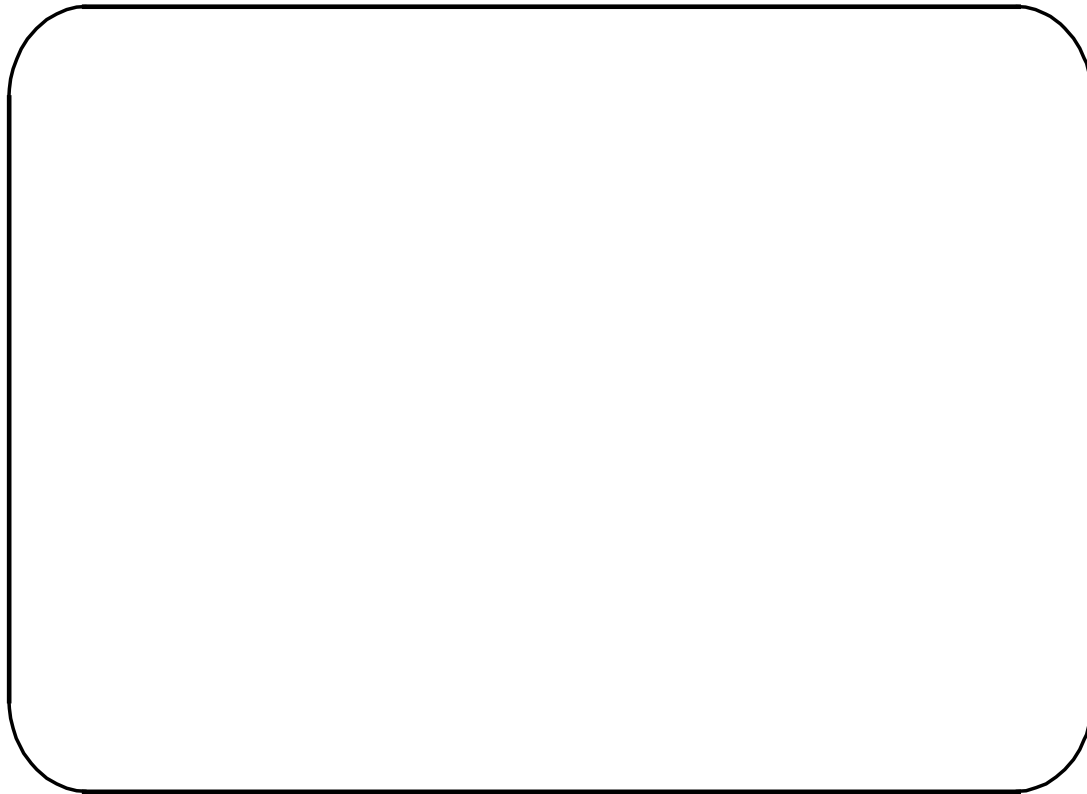
Example: "Boring 5" machining menu selected via softkey

```
%89
N8901  G90  F60  S500  M03  LF
N8902  G00  D01  Z50  T03  LF
N8903  X40  Y40  LF                                    Select 1st drilling position
N8904  R2=2  R3=-18  R4=1  R10=10  L89  P1  LF         Call drilling cycle, 1st hole
N8905  X80  Y70  LF                                    Select 2nd drilling position
N8906  R2=2  R3=-18  R4=1  R10=10  L89  P1  LF         Call drilling cycle, 2nd hole
N8907  Z50  LF
N8908  M30  LF
```
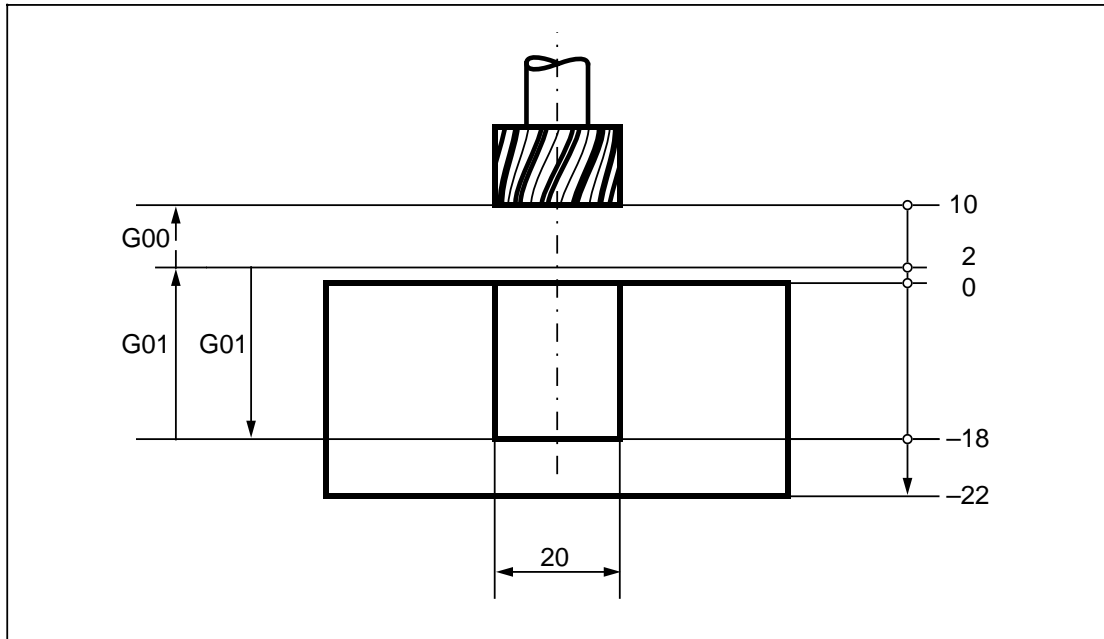
With L89, both the inward and outward movements are executed at the same feedrate. Once the final drilling depth has been reached, a dwell time can be programmed.

2.2.2 Drilling and milling patterns

The following parameters are used in cycles L900 to L930:

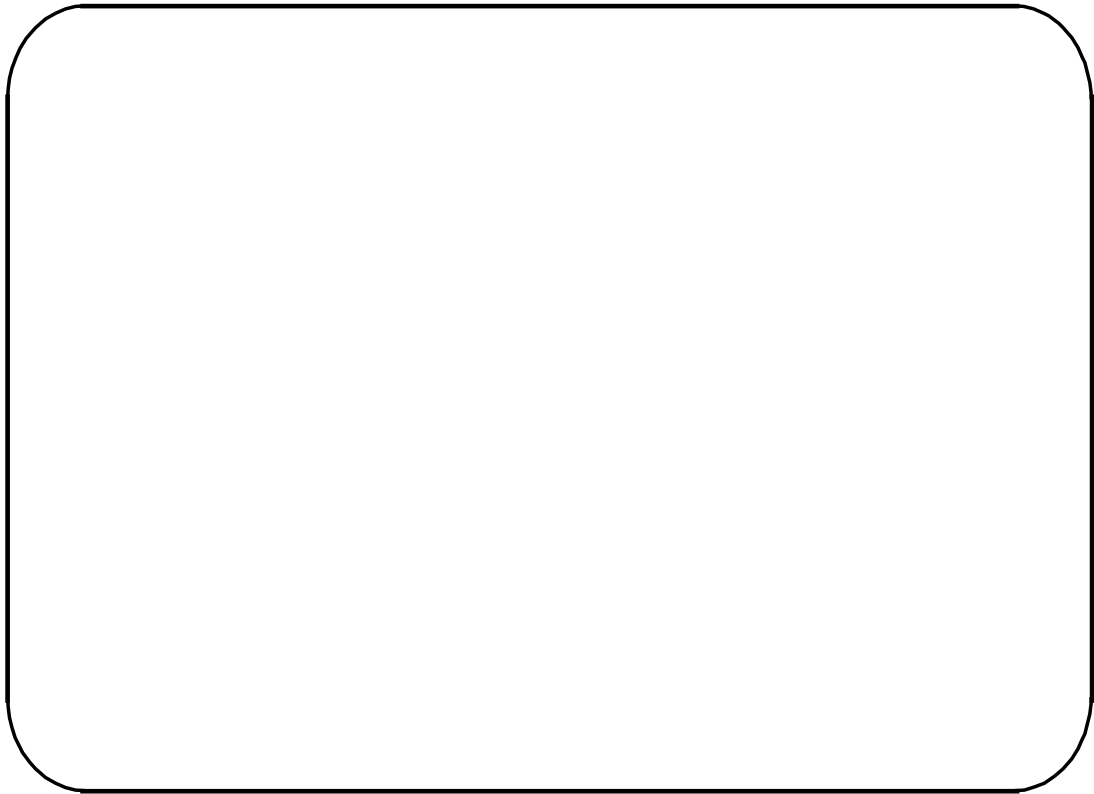| Symbol	Parameter	Description	
Zt	R01	Infeed depth	(incremental)
E1	R02	Reference plane	(absolute)
T	R03	Depth (slot, pocket, elongated hole; circular slot)	(absolute)
G	R06	Milling direction (G02/G03)	
E2	R10	Retraction plane	(absolute)
L	R12	Pocket length	(incremental)
B	R12	Slot width	(incremental)
B	R13	Pocket width	(incremental)
L	R13	Length (slot, elongated hole; angle for slot length)	(incremental)
Ff	R15	Feedrate (pocket surface)	
Ft	R16	Feedrate (pocket depth)	
Mw	R22	Centre point ... (horizontal)	(absolute)
Ms	R23	Centre point ... (vertical)	(absolute)
R	R24	Radius (corner, pocket)	
Wa	R25	Starting angle	
Wf	R26	Indexing angle	
	R27	Number of slots; holes; elongated holes	
	* R28	Number of drilling cycle (L81 to L89)	

_____

*   *With SINUMERIK 850/880, the number of the drilling cycle is entered in the part program as soon as the
    "Store drilling pattern" key is pressed. R28 is not available in the drilling patterns (L900, L905, L906).*

SINUMERIK System 800, Cycles, UMS 4 (PG)

2.2.2.1 L900 Drilling patterns

By using drilling cycles L81 to L89, L900 allows hole circles to be generated. During programming, either the "Drilling pattern" menu is selected and the R parameters are entered in the menu displays or the parameters are programmed directly as parameter assignments in the part program: subroutine L900 is active in the current plane.

If L84 (with encoder) is used, a feedrate value must be programmed in the part program before calling L900.

If L900 is used together with "Tapping without compensating chuck" on the SINUMERIK 880 GA2, please observe the following:

In cycle L900 an additional safety clearance of 1 mm in the drilling axis is programmed for traversing outside the drilling cycle. With UMS 48 and higher it is possible to suppress this by setting setting data SD 5000.6.

If tapping without compensating chuck is called in L 900 as the drilling cycle (L841), this bit must be set as otherwise alarm 3902 is output in cycle L 900.

Symbol	Parameter	Description
Mw	R22	Centre point of drilling pattern (horizontal) (absolute)
Ms	R23	Centre point of drilling pattern (vertical) (absolute)
R	R24	Radius
Wa	R25	Starting angle (referred to the horizontal axis)
Wf	R26	Indexing angle
	R27	Number of holes
	R28	Number of drilling cycle required (L81 to L89)

R28: Number of drilling cycle required (L81 to L89)

The parameters necessary for the desired drilling cycle must be defined in the part program (compare example N15).

Wf R26: Indexing angle

If 0 is stated as the indexing angle, the number of holes is divided into 360°.

Example: "DRILLING PATTERNS" machining menu selected by softkey (X/Y plane, drilling axis Z)

```
%900
N05  G90  G0  X100  Y100  Z20 D05  T04  LF
N10  G1  F130  S710  M03  LF
N15  R2=4  R3=-8  R10=10                          Drilling cycle L81 supplied
     R22=55  R23=55  R24=45  R25=0
     R26=45  R27=8  R28=81  LF
     L900  P1   LF                                Call hole circle
N20  Z50  LF
N25  M30  LF
```
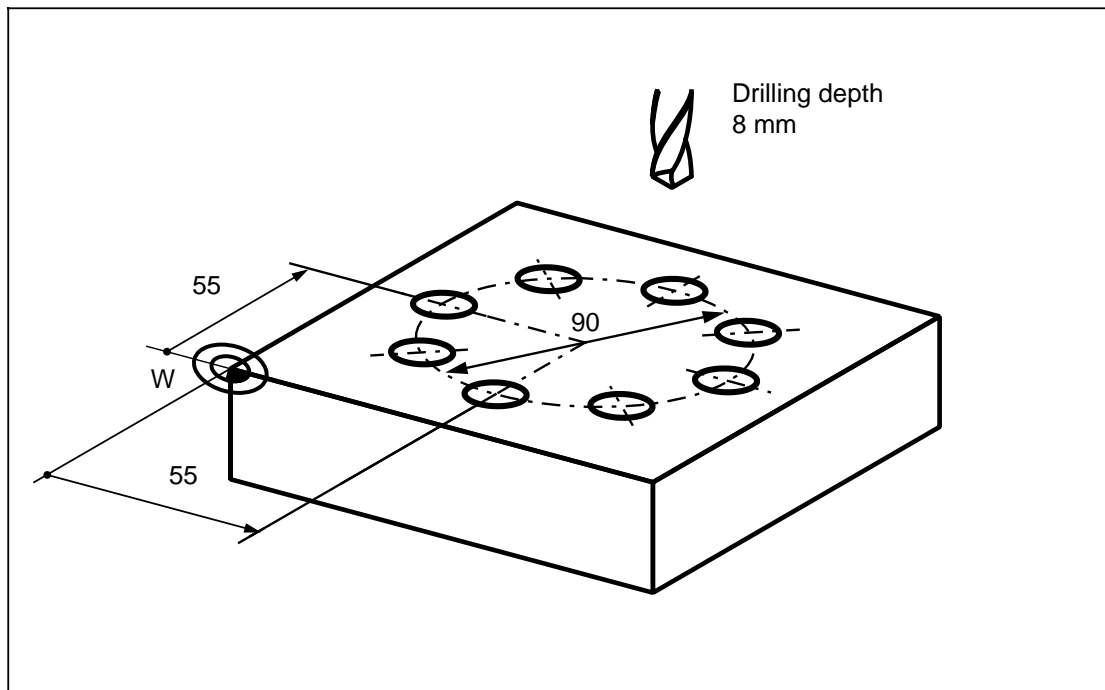
The parameters are assigned in two menu displays.
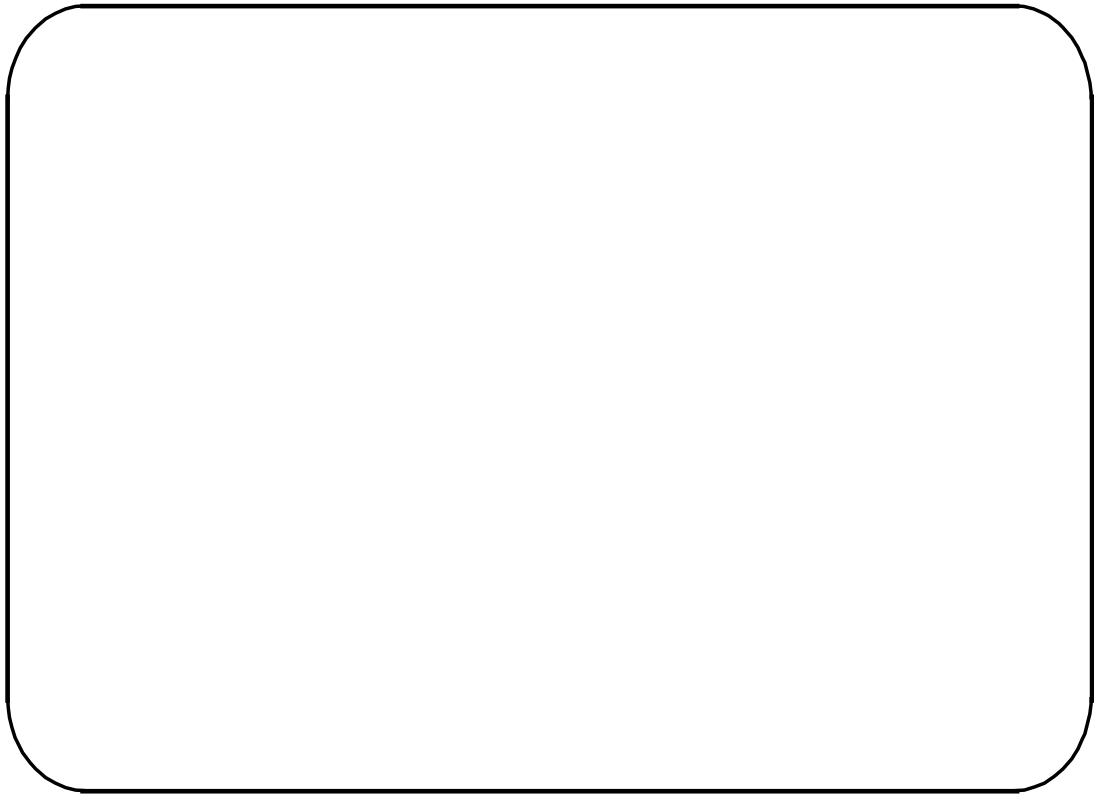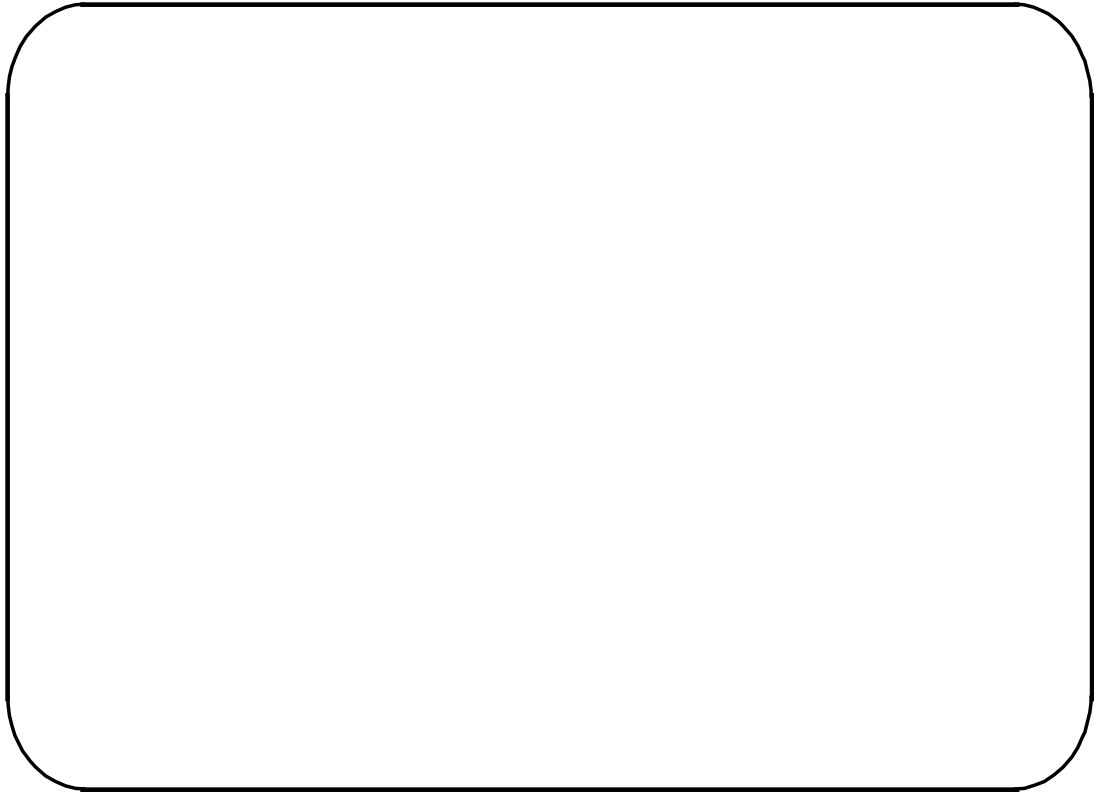
2.2.2.2 L901 "SLOT" milling pattern

During programming, either the "SLOT" menu is selected and the R parameters are entered in the menu displays or the parameters are programmed directly as parameter assignments in the part program: subroutine L901 is active in the current plane.

Symbol	Parameter	Description	
Zt	R01	Enter infeed depth without sign	(incremental)
E1	R02	Reference plane	(absolute)
T	R03	Slot depth	(absolute)
B	R12	Slot width	(incremental)
L	R13	Slot length	(incremental)
Ff	R15	Feedrate (pocket surface)	
Ft	R16	Feedrate (pocket depth)	
Mw	R22	Centre point of milling pattern (horizontal)	(absolute)
Ms	R23	Centre point of milling pattern (vertical) (referred to workpiece zero)	(absolute)
R	R24	Radius	
Wa	R25	Starting angle (referred to horizontal axis)	
Wf	R26	Indexing angle	
	R27	Number of slots	

Cycle L901 automatically selects and deselects the cutter radius compensation. The direction of milling is executed with G03.

Zt R01: Infeed depth (incremental)

If the infeed depth is assigned with R1 = 0, the infeed is executed immediately to pocket
depth at the feedrate. If the pocket cannot be milled with a single infeed, an infeed depth must
be entered. The milling process is repeated until the pocket depth is reached. If a residual
infeed depth of less than double R01 results, this is divided into two equal values. Enter the
infeed depth as an incremental value without sign.

R R24: Radius

For the radius value, enter the distance from the centre point to the slot edge.

Wf R26: Indexing angle

If 0 is stated as the indexing angle, the number of slots is divided into 360°.

B R12: Slot width

The milling cutter diameter must be smaller than 0.9 times the slot width. Failure to comply
with this results in error message 4102 (cutter radius too great). Similarly, the milling cutter
radius must not be less than half the slot width.

Example: **"SLOT" milling pattern machining menu selected via softkey**
(X/Y plane, infeed axis Z)

```
%901
N05  G90  G0  X50  Y30  Z20  D01  T01  S600  M03  LF        Select milling position
N10  R1=2.5  R2=2  R3=-5  R12=10  LF
     R13=15  R15=300  R16=100  R22=50  LF
     R23=30  R24=40  R25=45  R26=0  LF
     R27=4  L901  P1  LF                                     Call SLOT
N15  Z50  LF
N20  M30  LF
```

The parameters are assigned in two menu displays.
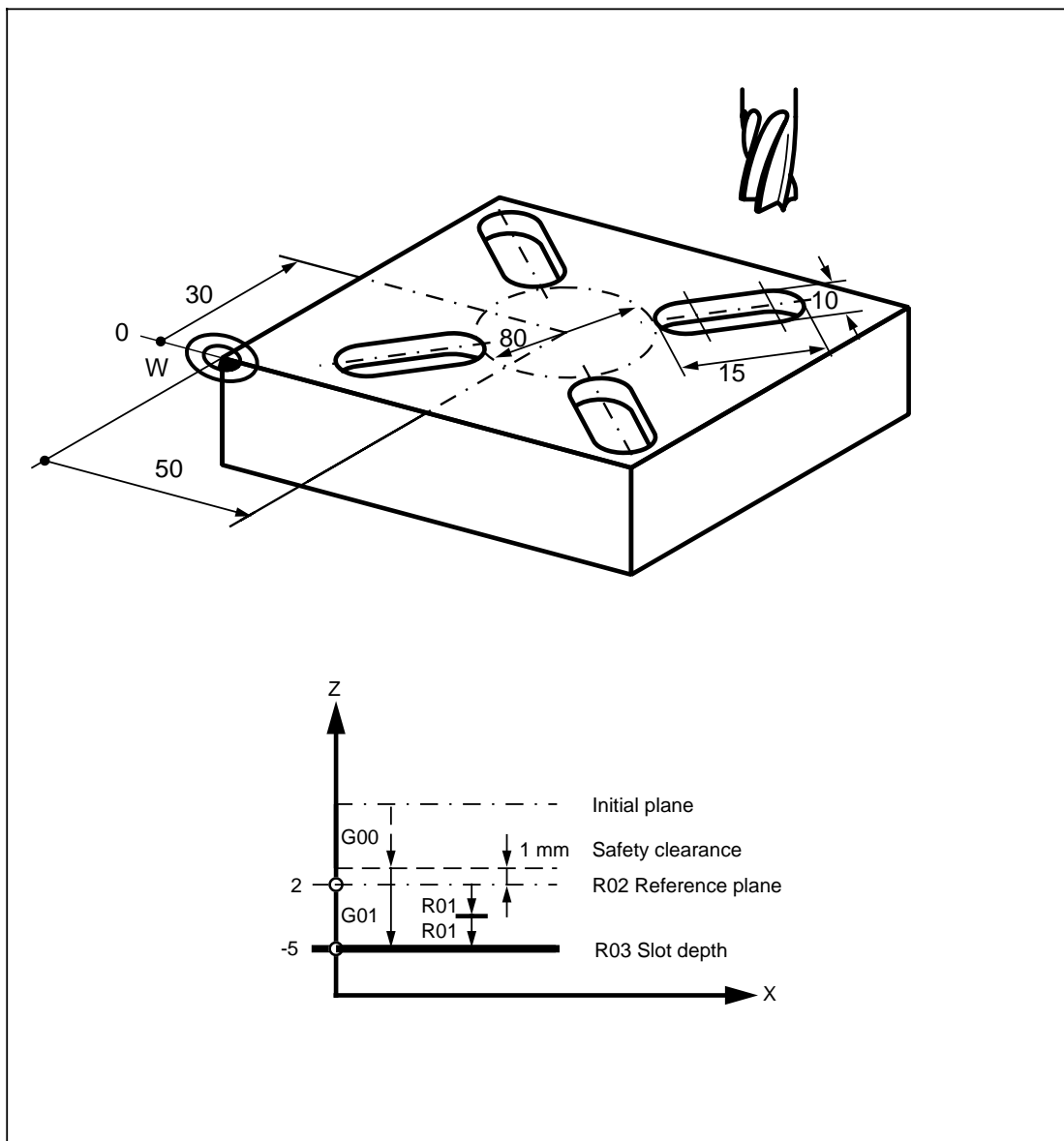
2.2.2.3 L902 "ELONGATED HOLE" milling pattern

During programming, either the "ELONGATED HOLE" menu is selected and the R parameters are entered in the menu displays, or these values are directly programmed as parameter assignments in the part program. Subroutine L902 is active in the current plane.

Symbol	Parameter	Description	
Zt	R01	Enter infeed depth without sign	(incremental)
E1	R02	Reference plane	(absolute)
T	R03	Elongated hole depth	(absolute)
L	R13	Elongated hole length	(incremental)
Ff	R15	Feedrate (pocket surface)	
Ft	R16	Feedrate (pocket depth)	
Mw	R22	Centre point of milling pattern (horizontal)	(absolute)
Ms	R23	Centre point of milling pattern (vertical) (referred to workpiece zero)	(absolute)
R	R24	Radius	
Wa	R25	Starting angle (referred to horizontal axis)	
Wf	R26	Indexing angle	
	R27	Number of elongated holes	

The cycle operates without cutter radius compensation (G41, G42). The enlongated hole width depends on the selected tool diameter.
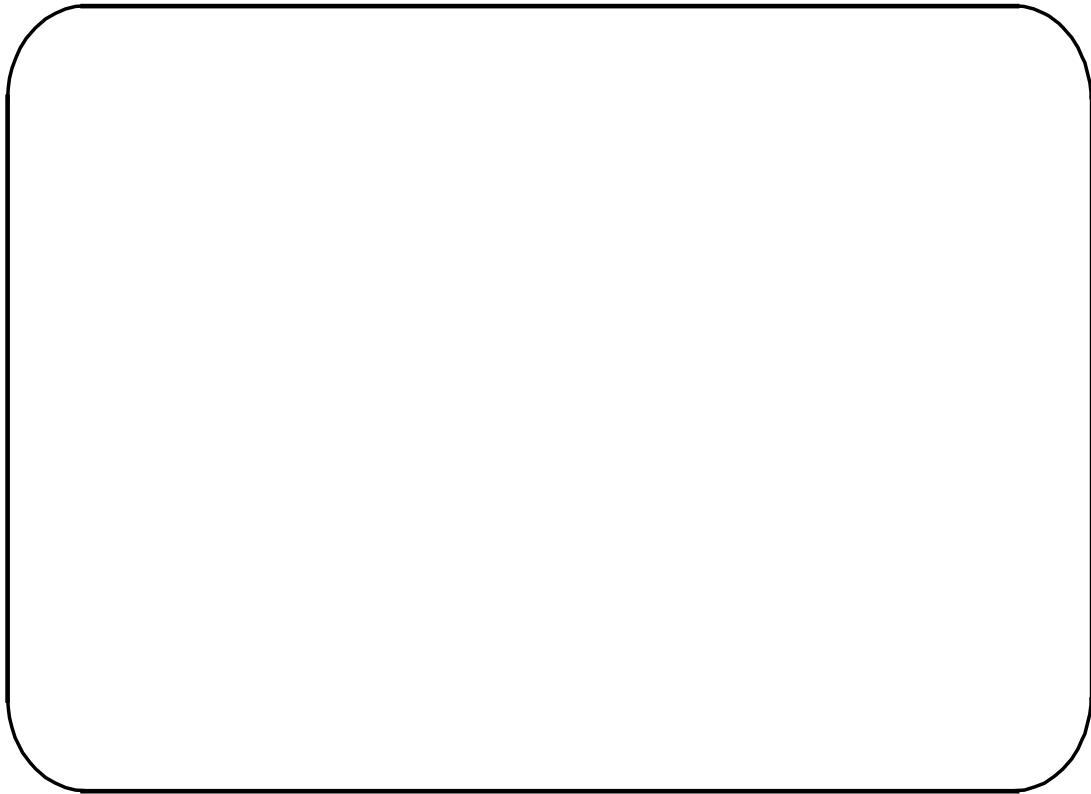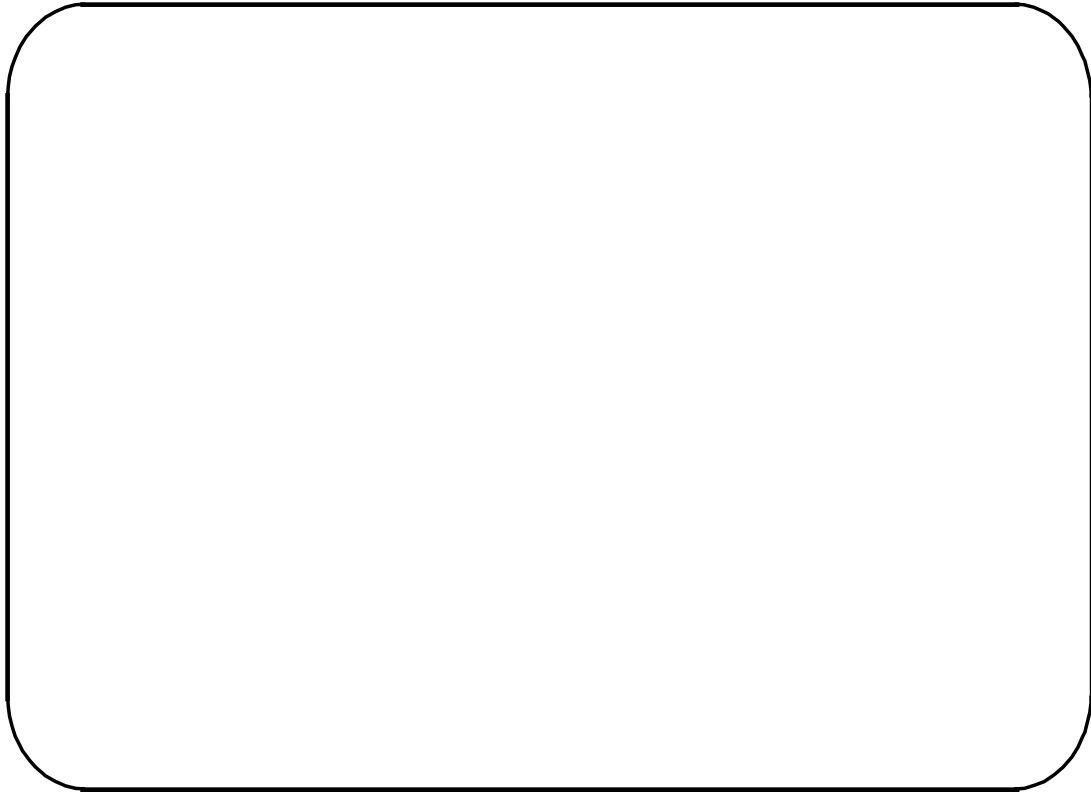
Zt R01: Infeed depth (incremental)

If the infeed depth is assigned with R1 = 0, the infeed is executed immediately to pocket depth at the feedrate. If the pocket cannot be milled with a single infeed, an infeed depth must be entered. The milling process is repeated until the pocket depth is reached. If a residual infeed depth of less than double R01 results, this is divided into two equal values. Enter the infeed depth as an incremental value without sign.

R R24: Radius

For the radius value, enter the distance from the centre point to the elongated hole edge.

Wf R26: Indexing angle

If 0 is stated as the indexing angle, the number of slots is divided into 360°.

Example: **"ELONGATED HOLE" milling pattern machining menu selected via softkey (X/Y plane, infeed axis Z)**

```
%902
N05  G90 G0  X50  Y30  Z20  D01  T01  S600  M03  LF          Select milling position
N10  R1=2.5  R2=2  R3=-5  R13=15
     R15=300  R16=100  R22=50  R23=30  R24=40
     R25=45  R26=0  R27=4  L902  P1  LF                      Call ELONGATED HOLE
N15  Z50  LF
N20  M30  LF
```
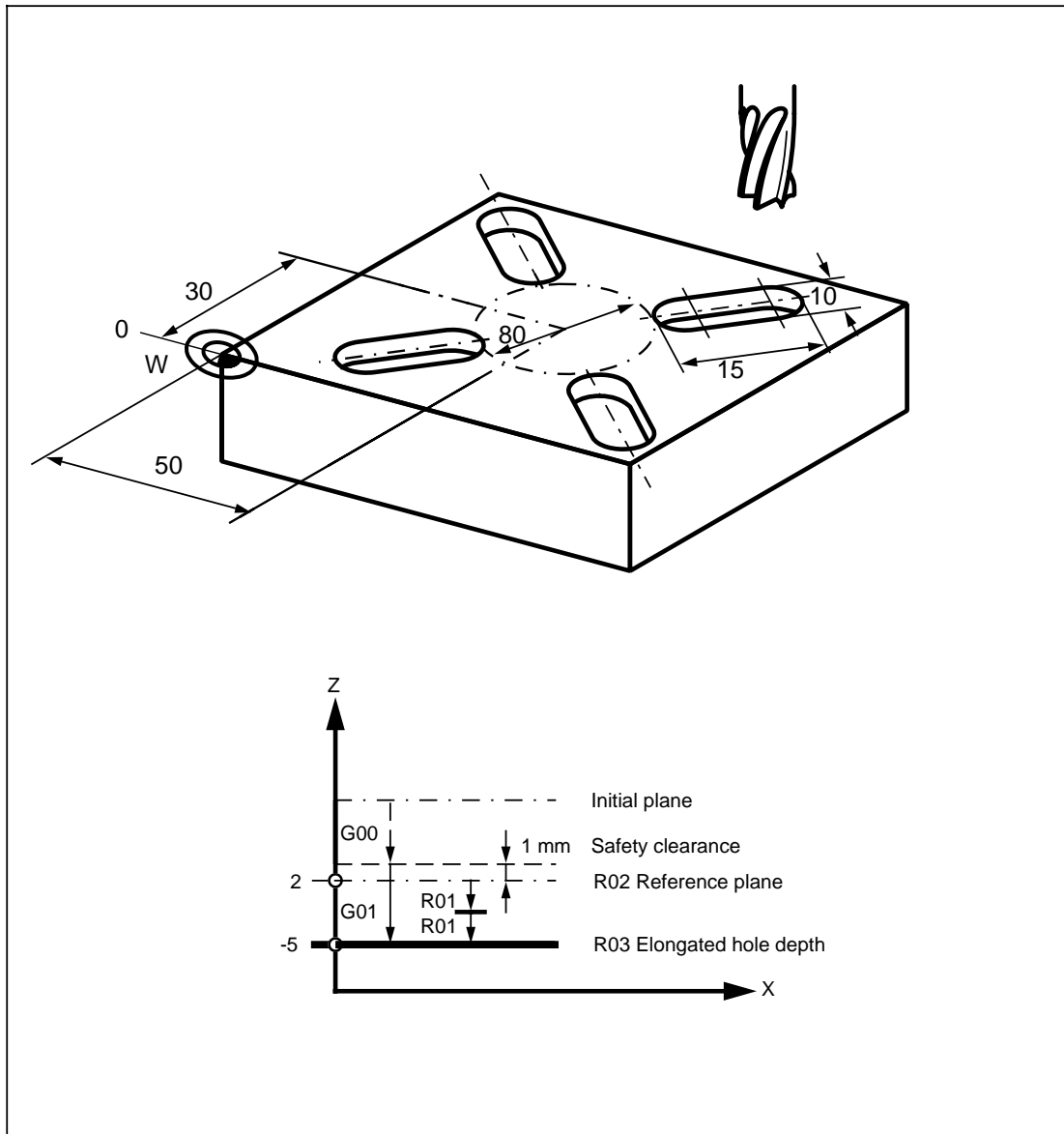
The parameters are assigned in two menu displays.

2.2.2.4 L903 Milling rectangular pocket

During programming, either the "RECTANGULAR POCKET" menu is selected and the R parameters are entered in the menu displays, or these values are directly programmed as parameter assignments in the part program. Subroutine L903 is active in the current plane.

The cycle is also used to mill circular pockets. If the values are programmed directly as parameter assignments, R parameters R12 and R13 are assigned with the values of the pocket diameter. In addition, R24 must be assigned with the pocket radius value.

Symbol	Parameter	Description	
Zt	R01	Enter infeed depth without sign	(incremental)
E1	R02	Reference plane	(absolute)
T	R03	Pocket depth	(absolute)
G	R06	Milling direction (G02/G03)	
L	R12	Pocket length	(incremental)
B	R13	Pocket width	(incremental)
Ff	R15	Feedrate (pocket surface)	
Ft	R16	Feedrate (pocket depth)	
Mw	R22	Centre point of pocket (horizontal)	(absolute)
Ms	R23	Centre point of pocket (vertical) (referred to workpiece zero)	(absolute)
R	R24	Corner radius	

In cycle L903, cutter radius compensation is deselected (G40). The milling cutter radius is automatically taken into account and this must be stored in the tool offset memory.

With setting data 5000, bit 0 = 0, a mode compatible with UMS 2 is possible, i.e. programs generated with UMS 2 can be run.

If programs created with UMS 3/60 are used, parameter R04 must be changed to R16, and setting data 5000 bit 1 must be set to 1.

Zt R01: Infeed depth (incremental)

If the infeed depth is assigned with R1 = 0, the infeed is executed immediately to pocket
depth at the feedrate. If the pocket cannot be milled with a single infeed, an infeed depth must
be entered. The milling process is repeated until the pocket depth is reached. If a residual
infeed depth of less than double R01 results, this is divided into two equal values. Enter the
infeed depth as an incremental value without sign.

G R06: Milling direction (G02/G03)

Program the milling direction (up-cut or down-cut milling) in R06 = 02/03.

L R12:   Pocket length (incremental)
B R13:   Pocket width (incremental)

If the milling cutter radius is equal to or greater than half of the smaller pocket side, error
message 4102 (cutter radius too great) is issued.

R R24: Corner radius

Make sure that the milling cutter radius is no greater than the desired corner radius. No error
message is issued.

Example: **"RECTANGULAR POCKET" machining menu selected via softkey**
(X/Y plane, infeed axis Z)

```
%903
N05  G90  G0  X40  Y30  Z20  D05  T04  S600  M03  LF          Select milling position
N10  R1=2.5  R2=2  R3=-5  R6=3  LF
     R12=35  R13=22  R15=300  R16=100  LF
     R22=40  R23=30  R24=8  L903  P1  LF                      Call rectangular pocket
N15  Z50  LF
N20  M30  LF
```

The parameters are assigned in two menu displays.
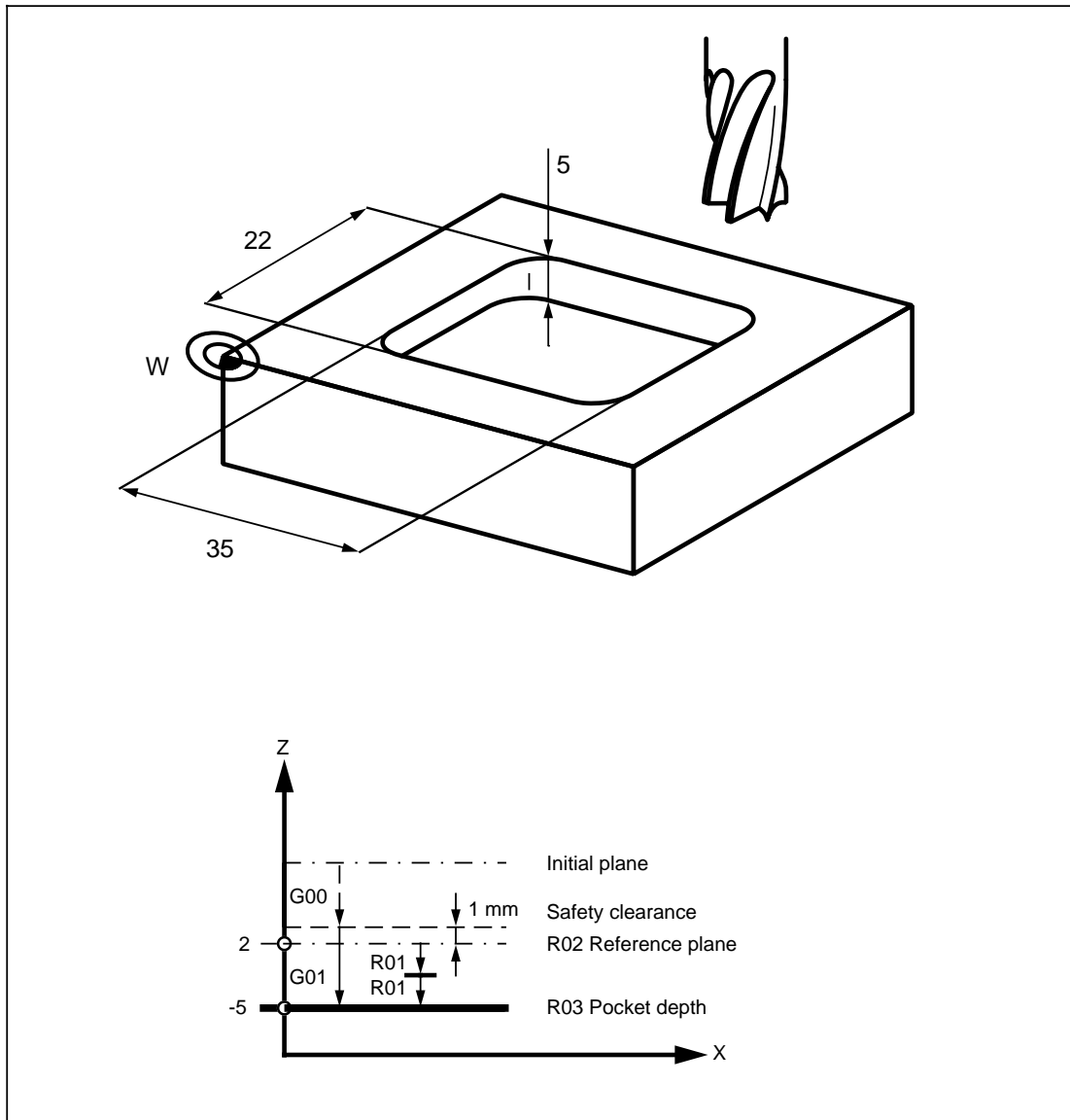
2.2.2.5 L904 "CIRCULAR SLOT" milling pattern

During programming, either the "CIRCULAR SLOT" menu is selected and the R parameters are entered in the menu displays, or these values are programmed directly as parameter assignments in the part program. Subroutine 904 is active in the current plane.

The cycle automatically selects and deselects the cutter radius compensation (G41 and G40 respectively).

Symbol	Parameter	Description	
E1	R02	Reference plane	(absolute)
T	R03	Circular slot depth	(absolute)
E2	R10	Retraction plane	(absolute)
B	R12	Circular slot width	(incremental)
Wf	R13	Angle for slot length (referred to horizontal axis)	
Ff	R15	Feedrate (pocket surface)	
Ft	R16	Feedrate (pocket depth)	
Mw	R22	Centre point of circular slot (horizontal)	(absolute)
Ms	R23	Centre point of circular slot (vertical) (referred to workpiece zero)	(absolute)
R	R24	Radius (part radius)	
Wa	R25	Starting angle (referred to horizontal axis)	
	R27	Number of slots	

SINUMERIK System 800, Cycles, UMS 4 (PG)

Wa R25: Starting angle

The starting angle refers to the horizontal axis of the first circular slot.
e.g. R25 = 0°.
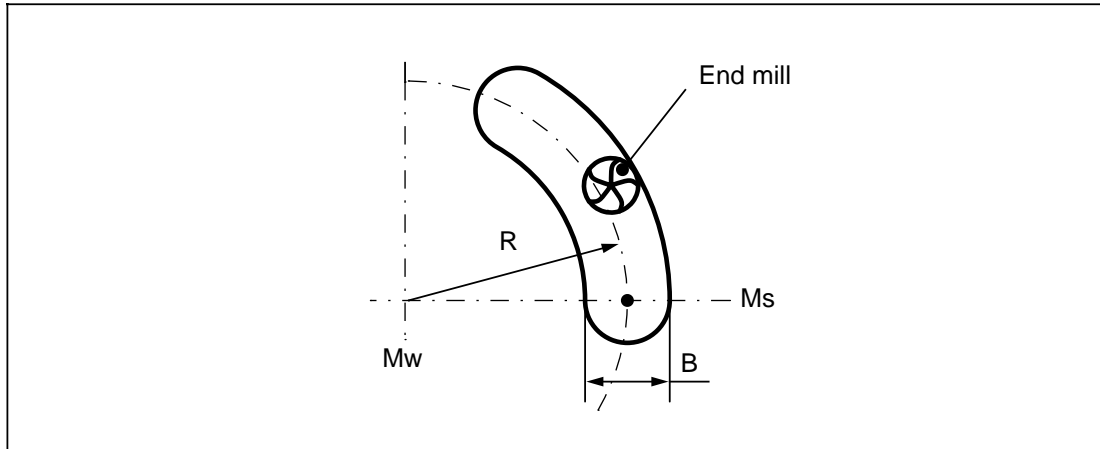
T R03: Circular slot depth (absolute)

The infeed is executed immediately to the programmed slot depth at the feedrate.

B R12: Slot width

The milling cutter diameter must not be less than half the slot width. If the milling cutter diameter is equal to or greater than the slot width, fault message 4102 is issued (cutter radius too great).

SINUMERIK System 800, Cycles, UMS 4 (PG)

Example: **"CIRCULAR SLOT" machining menu selected via softkey**
(X/Y plane, infeed axis Z)

```
%904
N05  G90 G0 X50 Y30 Z20 D05 T04 S600 M03  LF          Select milling position
N10  R2=4   R3=-5   R10=10  R12=6  LF
     R13=60 R15=300 R16=100  LF
     R22=55 R23=55  R24=40  R25=90  LF
     R27=3  L904 P1  LF                                Call circular slot
N15  Z50  LF
N20  M30  LF
```
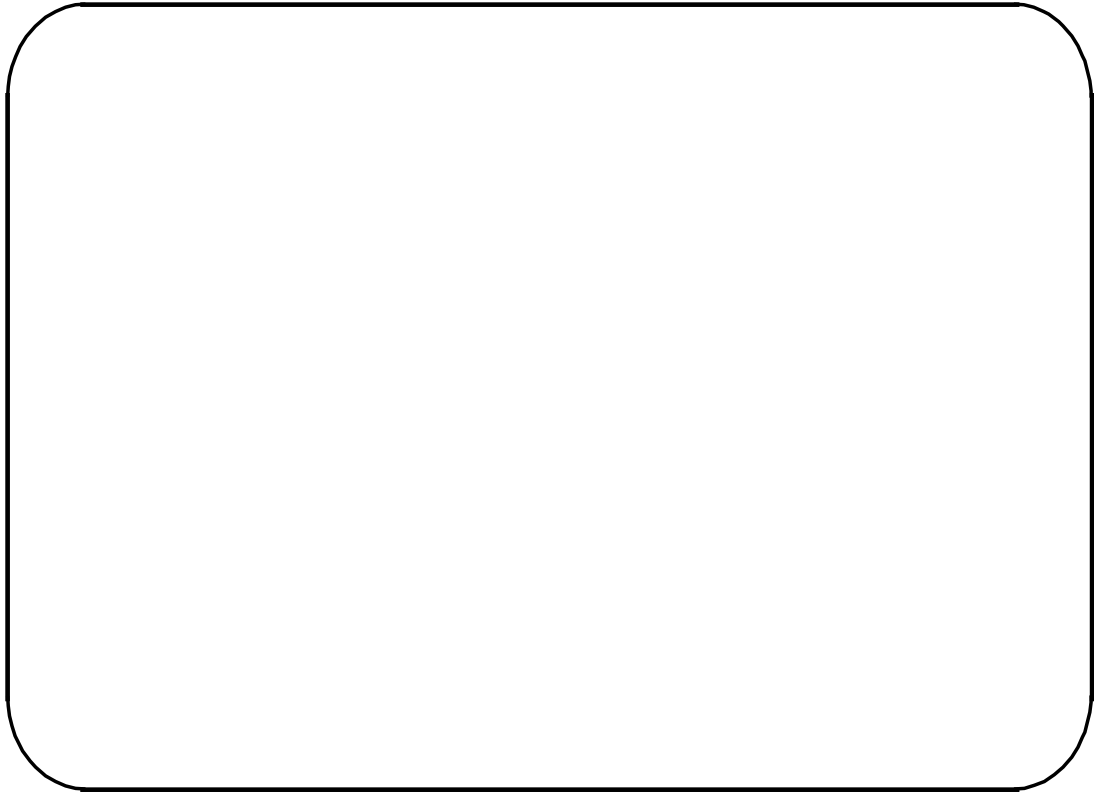
The parameters are assigned in two menu displays.
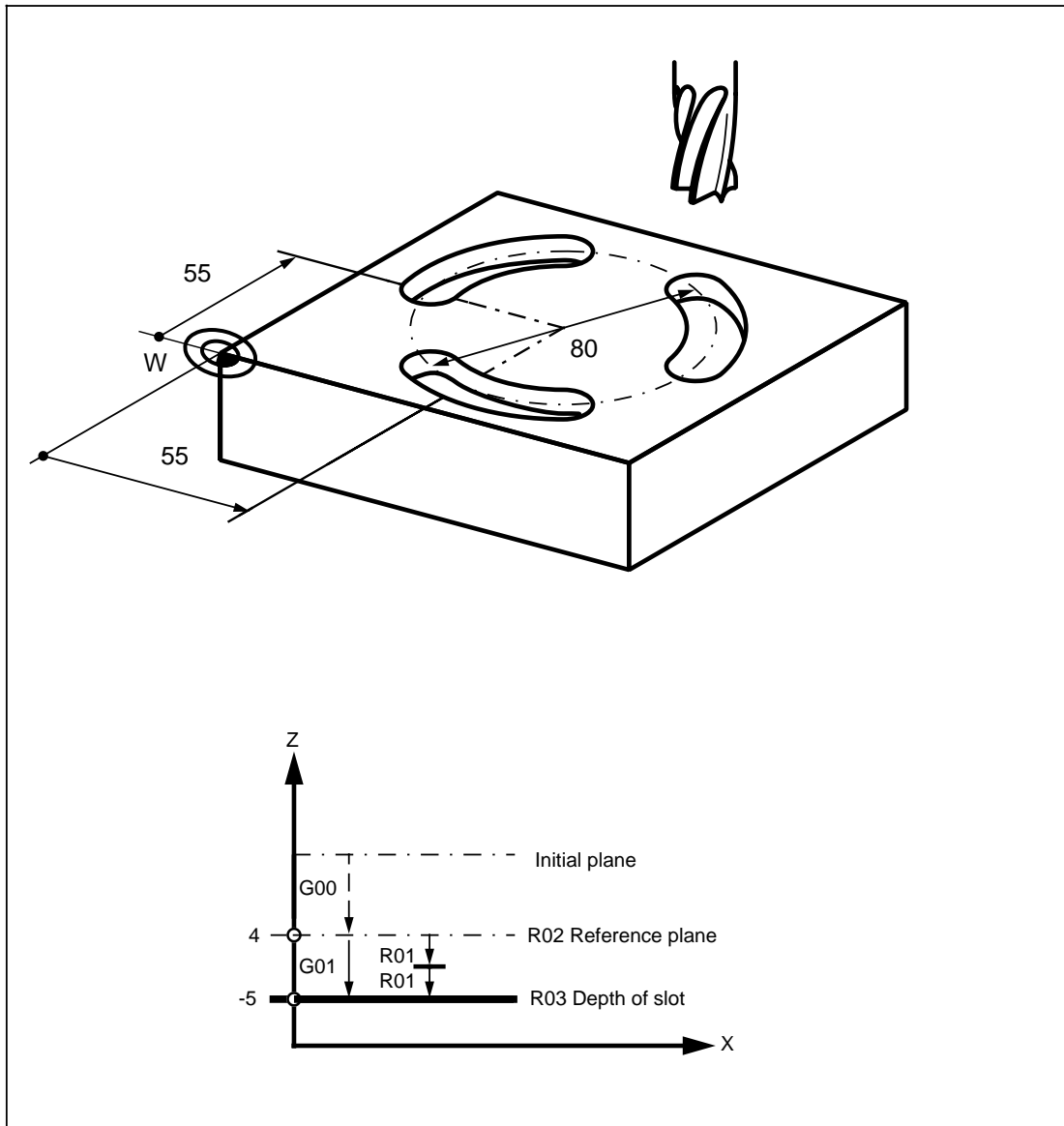
2.2.2.6 L905 "SINGLE HOLE" drilling pattern

During programming, either the "SINGLE HOLE" menu is selected and the R parameters are entered in the menu displays, or these values are programmed directly as parameter assignments in the part program. Subroutine L905 is active in the current plane.

Symbol	Parameter	Description	
Mw	R22	Centre point of hole (horizontal)	(absolute)
Ms	R23	Centre point of hole (vertical) (referred to workpiece zero)	(absolute)
	R28	Number of required drilling cycle (L81 to L89)	
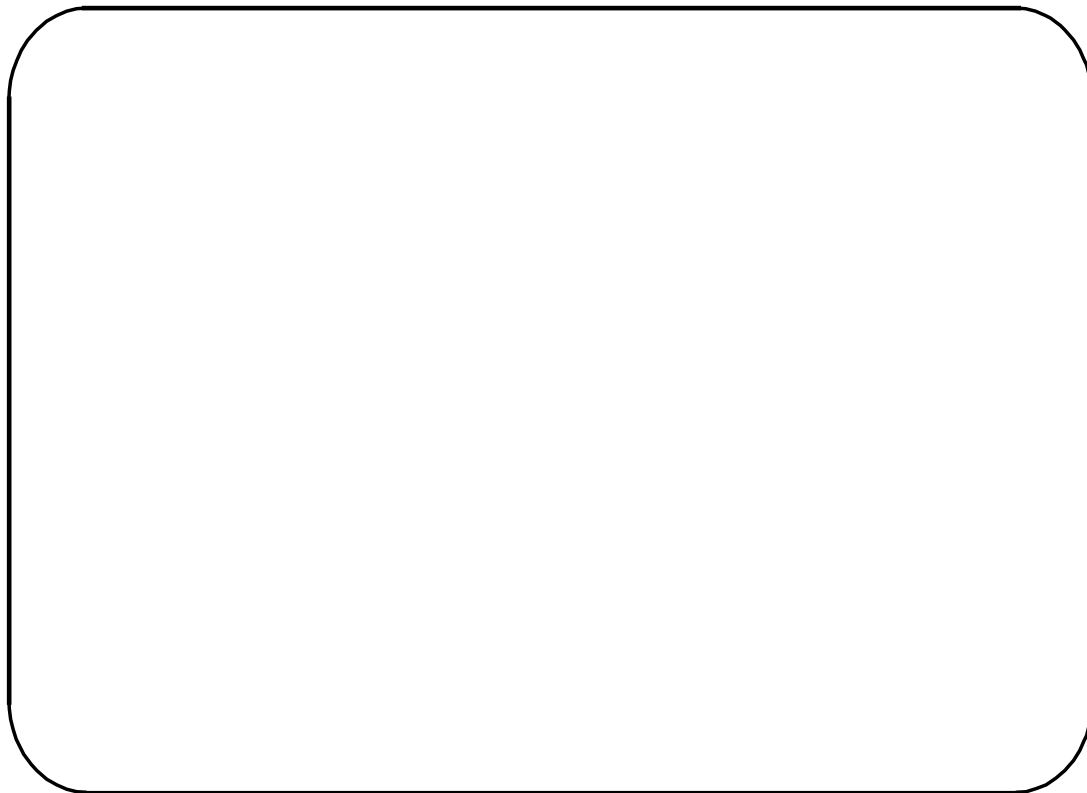
R28: Number of required drilling cycle (L81 to L89)

The parameters necessary for the required parameters must be defined in the part program.

2.2.2.7 L906 "ROW OF HOLES" drilling pattern

During programming, either the "ROW OF HOLES" menu is selected and the R parameters are entered in the menu displays, or these values are programmed directly as parameter assignments in the part program. Subroutine L906 is active in the current plane.

Symbol	Parameter	Description	
L1	R18	Distance from centre point	(incremental)
L2	R19	Hole spacing	(incremental)
Mw	R22	Centre point of hole (horizontal)	(absolute)
Ms	R23	Centre point of hole (vertical) (referred to workpiece zero)	(absolute)
Wa	R25	Starting angle (referred to horizontal axis)	
	R27	Number of holes	
	R28	Number of drilling cycle (L81 to L89)	

L1 R18: Distance from centre point (incremental)

R18 must be assigned the distance from the centre point (R22, R23) to the first hole in the row of holes.

L2 R19: Hole spacing (incremental)

R19 is the hole spacing and must be entered as an incremental value.

R28: Number of required drilling cycle (L81 to L89)

The parameters necessary for the required parameters must be defined in the part program (compare example N15).

Example: "ROW OF HOLES" processing menu selected via softkey
(X/Y plane, drilling axis Z)

```
%906
N05  G90  G0  X25  Y25  Z50  D05  T04  LF
N10  G1  F130  S710  M03  LF
N15  R2=4  R3=-8  R10=10                                    Supply drilling cycle
     R18=0  R19=15  R22=25
     R23=25  R25=45 R27=3  R28=81  LF
     L906  P1  LF                                           Call row of holes
N20  Z50  LF
N25  M30  LF
```


Drilling depth
8 mm

SINUMERIK System 800, Cycles, UMS 4 (PG)
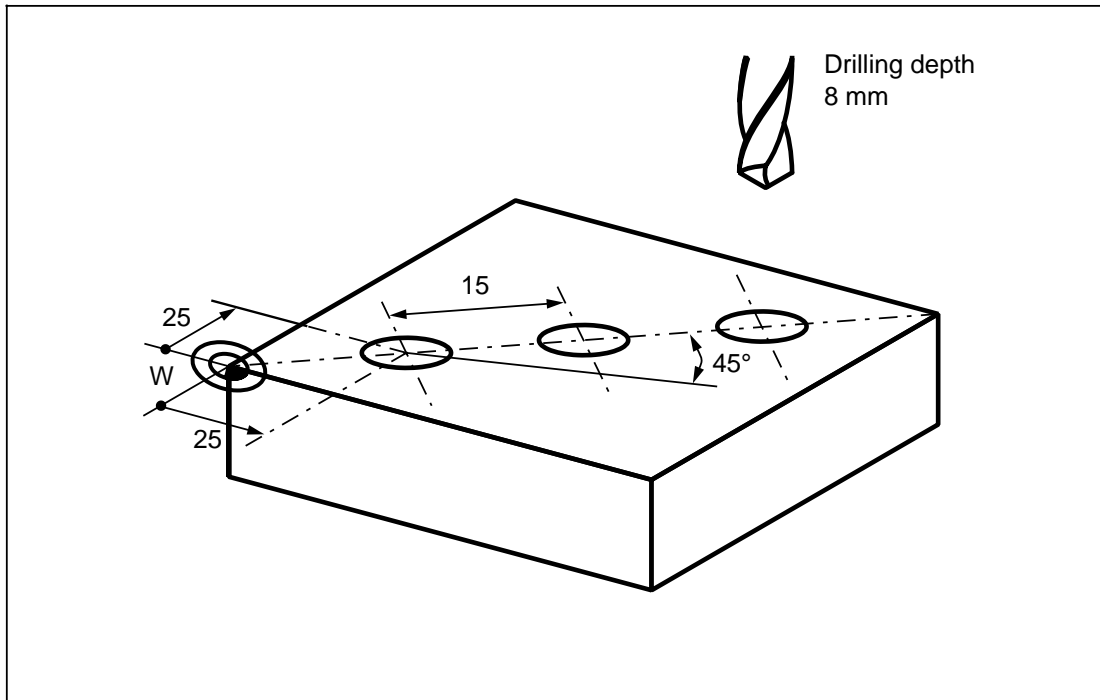
2.2.2.8 L930 Milling circular pocket

During programming, either the "Circular pocket" menu is selected and the R parameters are entered in the menu displays, or these values are programmed directly as parameter assignments in the part program. Subroutine L930 is active in the current plane.

Symbol	Parameter	Description	
Zt	R01	Enter infeed depth without sign	(incremental)
E1	R02	Reference plane	(absolute)
T	R03	Pocket depth	(absolute)
G	R06	Milling direction (G02/G03)	
Ff	R15	Feedrate (pocket surface)	
Ft	R16	Feedrate (pocket depth)	
Mw	R22	Centre point of circular pocket (horizontal)	(absolute)
Ms	R23	Centre point of circular pocket (vertical) (referred to workpiece zero)	(absolute)
	R24	Pocket radius	

In cycle L930, the cutter radius compensation is deselected (G40). The milling cutter radius is automatically taken into account and this must be stored in the tool offset memory.

If programs created with UMS 3/60 are used, parameter R04 must be changed to R16, and setting data 5000 bit 1 must be set to 1.

Zt R01: Infeed depth (incremental)

If the infeed depth is assigned with R1 = 0, the infeed is executed immediately to pocket depth at the feedrate. If the pocket cannot be milled with a single infeed, an infeed depth must be entered. The milling process is repeated until the pocket depth is reached. If a residual infeed depth of less than double R01 results, this is divided into two equal values. Enter the infeed depth as an incremental value without sign.

G R06: Milling direction (G02/G03)

After the plunge cut into the workpiece, the milling cutter describes a path that runs towards the outside in a spiral. Program the milling direction (up-cut or down-cut milling) in R06 = 02/03.

SINUMERIK System 800, Cycles, UMS 4 (PG)
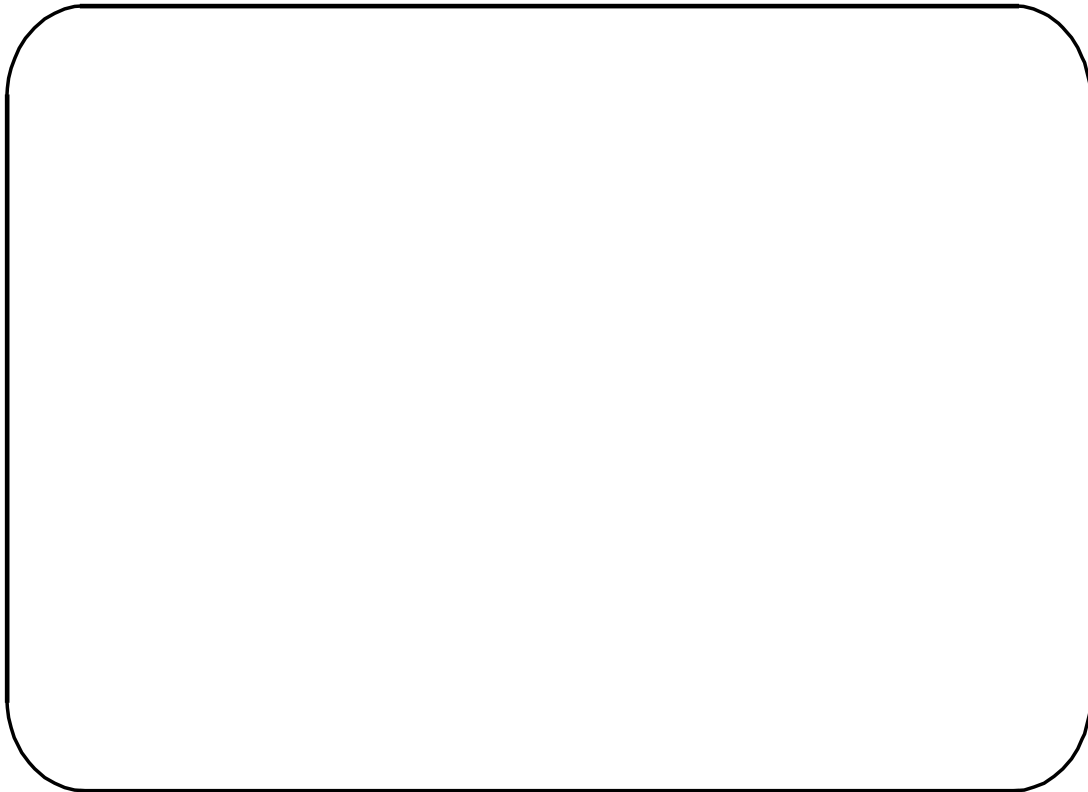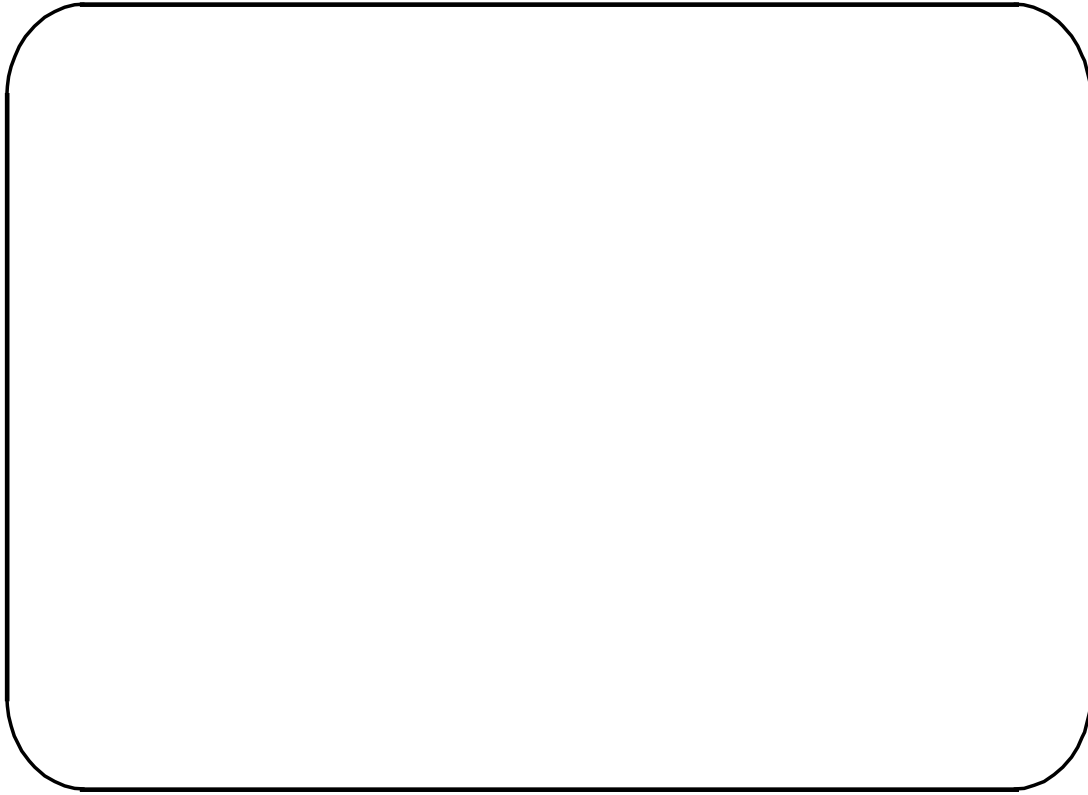
R R24: Pocket radius

If the miller radius is equal to or greater than the pocket radius, fault message 4102 is output
(cutter radius too great).

Example: "CIRCULAR POCKET" machining menu selected via softkey
(X/Y plane, infeed axis Z)

```
%930
N05  G90  G0  X50  Y30  Z20  D05  T04  S600  M03  LF          Select milling position
N10  R1=2,5  R2=2  R3=-5  R6=3
     R15=300  R16=100  R22=50  R23=30
     R24=12  L930 P1  LF                                      Call circular pocket
N20  Z50  LF
N35  M30  LF
```

The parameters are assigned in two menu displays.
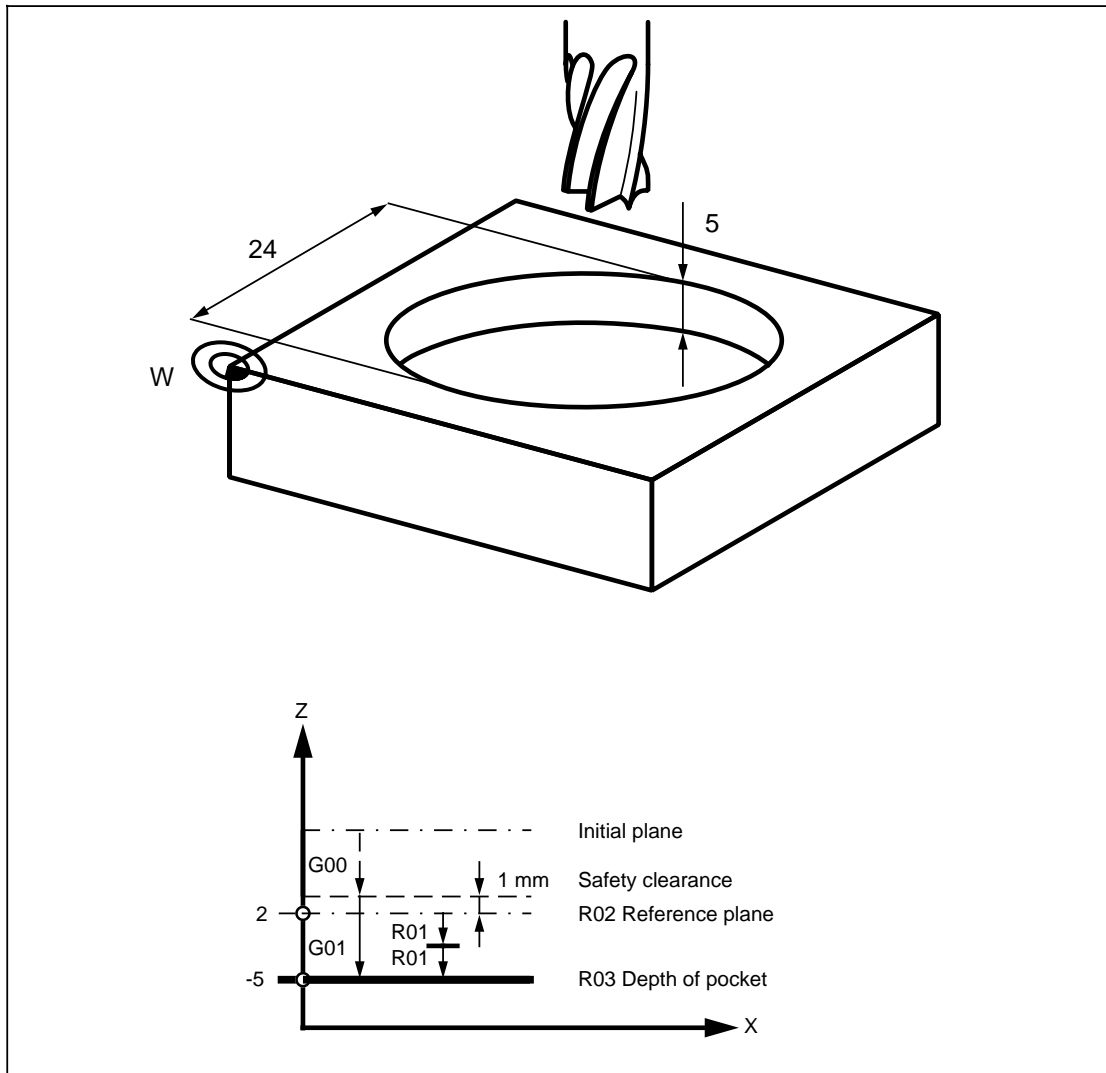
2.3 L999 Clear buffer memory

A series of control signals from the interface control are not registered directly to the working memory of the NC, but via the buffer memory. These signals (which can be selected, for example, by M functions) include:

- external additive ZO
- mirroring
- external tool offset

If these functions, which are actuated in the running (active) program, are to be effective in the block following their selection, the buffer memory must be cleared.

Otherwise, the selected control signal becomes active only several blocks later.

In each program, the buffer memory can be cleared by calling subrouting L999 once.

The program L999 should be defined as follows:

```
L999
@714  LF
M17  LF
```

Example: **Selection of external tool offset,**
e.g. after measurement of the tool

```
.
.
.
N15 M..  LF                          PLC removes read-in signal with the help of M function
                                              PLC executes the external tool offset
                                              PLC then gives read-in enable signal again

N20 G04 X..  LF                          Dwell greater than 1.2 * max. PLC scantime
N25 L999 P1  LF                                        Clear buffer memory
.
.
.
N.. M30  L
```

2.4 L960 Transfer of zero offset groups [1]

The cycles for transferring zero offset groups are not supplied together with the standard cycles and must be ordered separately.

L960_EZS
This program makes it possible to store up to 10 zero offset blocks with 5 axes in the SINUMERIK System 800 and to replace the data of the zero offset memory with these blocks.

This cycle transfers the input buffer parameters (MIB 200 to MIB 399) into the settable ZO memory coarse (G54 to G57) and vice versa.

The required block is selected via the R0 parameter. The contents of R0 are automatically stored in MD 18 by cycle L960. The ZO data can either be entered into the MIB parameters by the program or manually by using the input images. 12344.321 is to be entered for parameter R0 if transfer from the ZO memory to MIB parameters is selected. The transfer target is defined by the current MD 18.

L960_RPA
This cycle is used for transferring a maximum of 3 ZO blocks with 5 axes into R parameters (R240-R299) and vice versa. The parameters are assigned in the system menu image.

Prerequisites:

Both cycles are not processed in SIMULATION.

Special characteristics relevant if L960 is used together with the SIEMENS measuring cycles are described in the documentation of the measuring cycles.

The following parameters must be defined before calling L960_EZS:

R0=1   Transfer MIB 200 to MIB 219   ZO memory
R0=2   Transfer MIB 220 to MIB 239   ZO memory
:
:
R0=10   Transfer MIB 380 to MIB 399   ZO memory

R0=12344.231   Transfer ZO memory MIB 380 to MIB 399
                  The current ZO block is specified in NC-MD 18.
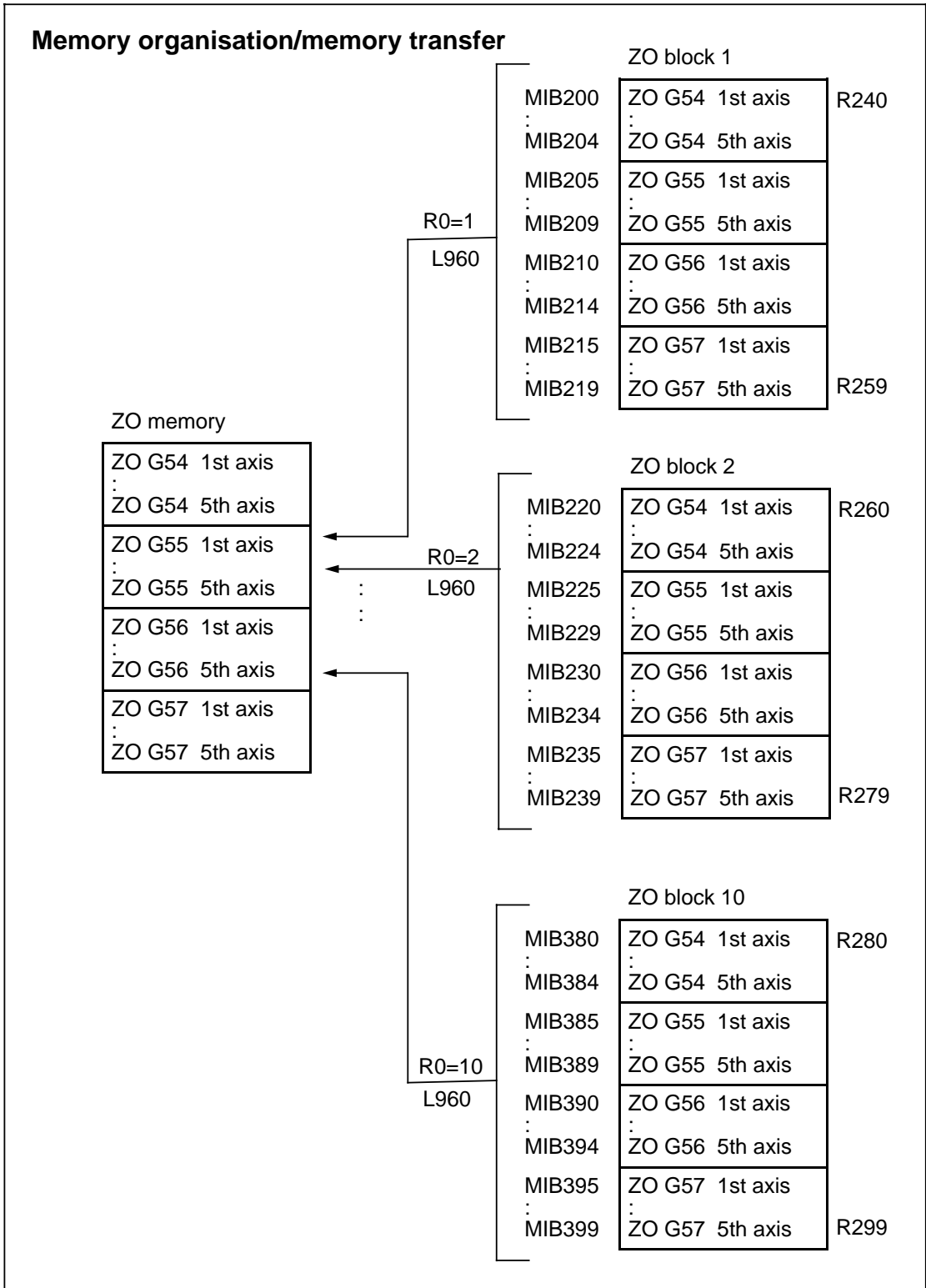
The MIB parameters are then reserved for this purpose only.

Programming example:

```
%MPF123
N5 R0=2 L960 P1      Transfer of the 2nd ZO block MIB 220 to MIB 239 into the ZO memory
:
:
N55 R0=12344.321 L960 P1        Transfer ZO memory MIB 220 to MIB 239
:
:
N2000 M30
```

_____

[1]   *for SINUMERIK 810/820, cycle L960 is described in the "Measuring Cycles" documentation.*

Memory organisation/memory transfer

ZO block 1

MIB200	ZO G54 1st axis	R240
:	:	
MIB204	ZO G54 5th axis	
MIB205	ZO G55 1st axis	
:	:	
MIB209	ZO G55 5th axis	
MIB210	ZO G56 1st axis	
MIB214	ZO G56 5th axis	
MIB215	ZO G57 1st axis	
:	:	
MIB219	ZO G57 5th axis	R259

R0=1
L960

ZO memory

ZO G54 1st axis
:
ZO G54 5th axis
ZO G55 1st axis
ZO G55 5th axis
ZO G56 1st axis
:
ZO G56 5th axis
ZO G57 1st axis
:
ZO G57 5th axis

R0=2
L960

ZO block 2

MIB220	ZO G54 1st axis	R260
:	:	
MIB224	ZO G54 5th axis	
MIB225	ZO G55 1st axis	
:	:	
MIB229	ZO G55 5th axis	
MIB230	ZO G56 1st axis	
:	:	
MIB234	ZO G56 5th axis	
MIB235	ZO G57 1st axis	
:	:	
MIB239	ZO G57 5th axis	R279

ZO block 10

MIB380	ZO G54 1st axis	R280
:	:	
MIB384	ZO G54 5th axis	
MIB385	ZO G55 1st axis	
:	:	
MIB389	ZO G55 5th axis	
MIB390	ZO G56 1st axis	
MIB394	ZO G56 5th axis	
MIB395	ZO G57 1st axis	
:	:	
MIB399	ZO G57 5th axis	R299

R0=10
L960

2.4.1   Generating the UMS

Each project comprises

- the user menu tree

- a link list L960LIST.LBD

- the two versions of the cycle

    –   L960_EZS.ZPL   Transfer of 10 ZO groups with MIB parameters
    –   L960_RPA.ZPL   Transfer of 3 ZO groups with R parameters

The following conditions must be fulfilled for being able to link a UMS file:

- The exit from the system menu tree to the user menu tree must be configured. The return jump into the system menu tree must be changed in the user menu tree if required.

- The link list contains the cycle in the version for MIB parameters. The required cycle version is to be entered there and the modified system menu tree is to be added.

- The alarm text for alarm no. 4200 ”Check definition R (Nxxxx)” is to be entered in an alarm text file; if the measuring cycles are used, this file already exists.

SINUMERIK System 800, Cycles, UMS 4 (PG)

Siemens AG

AUT V24
P.O. Box 31 80

D-91050 Erlangen

Federal Republic of Germany

From:	
Name	
Address of company**/**department	
Street:	
Postal code: Place:	
Telephone: **/**	
Fax: **/**	

Suggestions

Corrections

For Publication/Manual:

SINUMERIK System 800
Cycles, User Memory Submodule 4

User Documentation

Programming Guide

Order No.:   6ZB5 410-0BQ02-0BA5
Edition: January 94

Should you come across any printing errors when reading this publication, please notify us on this sheet. Suggestions for improvement are also welcome.

Suggestions and/or corrections