SINUMERIK 840C/840
Cycles

Programming Guide 08.96 Edition

User Documentation

SINUMERIK 840C/840
Cycles

Programming Guide

User Guide

Valid for:

Control	*Software version*
SINUMERIK 840C	1, 2, 3, 4, 5
SINUMERIK 840T/M	1, 2

08.96 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
05.90	6ZB5 410-0FL02-0BA0	**A**
10.90	6ZB5 410-0FL02-0BA1	**C**
07.91	6ZB5 410-0FL02-0BA2	**C**
12.93	6FC5 198-3AA50-0BP0	**C**
08.96	6FC5 198-5AA50-0BP0	**C**

Siemens quality for software and training
to DIN ISO 9001, Reg. No. 2160-01

Other functions not described in this documentation might be executable in the control. This does not, however, represent an obligation to supply such functions with a new control or when servicing.

This publication was produced on the Siemens 5800 Office System.

We have checked that the contents of this publication agree with the hardware and software described herein. The information given in this publication is reviewed at regular intervals and any corrections that might be necessary are made in the subsequent printings. Suggestions for improvement are welcome at all times.

Subject to change without prior notice.

Contents

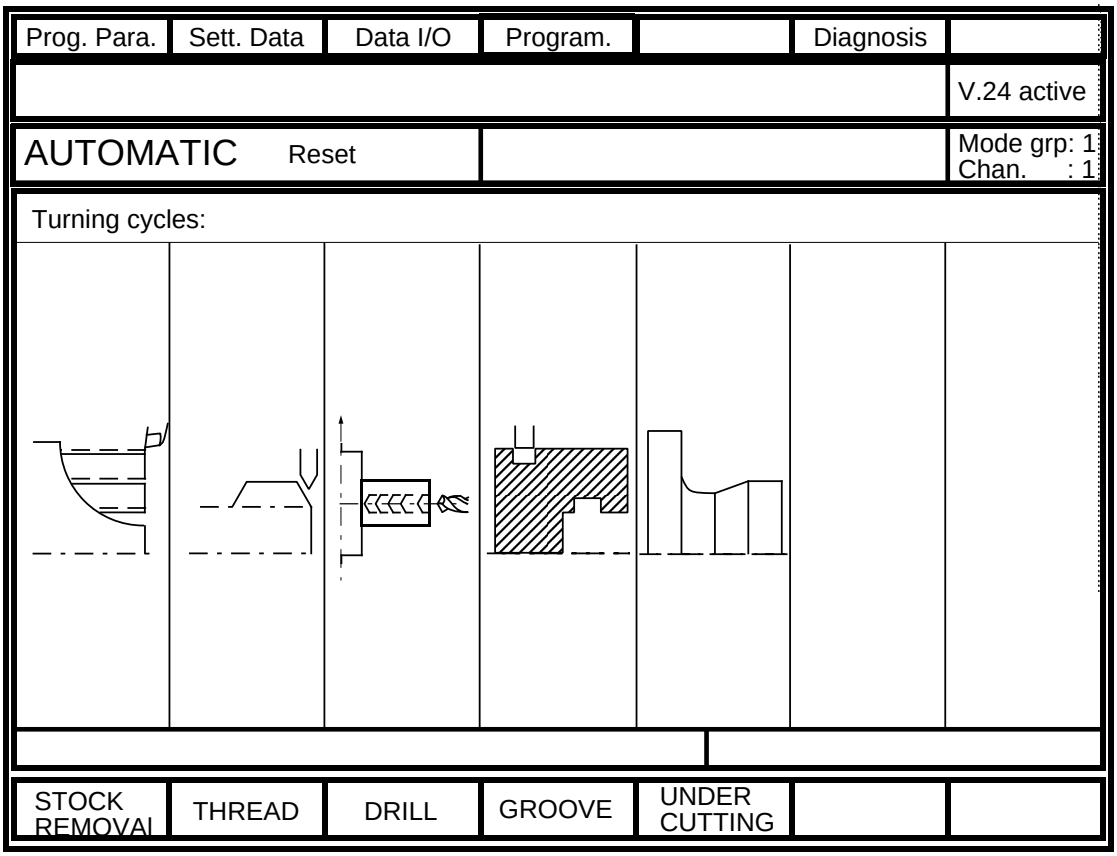
1 Introduction

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.

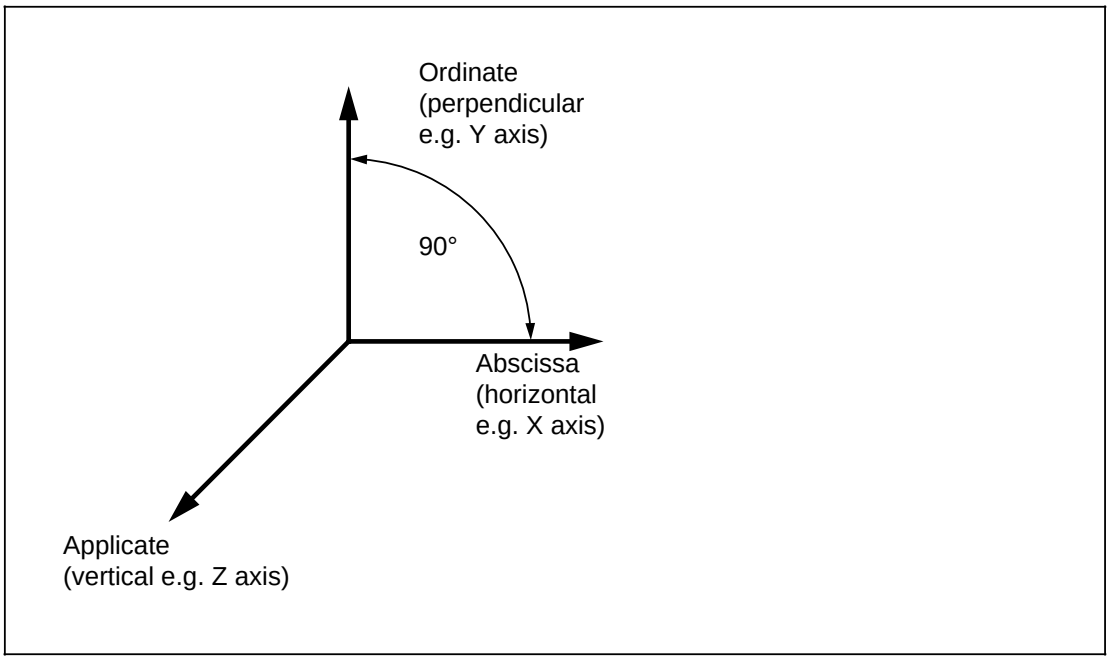
The cycles end uniformly with preparatory functions G00, G60 and G90. Any other G functions required when the program continues must be reprogrammed.

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 the SINUMERIK 840C/840 control.
- If the R parameters are 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.



- 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.

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 want to use 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 are 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)

Setting data

Special cycle functions can be activated via the setting data bits 5000.5 to 5000.7.

SD 5000.5 = 0 Thread cutting cycles L97 and L99 execute as before

= 1 In thread cutting cycles L97 and L99, retraction is executed with G0 and G620. The G command of the G600 group, which has been active prior to cycle call is retained beyond the cycle.

This setting data bit applies with UMS 54 and higher.

SD 5000.6 = 0 L900 runs as before (traversing a clearance distance of 1 mm along the drilling axis)

= 1 L990 without a clearance distance along the drilling axis

This setting data bit applies with UMS 48 and higher.

SD 5000.7 = 0 Overrun correction is not calculated

= 1 Overrun correction is calculated

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**
840T	041 xx
840M	042 xx
840C T Version	061 xx
840C M Version	062 xx

Example:
SINUMERIK 840C T Version with NC Software Version 2: MD157 = 06102

Machine data (MD) 19

The P number of the TO memory can be written in machine data MD 19 in the SINUMERIK 840C*I*840. The address of the following cutting edge is given here.

SINUMERIK 840 MD 19	= 10	(standard value)
	= 5 to 9	(variable P number)
	= 0	(D = n+1)

The following alarms are output in the cycles:

Alarms	Description
4100	No D number active (L901-904, L930, L93-L95)
4101	Tool radius = 0 (L903, L930)
4102	Cutter radius too great (L901, L903, L904, L930)
4103	Tool too wide (L93)
4120	No spindle direction programmed (L84)
4121	Spindle not inside tolerance range (L84)
4140	Machined part diameter too small (L94)
4153	Thread length too short (L84)
4180	Option not available (L94)
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.

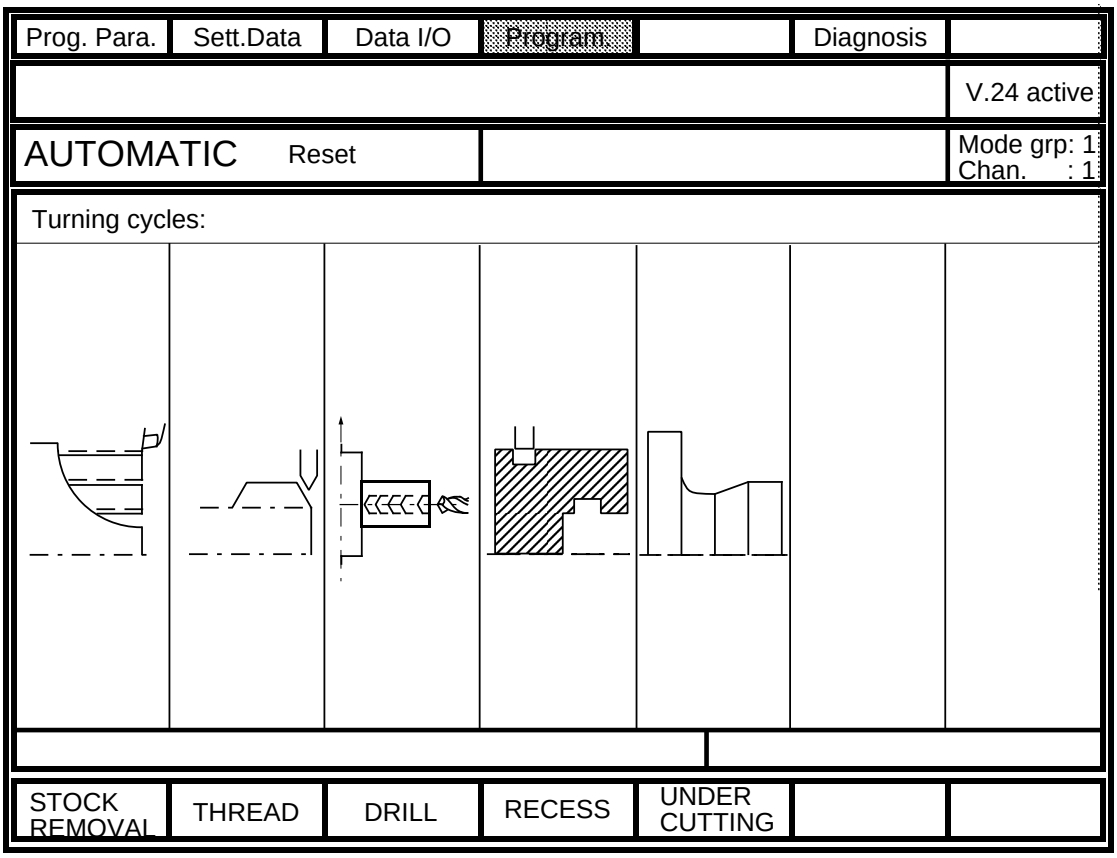
If the cycles are supplied with data via menu displays, several storage softkeys are available for selection:

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

END OF SECTION

2 Cycles

2.1 Machining cycles for turning



2.1.1 L93 Recessing cycle (prerequisite: blueprint programming)

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

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

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.

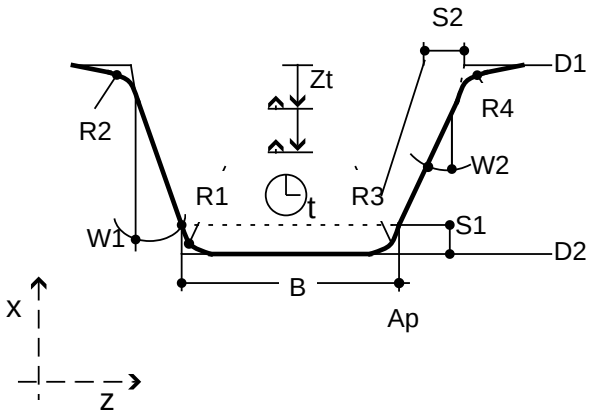
In connection with tool management in the 840C/840 controls and L93, the cycle machine data (MDZ) 7000 Bit 4 must be set to 1.

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 of 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 recess base	(incremental)
S2	R25	Finishing cut depth of flanks	(incremental)
Zt	R26	Infeed depth, enter without sign	(incremental)
B	R27	Width of recess	(incremental)
t	R28	Dwell at recess depth	
W1	R29	Angle (0 to 89 degrees)	
R1	R30	Radius or chamfer at recess base	
D2/L2	R31	Recess diameter or length of recess depth	(face) (absolute)
R2	R32	Radius or chamfer at recess edge	
R3	R33	Radius or chamfer at recess edge	
R4	R34	Radius or chamfer at recess edge	
W2	R35	Angle (0 to 89 degrees)	

Prog. Para.	Sett. Data	Data I/O	Program		Diagnosis	
						V.24 active
AUTOMATIC Reset						Mode grp: 1 Chan. : 1

Turning cycles: Recessing outside right



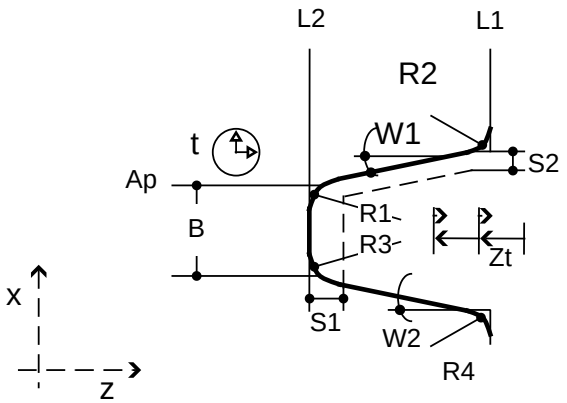
D1	R21 = 0
Ap	R22 = 0
S1	R24 = 0
S2	R25 = 0
Zt	R26 = 0
B	R27 = 0
t	R28 = 0
W1	R29 = 0
R1	R30 = 0
D2	R31 = 0
R2	R32 = 0
R3	R33 = 0
R4	R34 = 0
W2	R35 = 0

Outside diameter (abs.)

STORE MENU	STORE CHOICE	STORE				

Prog. Para.	Sett. Data.	Data I/O	Program.		Diagnosis	
						V.24 active
AUTOMATIC Reset						Mode grp: 1 Chan. : 1

Turning cycles: Recessing right external



L1	R21 = 0
Ap	R22 = 0
S1	R24 = 0
S2	R25 = 0
Zt	R26 = 0
B	R27 = 0
t	R28 = 0
W1	R29 = 0
R1	R30 = 0
L2	R31 = 0
R2	R32 = 0
R3	R33 = 0
R4	R34 = 0
W2	R35 = 0

Start length (abs.)

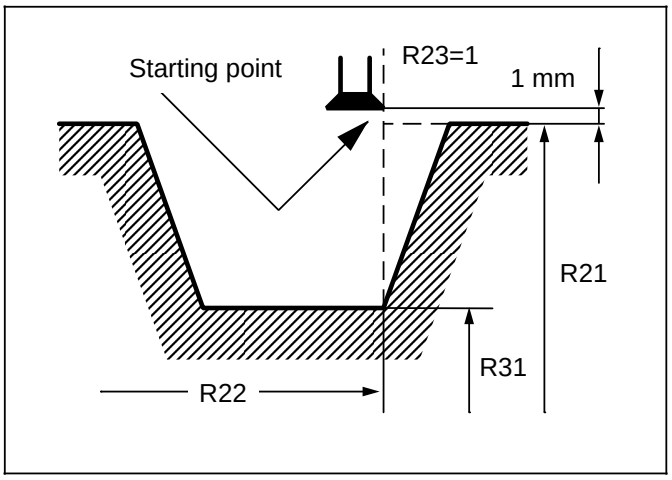
STORE MENU	STORE CHOICE	STORE				

R10: Type of machining

R10 defines the type of recess:

Longitudinal cut: R10 = 0
Facing cut: R10 = 1

D1/L1 R21: External or internal dimension or starting length (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 cut traversing is first in the Z direction, for an internal cut in X direction.

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

R23: Control parameter

The control parameter defines the starting point:

Longitudinal cut:			Facing cut:		
23 =	1	Outside right	R23 =	1	Right inside
	1	Inside right		1	Left inside
	–1	Outside left		–1	Outside right
	–1	Inside left		–1	Left outside

S1 R24: Finishing cut depth at recess base (incremental)
S2 R25: Finishing cut depth of flanks

Finishing cut depths R24/R25 can be input with different values.

Stock removal (roughing) is effected down to the programmed finishing cut depth. Then the finishing incrementing is removed parallel to contour with the same tool.

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

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

Zt R26: Infeed depth (incremental)

Infeed depth programming permits determination of whether the recess depth is to be effected in one or more cuts. If several cuts are required, the tool is retracted by 1 mm for chip breaking after each infeed (compare stop 2).

B R 27: Width of recess (incremental)

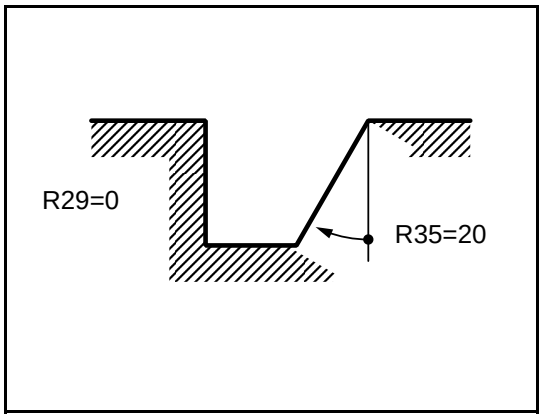
If the width of recess is wider than the cutting tool, the infeed is divided into equal parts. The maximum infeed depends on the tool width. It is 95 % of the tool width *after* deduction of the cutter radii. This guarantees overlapping cuts.

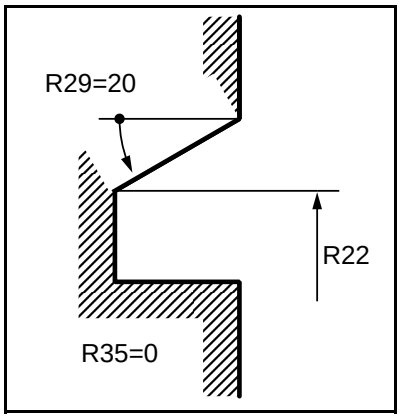
t R28: Dwell at recess depth

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

W1 R29: Angle
W2 R35: Angle

The edge angle can be between 0 and 89 degrees. With longitudinal cuts, enter the angle from the perpendicular axis, and with face cuts the angle from the horizontal axis.





D2/L2 R31: Recess diameter or length of recess depth (face) (absolute)

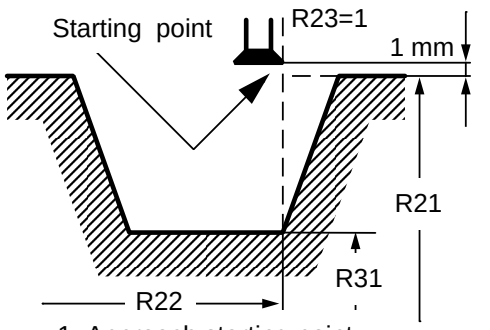
R31 determines the recess depth:

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

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

Signs + = Radius
– = Chamfer

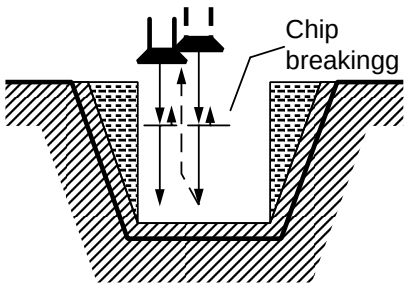
Machining sequence:



1. Approach starting point

1st step:

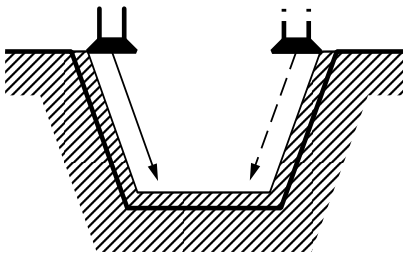
Automatic approach to programmed starting point.



2. Radial grooves

2nd step:

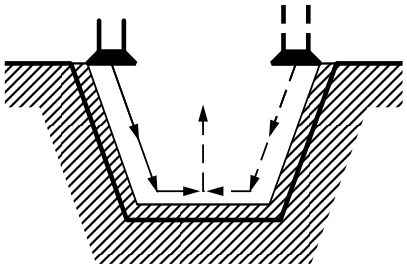
Cut perpendicular to the turning axis in one or more cuts. Before retraction from the cut 1 mm is retracted in Z direction from the second step on.


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 edge width.

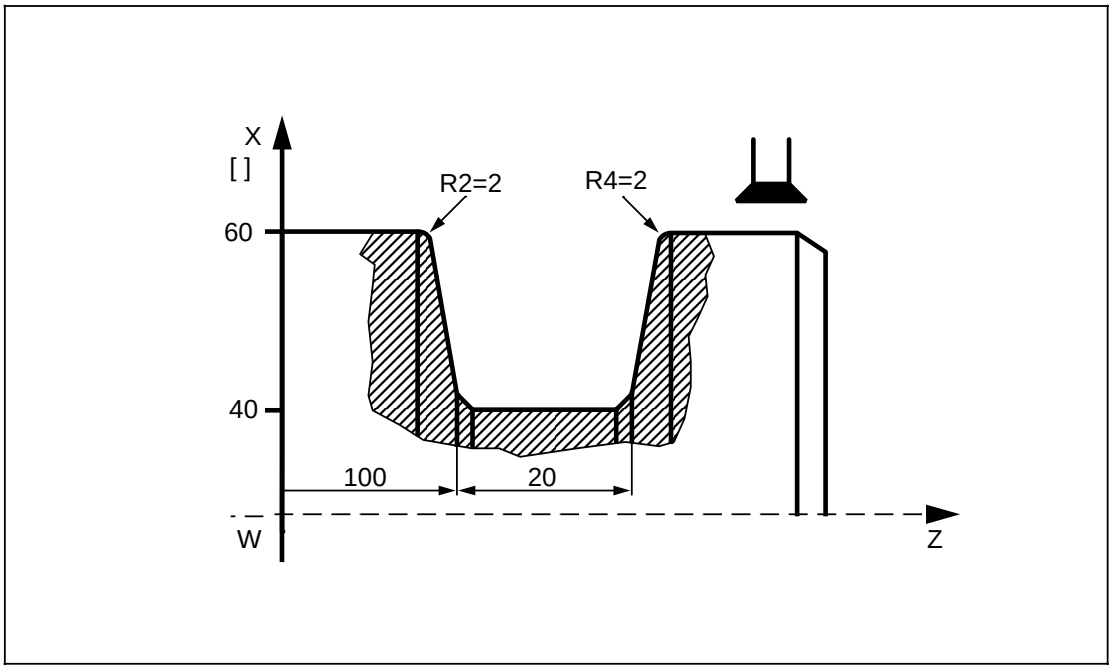

4. Finish cut

4th step

Cutting the finishing increment parallel to contour down to the centre of cut.

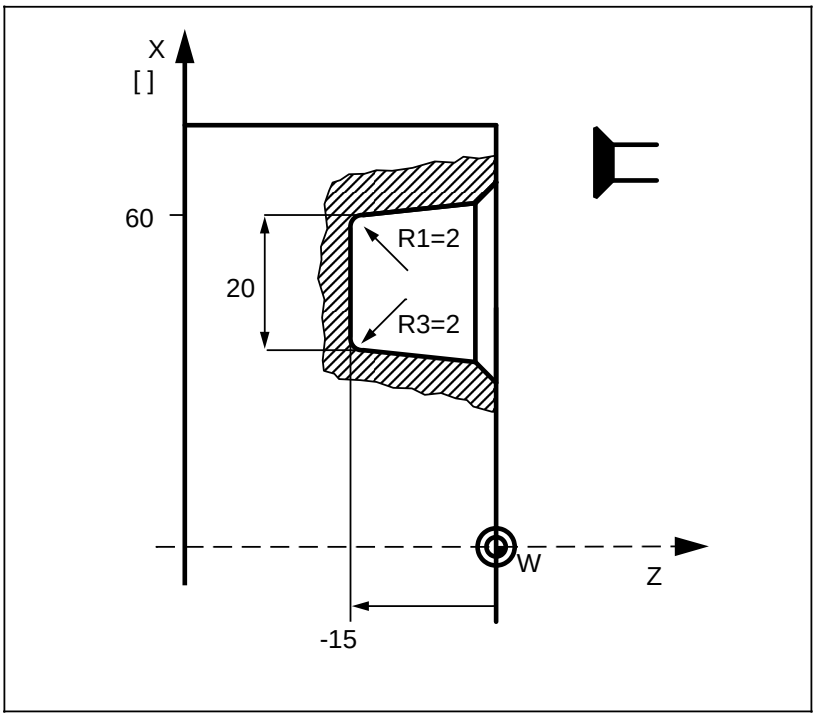
Example 1: "OUTSIDE LEFT" longitudinal cut selected by softkey

```
%1
N05  G95 G0 X65 Z105 D03 T03  S500  M04  LF          Select recess 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 recessing cycle
N40  G0  X100  Z200  LF
N45  M30  LF
```



Example 2: "RIGHT OUTSIDE" facing cut selected by softkey

```
%2
N05  G95 G0 X65 Z10 D03 T03 S500 M04  LF          Select recess 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 recessing cycle
N40  G0  X100  Z200  LF
N45  M30  LF
```

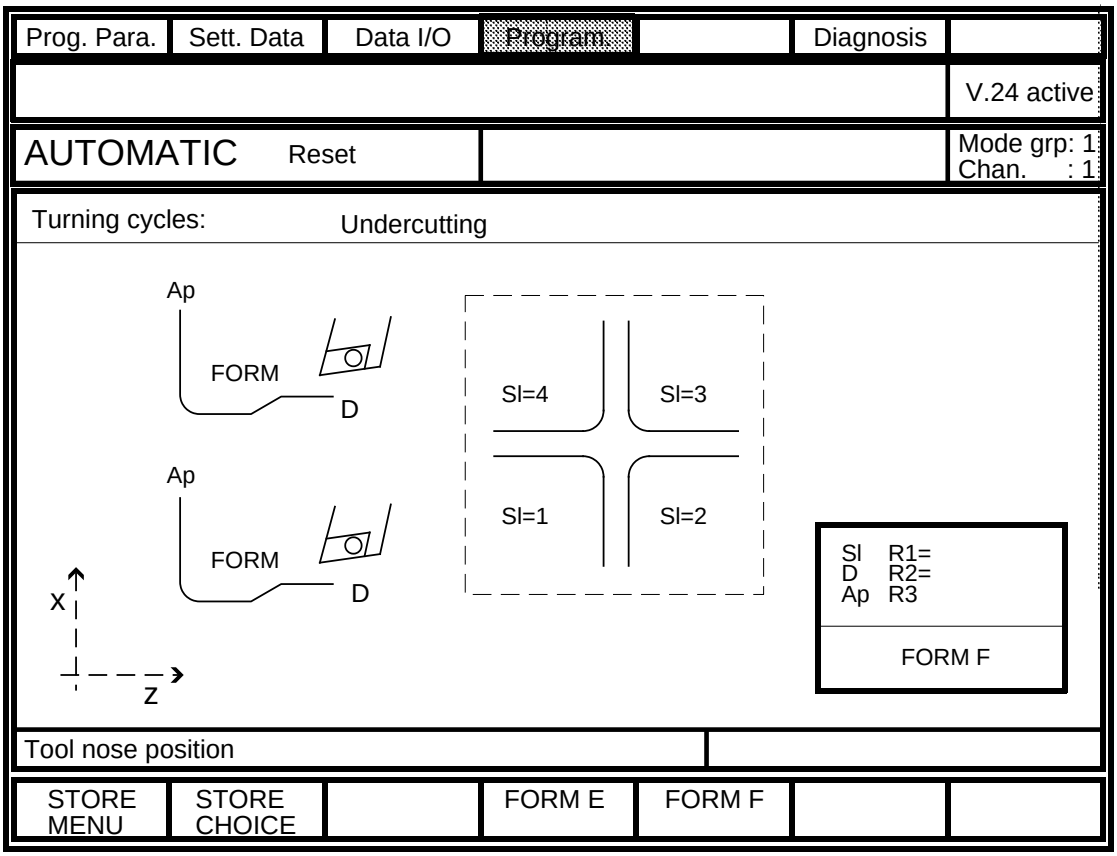


2.1.2 L94 Undercut cycle

Undercut cycle L94 permits form E and F undercuts for normal use according to DIN 509 with a machined part diameter of > 18 mm. The TNRC is automatically selected in the cycle.

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	Initial point of the contour in X
Ap	R03	Initial 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 of the tool nose position (1 to 4)

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

D R02: Initial 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: Initial 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/L96 Stock removal cycle with/without relief cut elements

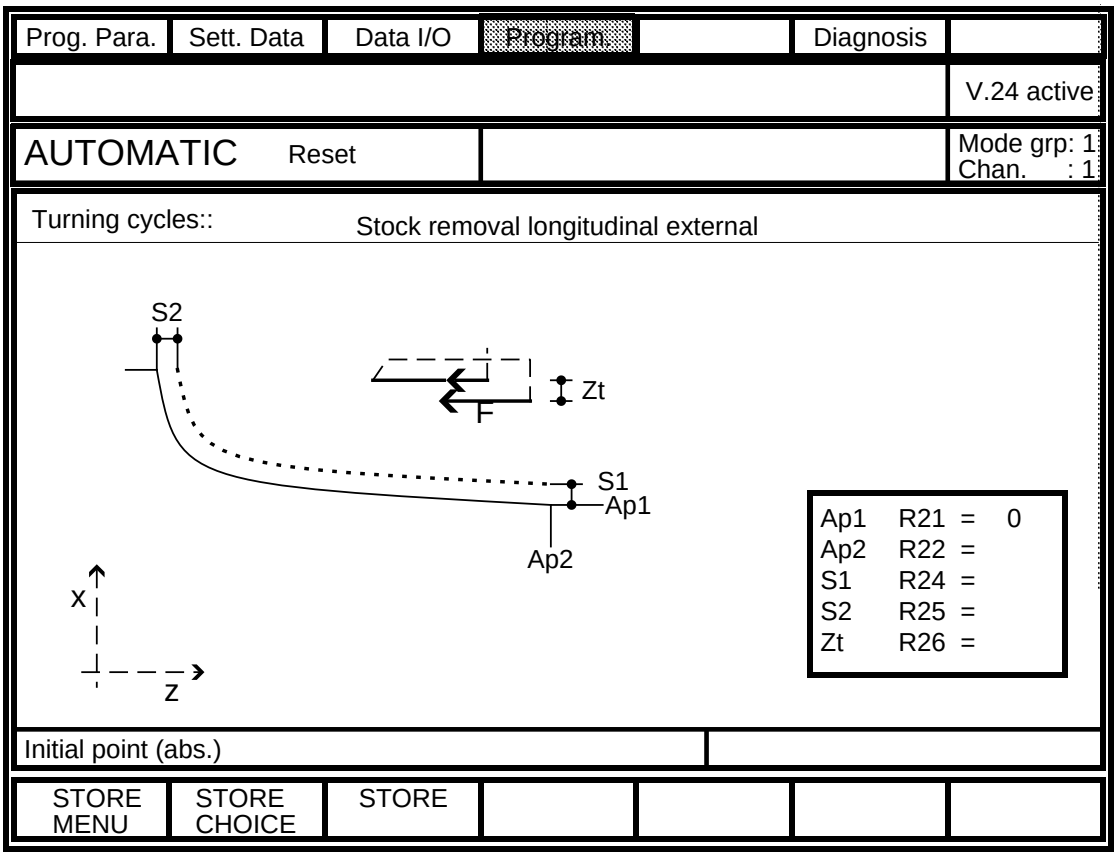
Stock removal cycles L95/L96 permit paraxial machining of a blank with the contour programmed in a subroutine. The machining cycles can be called from any position where there is no danger of collision. The control calculates the starting point automatically using the final contour description.

- 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	Initial point of the contour in X (absolute)
Ap2	R22	Initial 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	Cutter radius compensation (G41, G42)
F	R28	Feedrate (L95 only)
	R29	Type determination for roughing and finishing
	R30	Feed factor for infeed with relief cutting (L95 only)



R20: Subroutine number

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

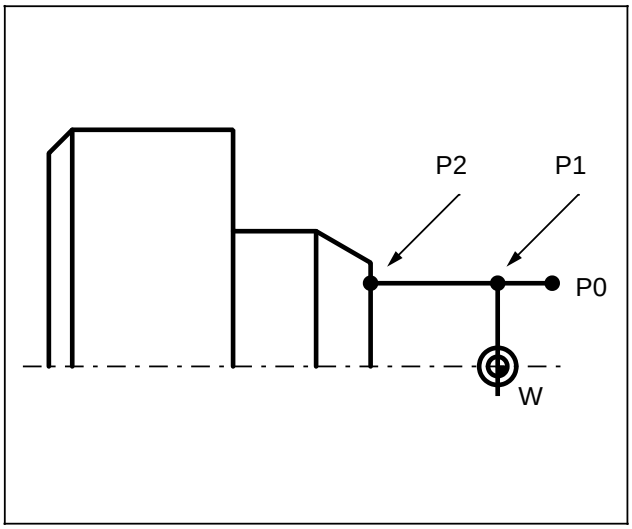
The subroutine can comprise any number of blocks, but at least two. A traverse path must be programmed in each block. The final contour can be defined with blueprint programming (compare Programming Guide). Skippable blocks are permitted in the contour.

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

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

Ap1 R21: Initial point of the contour in X
Ap2 R22: Initial point of the contour in Y

The parameters R21 and R22 must be supplied with the initial points of the contour. 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 initial points R21 and R22 must be shifted accordingly.



Example:

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

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

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

Cycle L95 approaches this start point as follows:

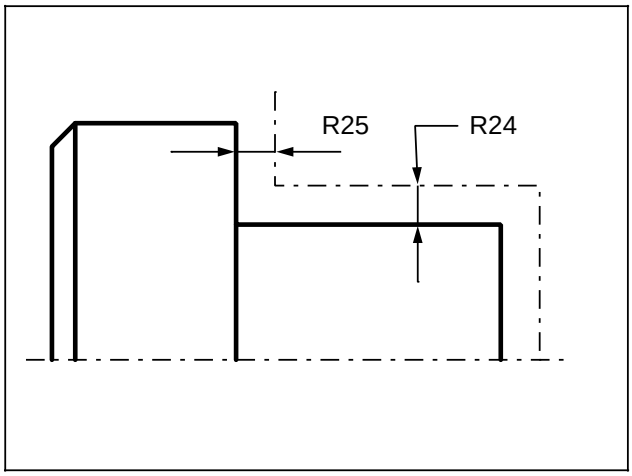
In the case of longitudinal machining, the start point in cycle L95 is approached with both axes simultaneously. This applies to the values 11**/**21**/**31**/**41**/**13**/**23**/**33**/**43 of parameter R29.

In the case of face machining too, the start point is always approached with both axes simultaneously, except for finishing (R29=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 approaching the start point for face machining:

R29	R24 (finishing allowance in X)	R25 (finishing allowance in Z)	Approach strategy
12 and 14	any	any	X**/**Z together
22 and 24	R24=0	R25=0	X**/**Z together
22 and 24	R24=0	R25 is not equal 0	first Z, then X
22 and 24	R24 0	R25 = 0	first Z, then X
22 and 24	R24 0	R25 is not equal 0	first Z, then X
32 and 34	any	any	X**/**Z together
42 and 44	any	any	X**/**Z together

S1 R24: Finishing allowance in X (incremental)
S2 R25: Finishing allowance in Z (incremental)



The contour is shifted by the amount of finishing allowance entered (R24, R25). In the "roughing" machining mode, roughing is carried out down to this finishing allowance .
In the "finishing" machining mode, a single cut is taken parallel to the contour down to the finishing increment. Roughing depth R26 depends on the machining allowance (e.g. cast iron part) and need not to be programmed.
If R24**/**R25 are given as 0, the tool travels along the contour final dimensions.

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

During roughing the cycle checks whether the present depth of cut is less than twice the roughing cut depth R26. If the present depth of cut would be less than twice the roughing cut depth, then the following applies to the last two cuts:

R26 roughing cut depth = present depth of cut/2.

R27: Cutter radius compensation (G41, G42)

The cycle selects and deselects the cutter 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 relief cut

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 depth is derived from R28 · 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.

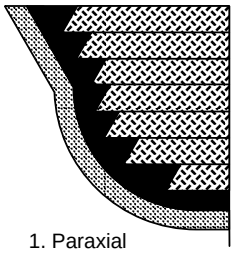
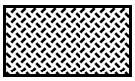
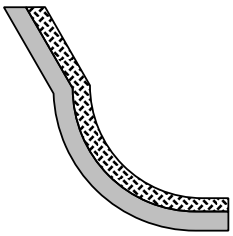
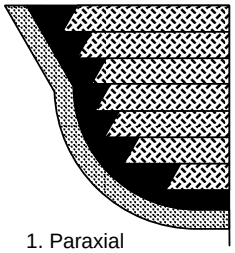
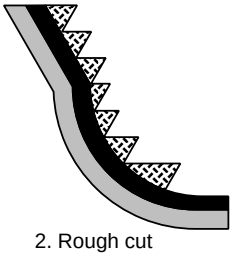
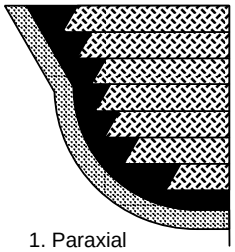
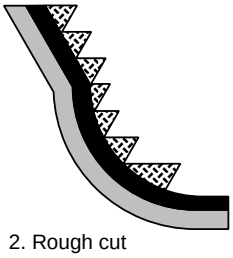
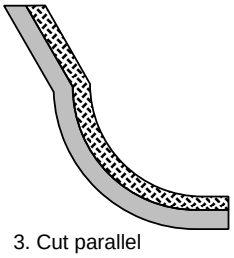
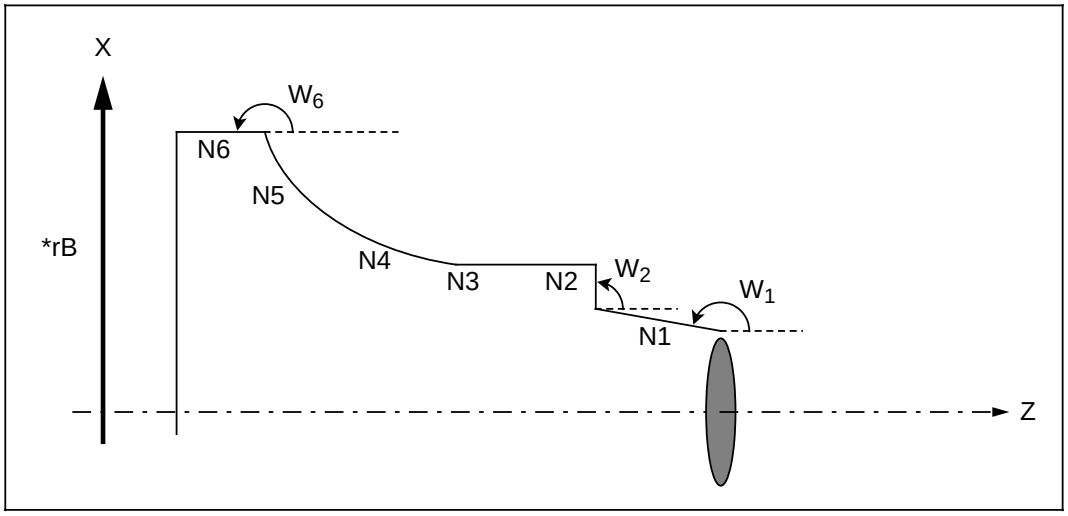
 1. Paraxial roughing	 Removed material		R29 = 11 longitud. (Z) external R29 = 12 face (X) external R29 = 13 longitud. (Z) internal R29 = 14 face (X) internal
 Finish cut parallel to contour			R29 = 21 longitud. (Z) external R29 = 22 face (X) external R29 = 23 longitud. (Z) internal R29 = 24 face (X) internal *)
 1. Paraxial roughing  2. Rough cut parallel to contour			R29 = 31 longitud. (Z) external R29 = 32 face (X) external R29 = 33 longitud. (Z) internal R29 = 34 face (X) internal *)
 1. Paraxial roughing  2. Rough cut parallel to contour  3. Cut parallel to contour			R29 = 41 longitud. (Z) external R29 = 42 face (X) external R29 = 43 longitud. (Z) internal R29 = 44 face (X) internal *)

Table 2.1 Explanation of machining types R29

*) In these cases, the cycle activates the cutter radius compensation (CRC) 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 CRC itself. CRC 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

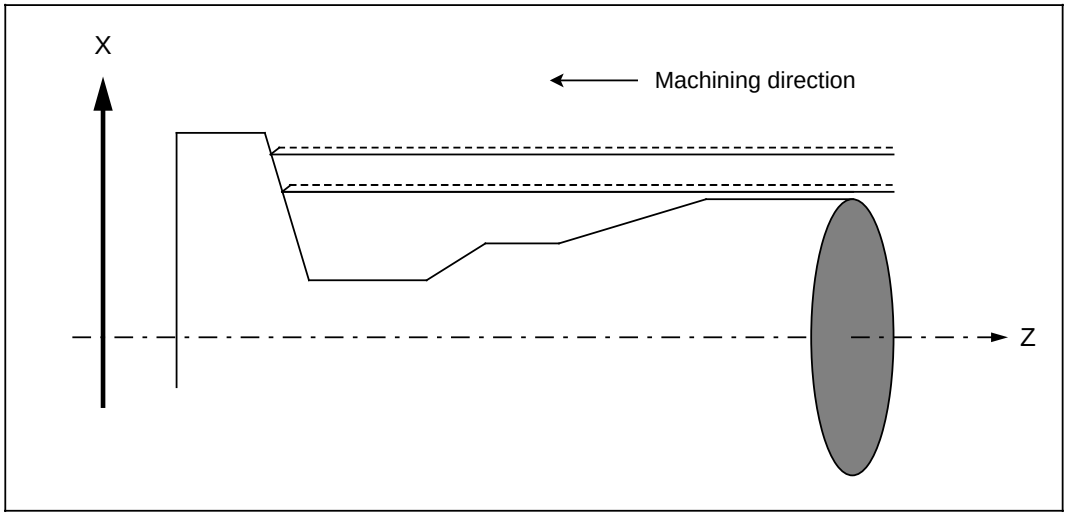
- With cycles L95 and/or L96 it is possible to machine rising contours. A contour element containing the maximum contour diameter must be programmed at the end of the contour definition, where the final diameter must be larger than the initial diameter. "Rising" means that all the contour sections programmed 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 a rising contour. The contour section N1 includes 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.

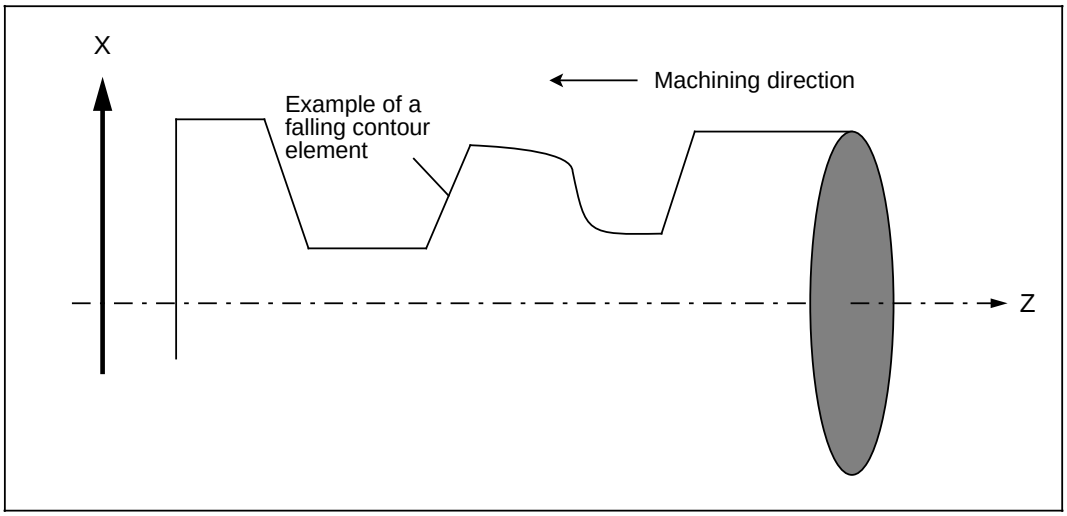
1 relief cut element



Several relief cut elements can follow each other in succession.

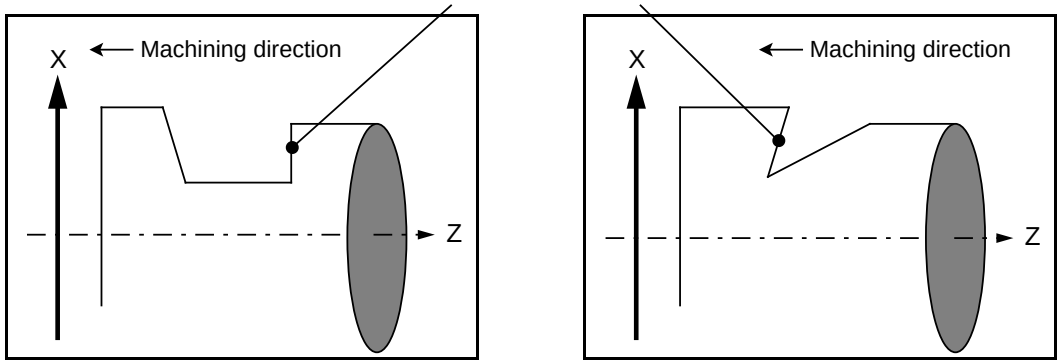
Relief cut elements contain ”falling” contour sections, which as in the ”rising” contour sections, have an angle of between 0° and 90°, $0 < W_i < 90°$, to the positive horizontal axis.

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 initial point and end 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).

Example 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 falling contour elements (see example), as this would cause a collision when the tool is withdrawn after roughing.

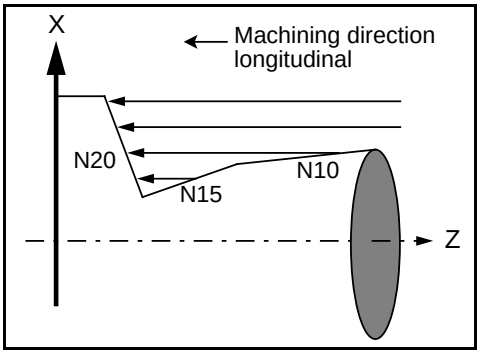
- A defined final contour can be roughed paraxially either parallel to the horizontal axis (longitudinal machining) or to the vertical axis (face machining). This is defined in parameter R29.

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

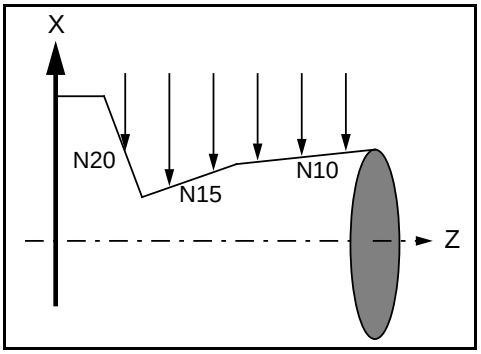
  To ensure that the correct type of machining is used for a workpiece for a defined contour, the following explains which contour types should be machined longitudinally, which contour types should be face machined and which types can be machined in both ways. The contours are therefore described as either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

  A longitudinal contour is a contour on which all intersection points in longitudinal machining lie on rising contour elements which are parallel to the horizontal axis.

Example of longitudinal contour



The intersection points ascertained for longitudinal machining are located in N20 only, i.e. in a rising contour element

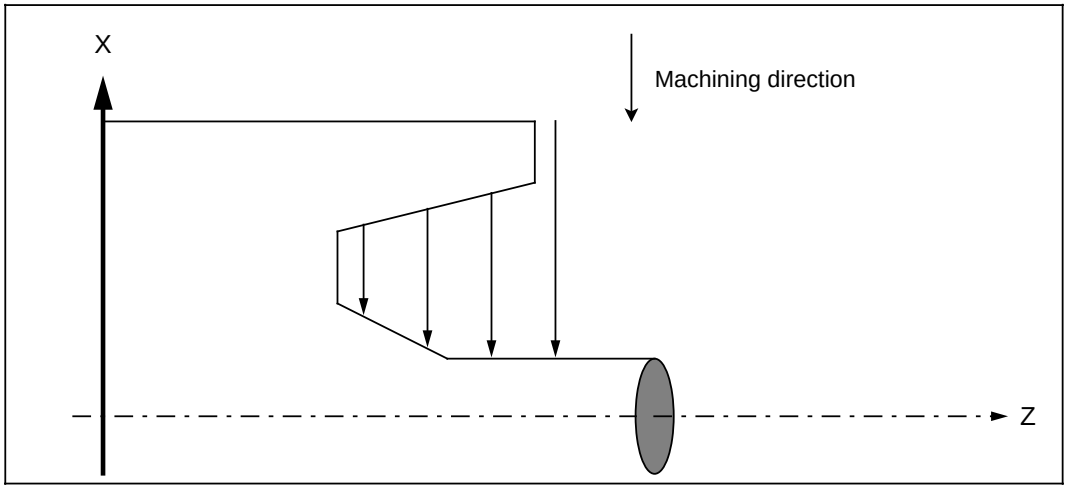


By comparison:

Face machining would also produce points of intersection on "falling" contour elements in N10 and N15

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

Example of a face contour



In this context, 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 an odd number) and face contours 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 horizonal axis is contained in any ”falling” path sections.

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

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

  Finishing allowance in X c · finishing allowance in Z

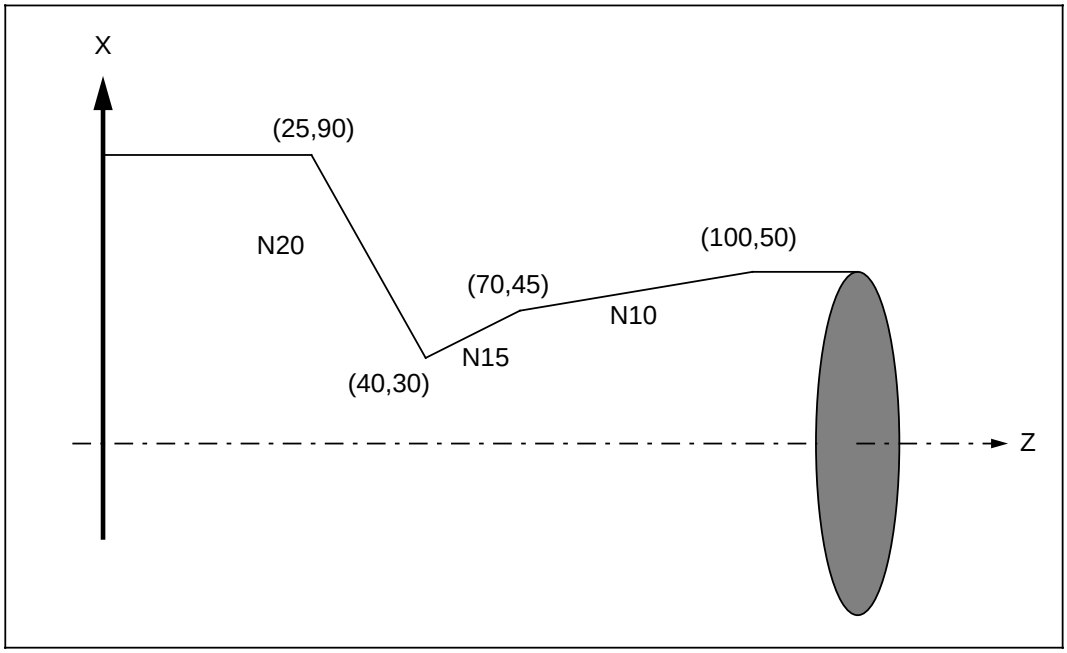
  with

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

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

Example:

The following contour (which is actually a longitudinal contour) is to be face machined.



When selecting the finishing allowance the following applies:

The contour contains two falling 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 finishing allowance in Z of 3.5 mm is selected the finishing allowance in X must be at least 1.75 mm or conversely, with a finishing allowance in X of 0.5 mm, the finishing allowance in Z must not be greater than 1 mm.

This relationship between the finishing 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 longitudinal external" machining type selected via softkey

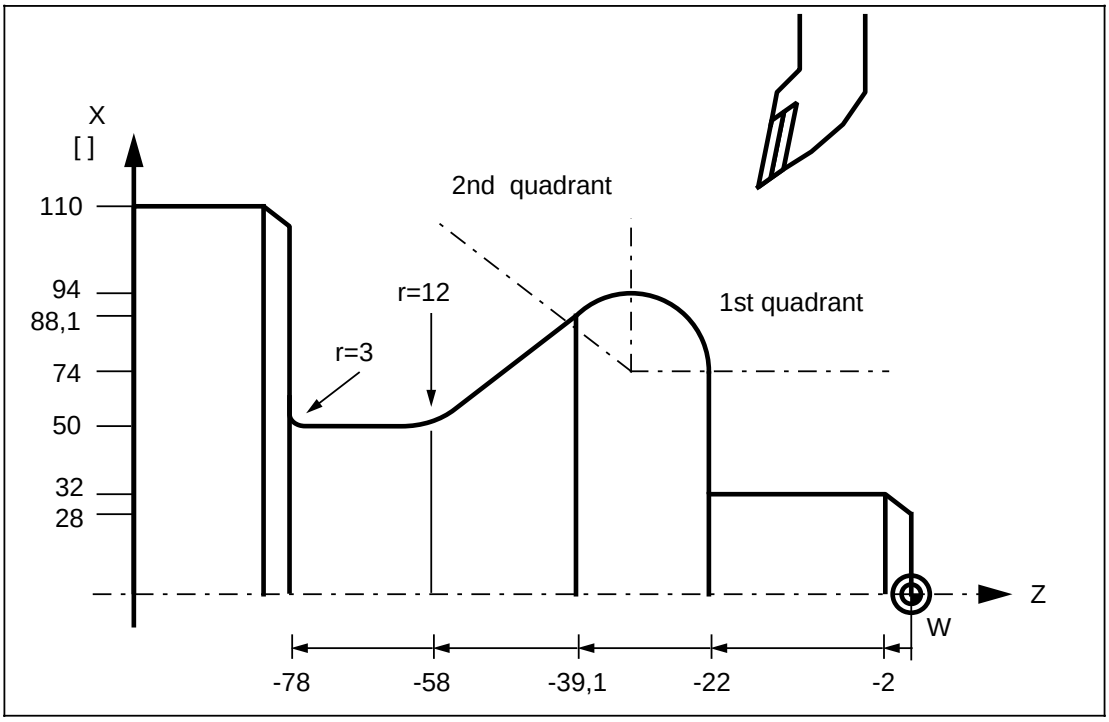
```
%1
N05  G95  G0  X120  Z10  D01 T01 S1000 M04  LF          Select stock removal
N10  R20=105  R21=28  R22=0  R24=1  LF                  position
     R25=1  R26=5  R27=42 R28=.2  LF
     R29=41  R30=.5  L95  P1  LF                        Call stock removal cycle
N20  G0  X200  Z200  LF          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

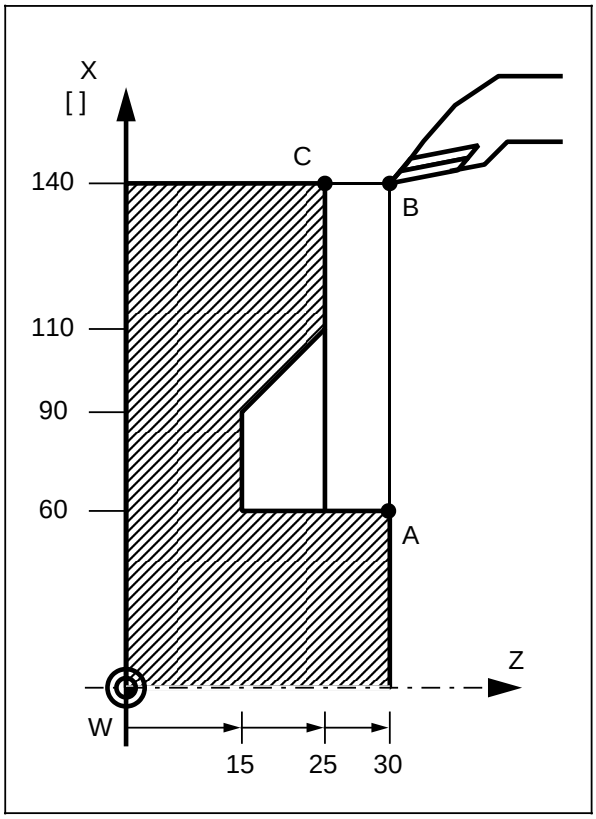
Relief cut element: this radius extends over more than one quadrant and must therefore be divided into two steps (compare N65 and N70).



Example 2: "Finishing facing external" machining type selected via softkey

```
%2
N05  G96  G0  X80 Z40 D01 T01 S2000 M04  LF          Select stock removal position
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                     Call stock removal cycle L95
N20  G0  X200  Z200  LF                                  with relief cut element
N25  M30  LF

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



Corner point B also represents the direction-change point in the finishing cycle.

The cycle calculates this point from points A and C.

2.1.4 L97 Thread cutting cycle

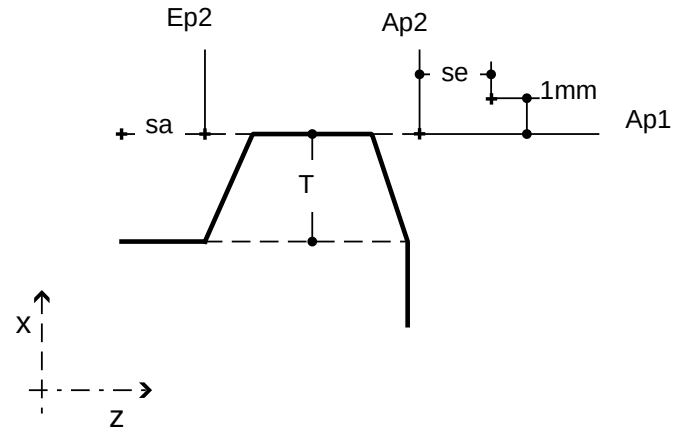
Using this cycle, external threads, internal threads, taper threads and transversal threads can be cut. Infeed is automatic and is degressively quadratic, the cut cross-section thus remains constant.

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	Initial point of thread in X (absolute)
Ap2	R22	Initial point of thread in Z (absolute)
	R23	Number of idle passes
T	R24	Thread depth (incremental), sign required to define internal or external thread: + = internal thread / - = external thread, transversal thread
S	R25	Finishing allowance (incremental)
se	R26	Run-in 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)

Prog. Para.	Sett.-Data	Data I/O	Program		Diagnosis	
						V.24active
AUTOMATIC Reset						Mode grp: 1 Chan. : 1

Turning cycles: Thread cutting external



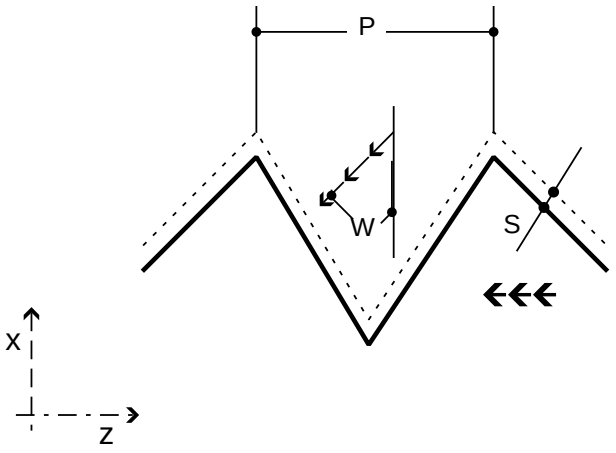
Ap1	R21 = 0
Ap2	R22 = 0
T	R24 = 0
se	R26 = 0
sa	R27 = 0
Ep2	R32 = 0

Initial point(abs.)

						PITCH

Prog. Para.	Sett.-Data	Data I/O	Program		Diagnosis	
						V.24 active
AUTOMATIC Reset						Mode grp: 1 Chan. : 1

Turning cycles: Thread cutting external



P	R20 = 0
S	R25 = 0
W	R29 = 0
	R23 = 0
	R28 = 0

Thread pitch

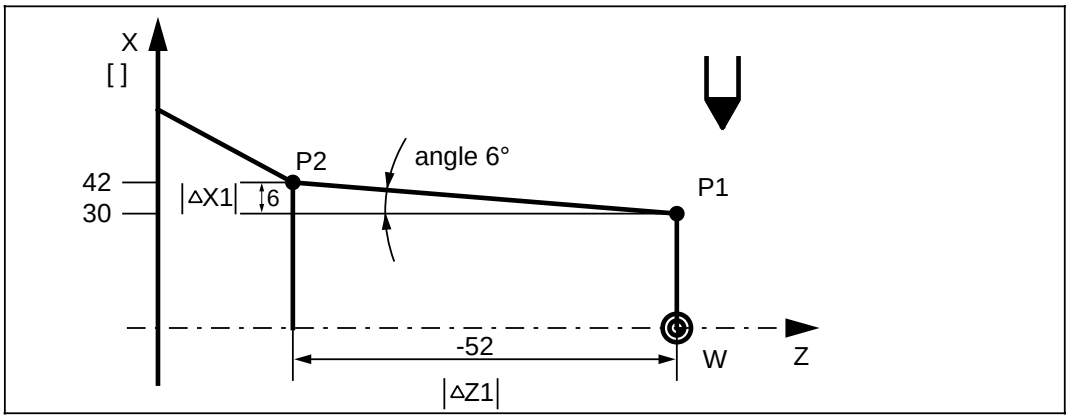
STORE MENU	STORE CHOICE	STORE				

Thread cutting cycles: differentiation between transversal and longitudinal threads

Both longitudinal and transversal threads are possible with L97 and L98. The differentiation depends on the angle resulting from the initial point of the thread (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).

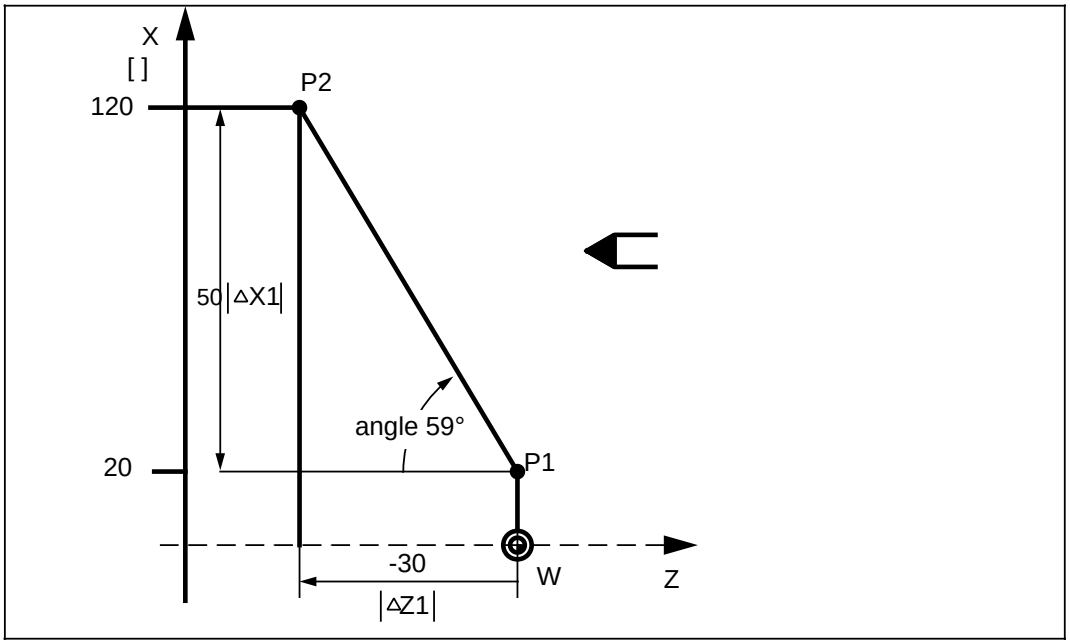
- Longitudinal thread
  The amount | Z1 | must be greater than or equal to the amount | X1 |, i.e. the angle is less than or equal to 45°.

L99			L97		
Ap1	R11=30	Initial point in X	Ap1	R21=30	Initial point in X
Zp1	R12=42	First intermediate point in X	Ap2	R22=0	Initial point in Z
Ap2	R21=0	Initial 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



- Transversal thread
  The amount | Z1 | must be less than the amount | X1 |, i.e. the angle is greater than 45°.

L99			L97		
Ap1	R11=20	Initial point in X	Ap1	R21=20	Initial point in X
Zp1	R12=120	First intermediate point in X	Ap2	R22=0	Initial point in Z
Ap2	R21=0	Initial 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



P R20: Thread pitch

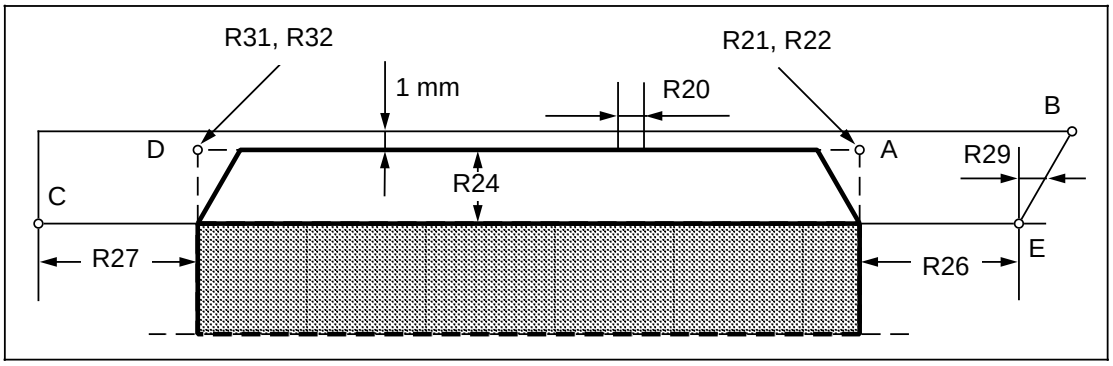
The thread pitch must be entered as a paraxial value without sign.

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

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

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

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



As from software version 5.6, SINUMERIK 840C is provided with a machine-protecting acceleration ramp of the drilling axis for threading. The setpoint speed of the infeed axis is no longer immediately coupled to the actual speed of the spindle, but is accelerated at a ramp. The material entering point, at which the setpoint speed is to be reached at the latest, is therefore programmed additionally in the thread block (see SINUMERIK 840C Programming Guide, Section 8.8 Threading (G33/G34/G35/G36/G63).

For SINUMERIK 840C, the cycle L97/L99 has been adapted to this new function in UMS 54.

The thread block still begins at the starting point, which is displaced to the front by the run-in path. The original start point (A) is defined by the cycle as material entering point.

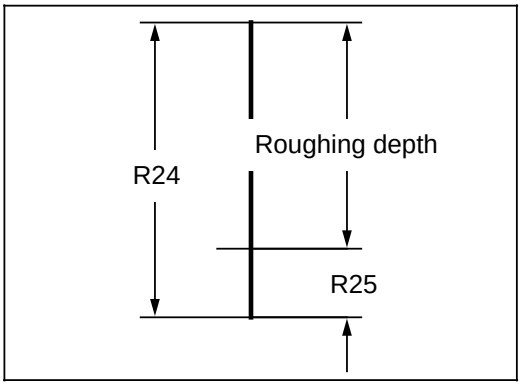
R23: Number of idle passes

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 external, internal or transversal thread. (+ internal thread, external thread, transversal thread).

S R25: Finishing cut allowance (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.

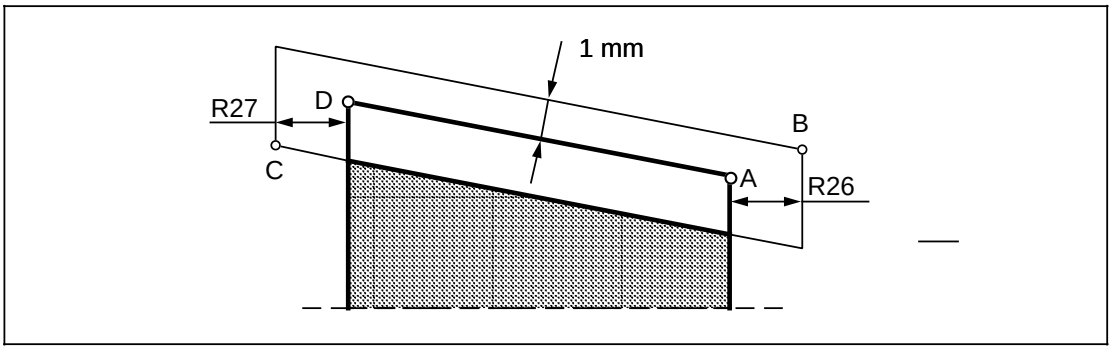
After the roughing cuts have been completed, a finishing cut is made and then the idle passes programmed under R23 are executed.

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

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

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

In the case of taper threads, the control calculates the run-in and run-out paths in relationship to the taper and determines the corner points B and C.



R28: Number of roughing cuts

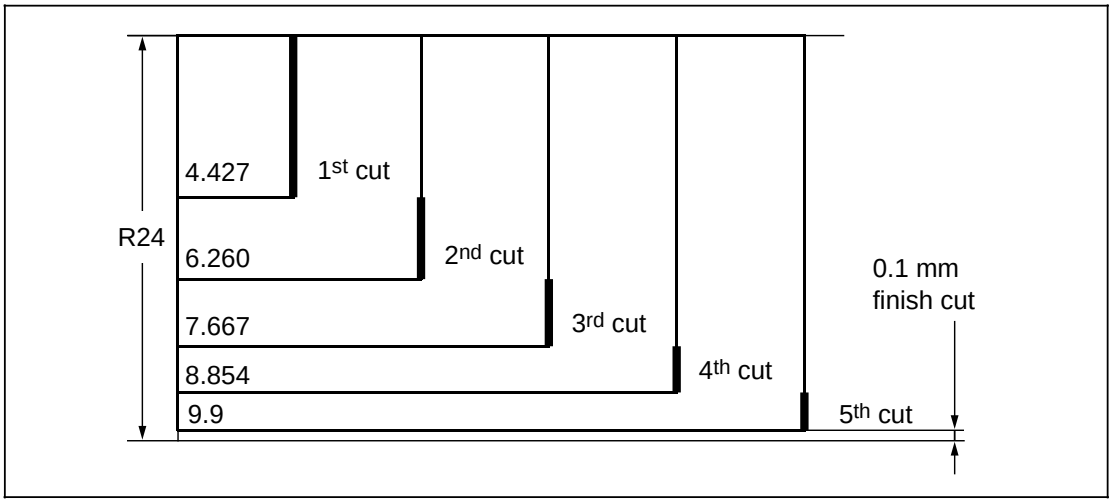
The parameter value determines the number of thread roughing cuts. The control automatically calculates the individual infeed depths for constant cut cross-section. This ensures that the cut pressure from the first to the last roughing cut remains the same.

The depth of the current cut depth t is calculated by the following formula:

$$t = \frac{t}{\sqrt{R28}} \cdot \sqrt{i}$$

t=R24 - R25
i=current cut

Example:

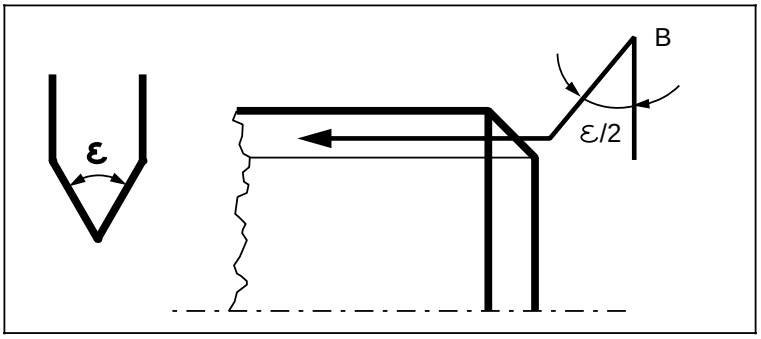


Thread depth:	R24=10 mm
Number of roughing cuts:	R28=5
Finishing allowance:	R25=0.1 mm

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.



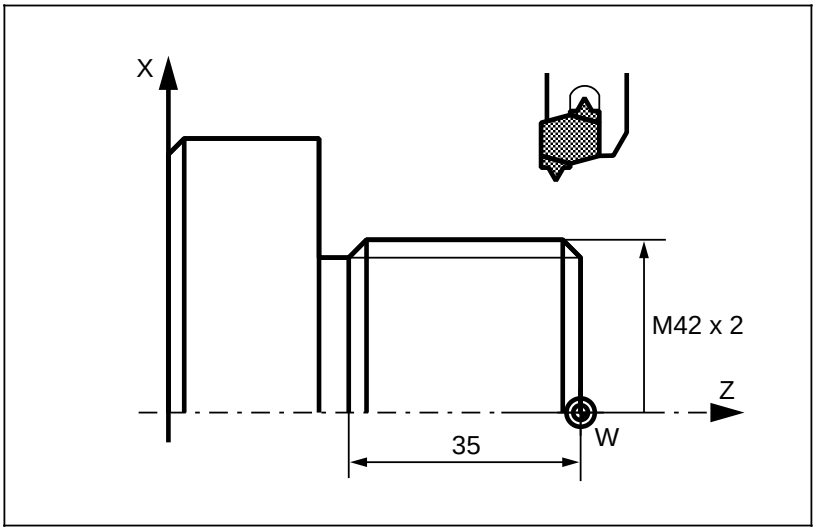
Metric thread 60°
/2=30°
R29=30

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
```



2.1.5 L99 Chaining of threads (four-point thread cutting cycle)

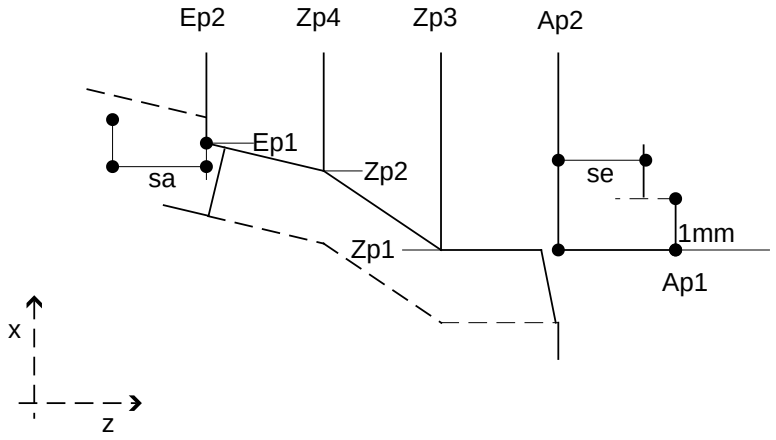
Cycle L99 permits several threads in sequence to be cut, including transversal threads, with different pitches.

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	Initial 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	Initial 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	End point of thread in Z	(absolute)
S	R25	Finishing allowance	(incremental)
se	R26	Run-in path	(incremental)
sa	R27	Run-out path	(incremental)
	R28	Number of roughing cuts	
W	R29	Infeed angle	
	R35	Number of idle passes	
T	R36	Thread depth with sign, depending on whether the thread is internal or external: + = internal thread - = external thread, transversal thread	(incremental)
P1	R41	Thread pitch 1	
P2	R42	Thread pitch 2	
P3	R43	Thread pitch 3	

Prog. Para.	Sett. Data.	Data I/O	Program		Diagnosis	
						V.24 active
AUTOMATIC Reset						Mode grp: 1 Chan. : 1

Turning cycles: Thread - Thread



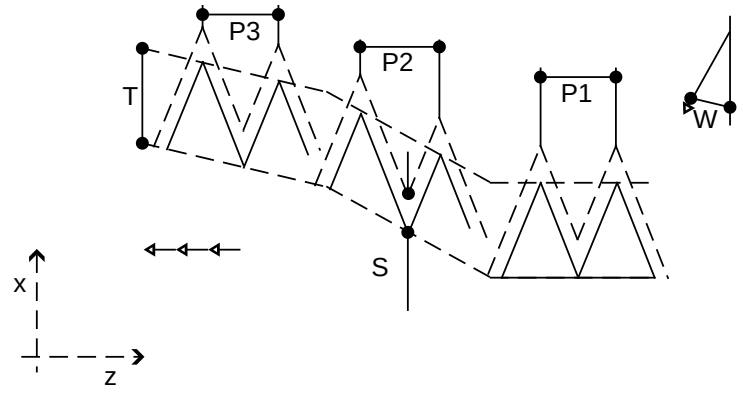
Ap1	R11=
Zp1	R12=
Zp2	R13=
Ep1	R14=
Ap2	R21=
Zp3	R22=
Zp4	R23=
Ep2	R24=
se	R26=
sa	R27=

Initial point (abs.)

						Pitch

Prog. Para.	Sett. Data	Data I/O	Program		Diagnose	
						V.24 active
AUTOMATIC Reset						Mode grp: 1 Chan. : 1

Turning cycles: Thread - Thread

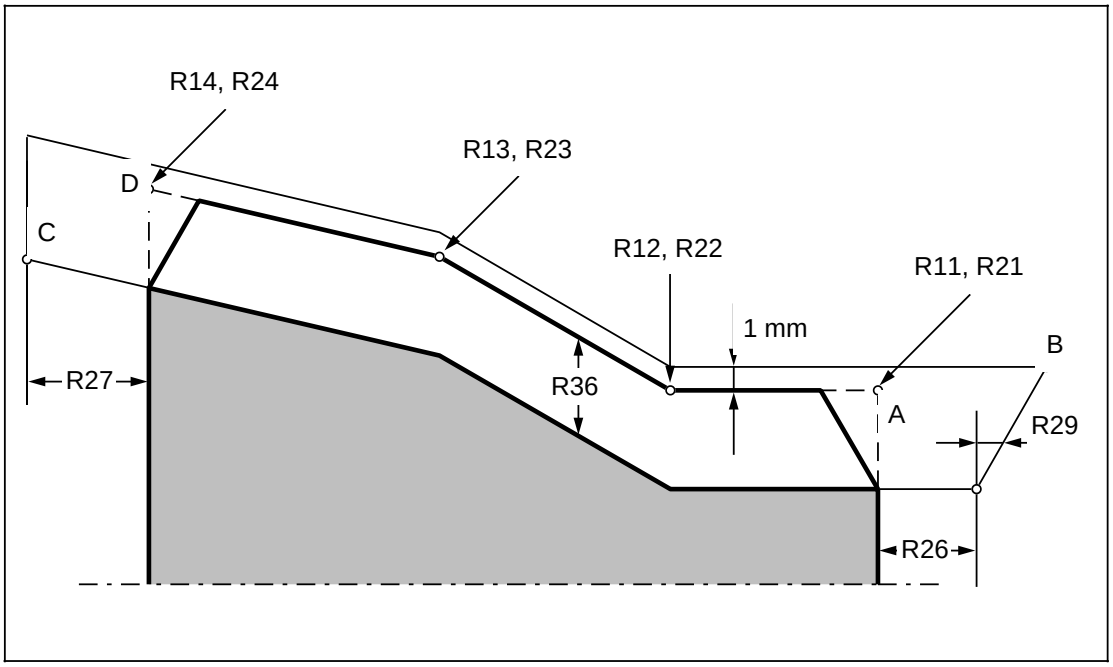


S	R25=
W	R29=
T	R36=
P1	R41=
P2	R42=
P3	R43=
	R28=
	R35=

Finishing allowance

STORE MENU	STORE CHOICE					

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



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

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

As from software version 5.6, SINUMERIK 840C is provided with a machine-protecting acceleration ramp of the drilling axis for threading. The setpoint speed of the infeed axis is no longer immediately coupled to the actual speed of the spindle, but is accelerated at a ramp. The material entering point, at which the setpoint speed is to be reached at the latest, is therefore programmed additionally in the thread block (see SINUMERIK 840C Programming Guide, Section 8.8 Threading (G33**/**G34**/**G35**/**G36**/**G63).

For SINUMERIK 840C, the cycle L99 has been adapted to this new function in UMS 54.

The thread block still begins at the starting point, which is displaced to the front by the run-in path. The original start point (A) is defined by the cycle as material entering point.

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.

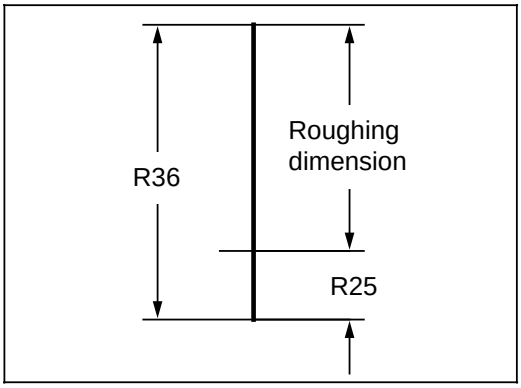
Zp2 R13: Second intermediate point of thread in X (absolute)
Zp4 R23: Second intermediate point of thread in Z (absolute)

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: End point of thread in X (absolute)
Ep2 R24: End point of thread 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.

S R25: Finishing allowance (incremental)



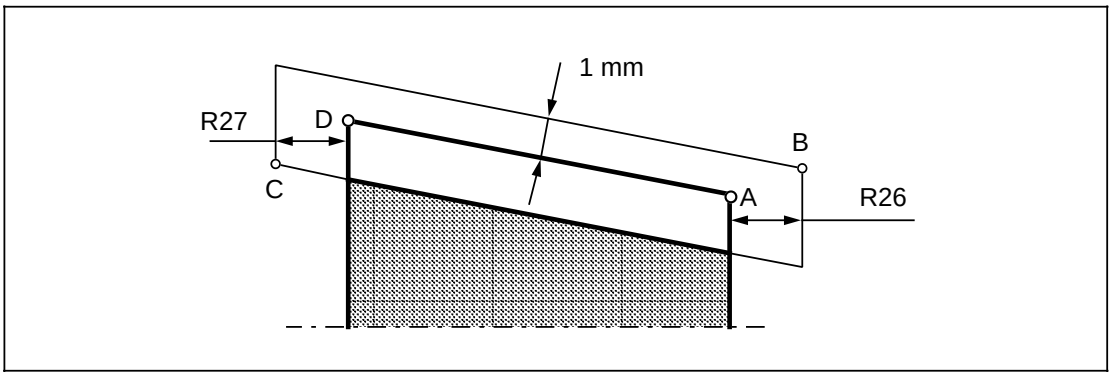
If a finishing allowance is programmed in R25, it is subtracted from the thread depth R36 and the remaining value is divided into roughing cuts.

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

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

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

In the case of a taper thread, the control converts the run-in and run-out paths into the taper ratio and defines corner points B and C.



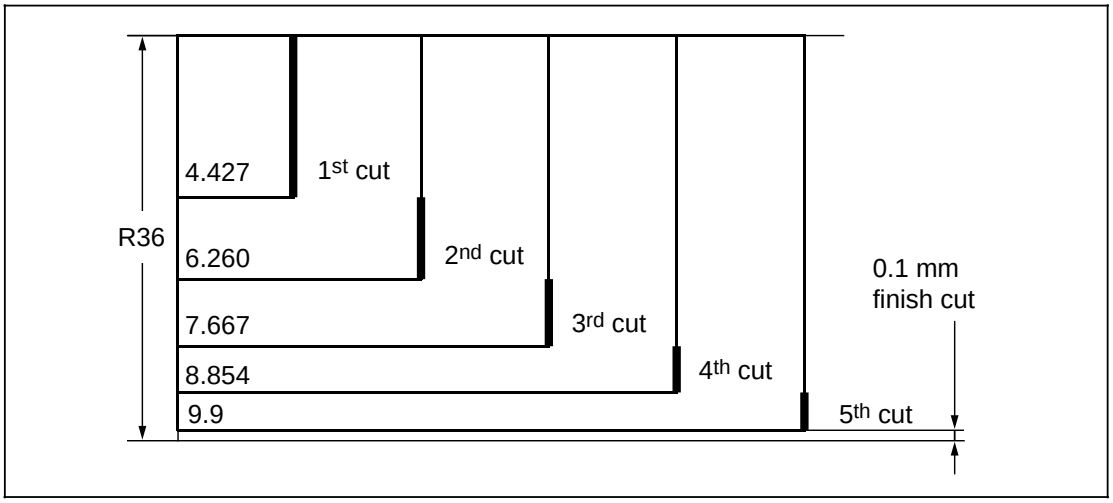
R28: Number of roughing cuts

The parameter value defines the number of thread roughing cuts. The control automatically calculates the individual infeed depths at constant cut 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 by the following formula:

$$t = \frac{t}{\sqrt{R28}} \cdot \sqrt{i}$$

t=R24 - R25
i=current cut

Example:

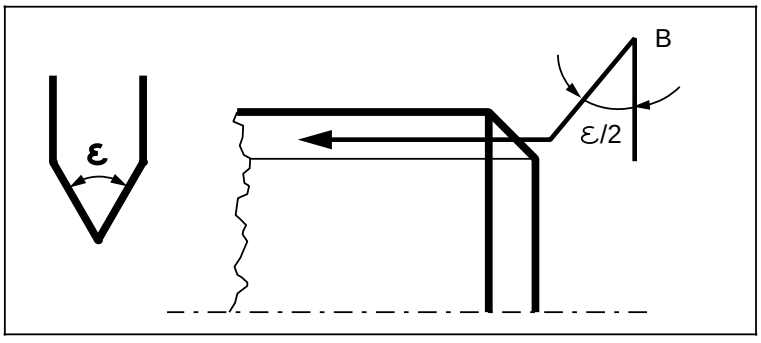


Thread: R36=10 mm
Number of roughing cuts: R28=5
Finishing allowance: R25=0.1 mm

W R29: Infeed angle for 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°
ε/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)

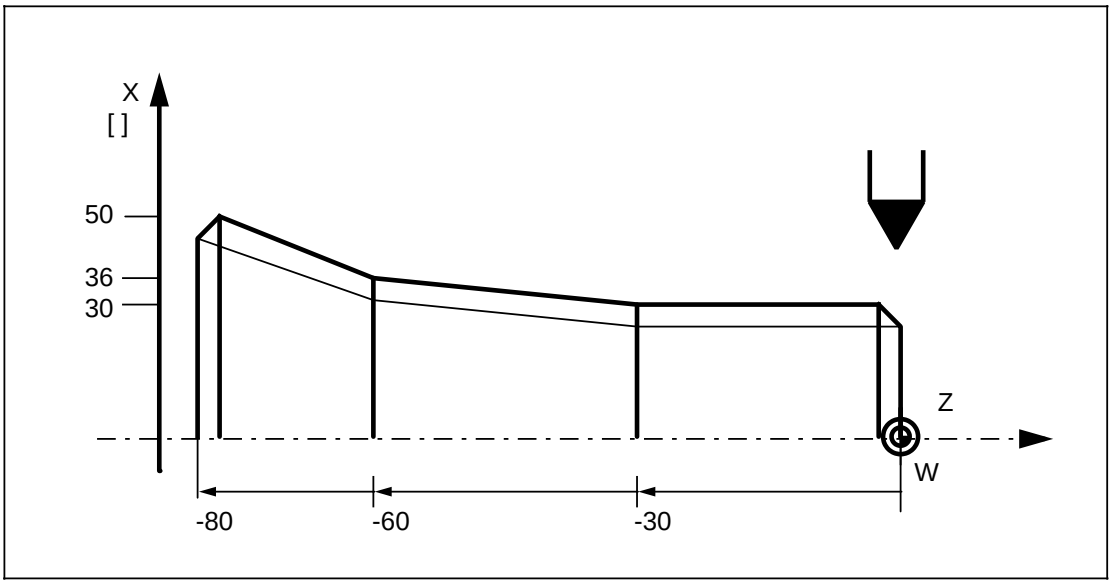
The thread depth is entered in parameter R36. The sign determines the infeed direction, i.e. whether it is an external or internal thread. (+ = internal thread, - = external thread, transversal thread).

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
N15  G0     X200  Z200  LF                                   Call thread cutting cycle
N20  M30  LF
```

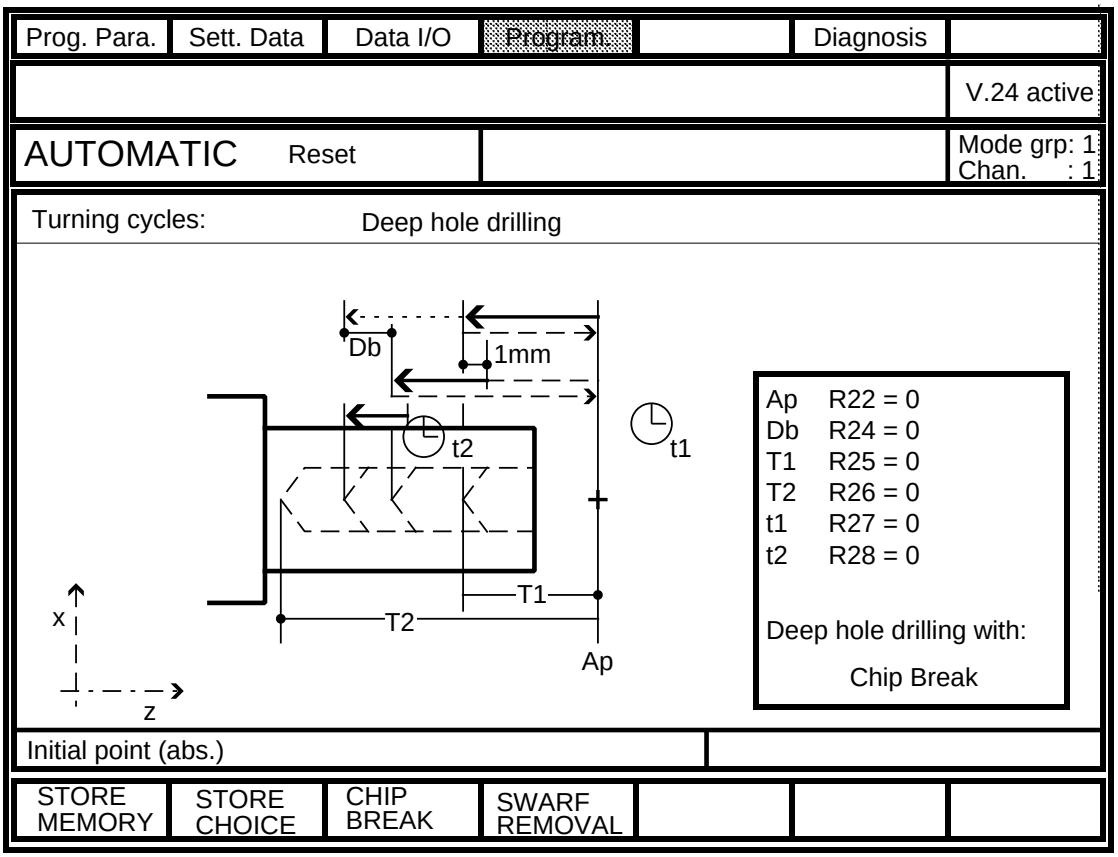


2.1.6 L98 Deep hole drilling cycle

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 is retracted by 1 mm for chip breaking each time the drill depth is reached.

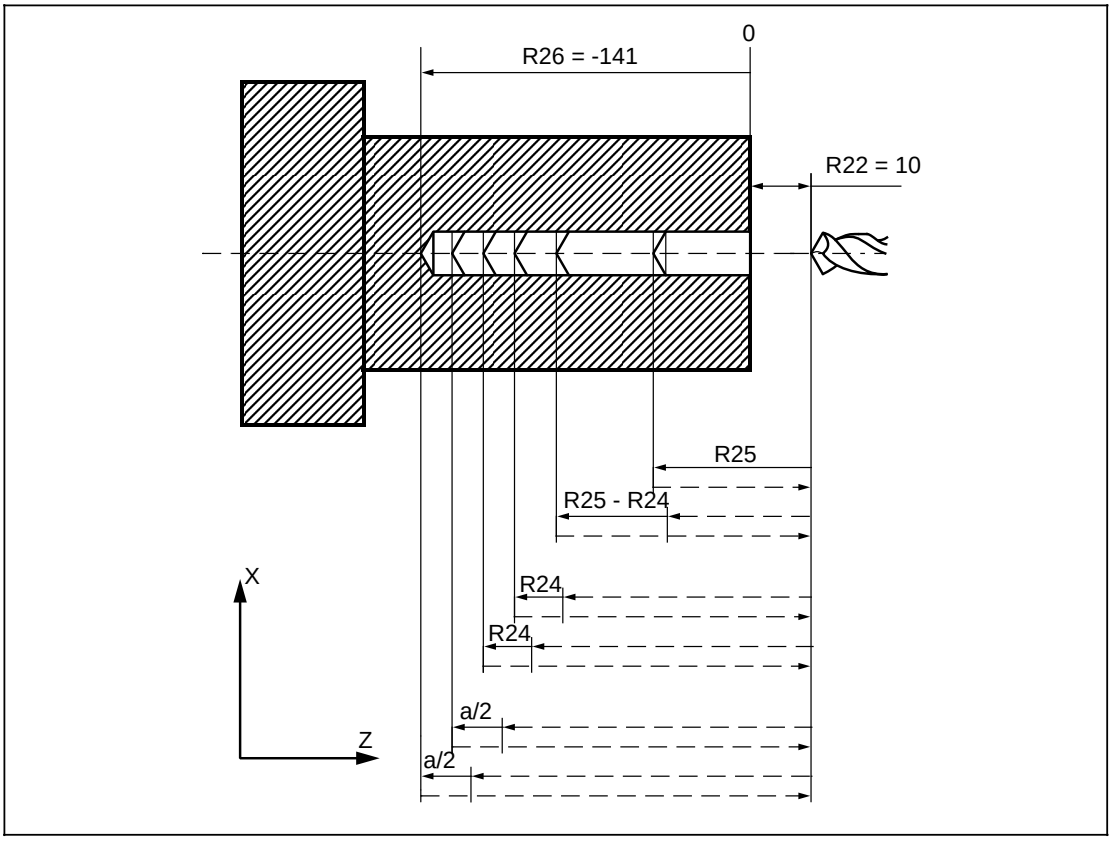
If R11 is assigned with 1, the drill travels to the reference plane for chip removal each time the drill depth isreached.

Ap R22: Initial point in Z (absolute)

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

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

```
%98
N05  G95 G1 XO 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.

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.

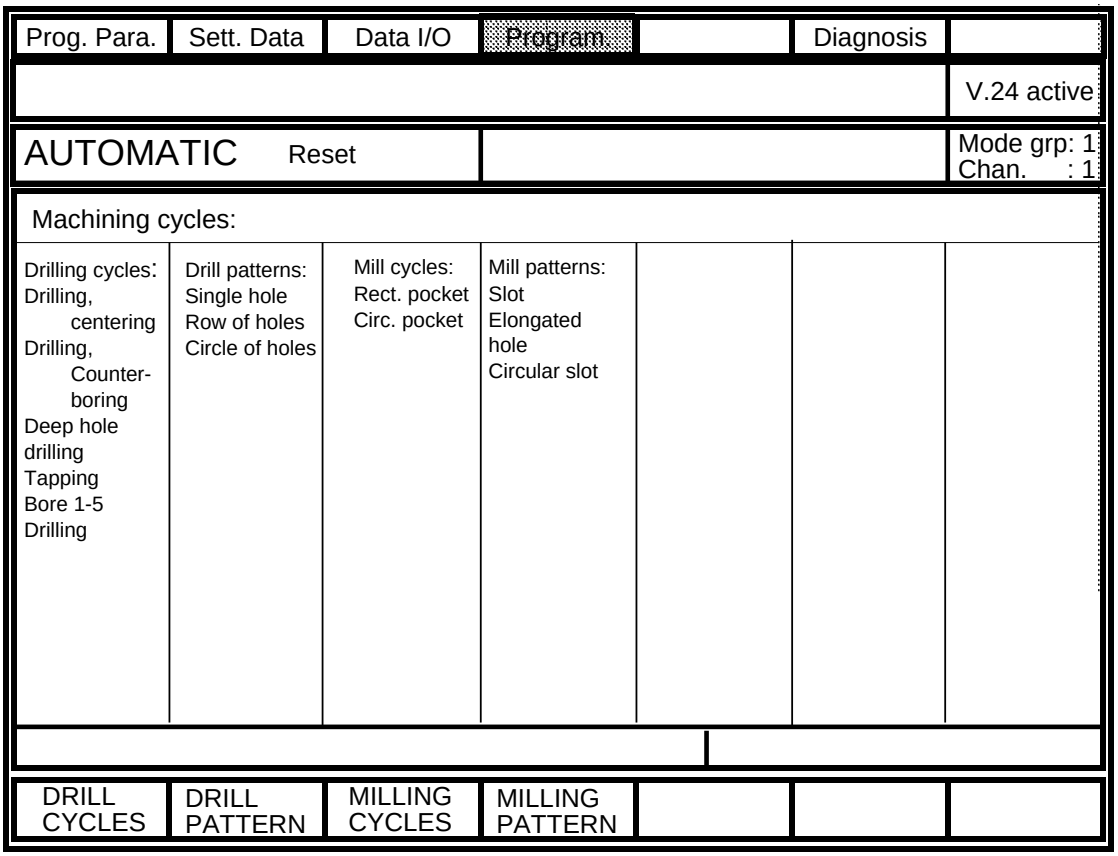
2.2 Machining cycle for drilling and milling

(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
L82 Drilling, counterboring
L83 Deep hole drilling
L84 Tapping (with or without encoder)
L85 Bore 1
L86 Bore 2
L87 Bore 3
L88 Bore 4
L89 Bore 5

L900 Hole circle drilling pattern
L905 Single hole drilling pattern
L906 Row of holes drilling pattern
L901 Slot milling pattern
L902 Elongated hole milling pattern
L903 Mill rectangular pocket
L904 Circular slot milling pattern
L930 Mill circular pocket



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 plane, 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 (see 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
                                                       and automatic call L81
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.

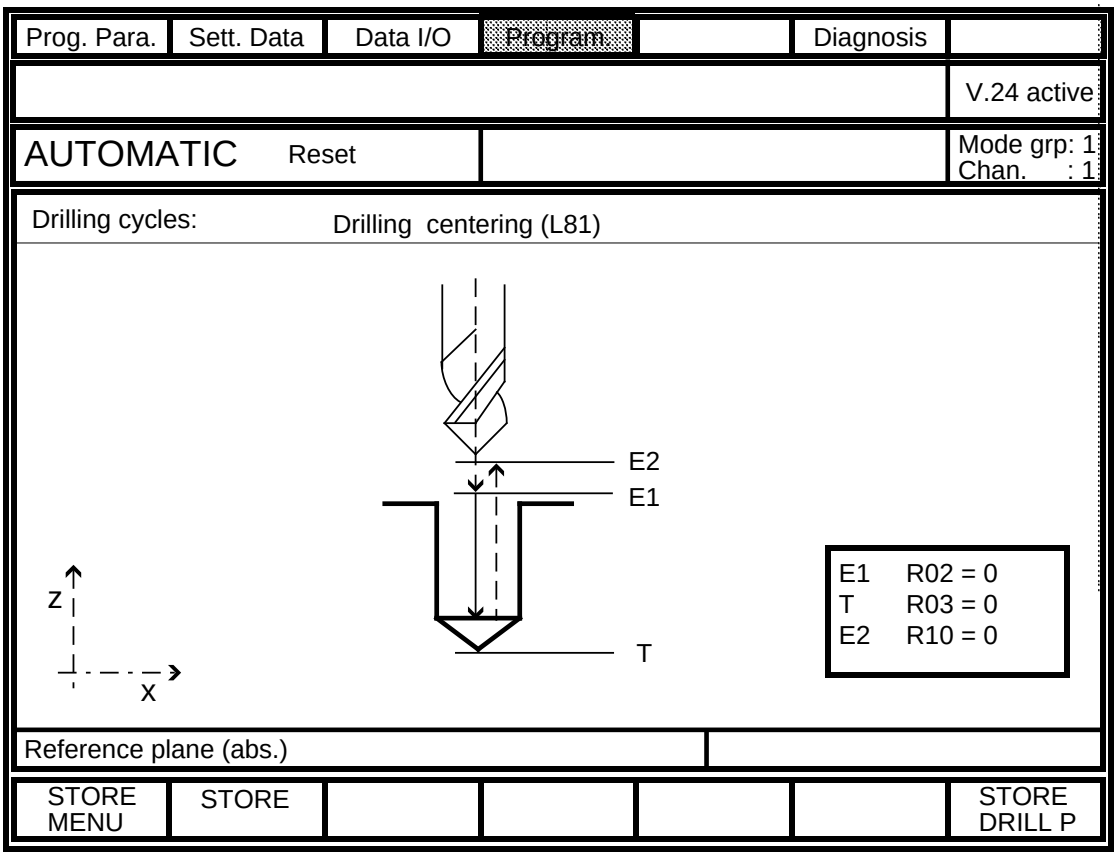
The following parameters are used in cycles L81 - L89:

Symbol	Parameter	Description
t1	R00	Dwell time at the initial point (for chip removal)
T1	R01	Enter first drilling depth without sign (incremental)
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 rotation for retraction (M03/M04)
M_	R07	Direction of rotation (M03/M04)
	R08	Tapping with and without encoder
P	R09	Thread pitch (only for tapping with encoder)
E2	R10	Retraction plane (absolute)
	R11	Deep hole drilling with chip breaking or swarf removal (L83)
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

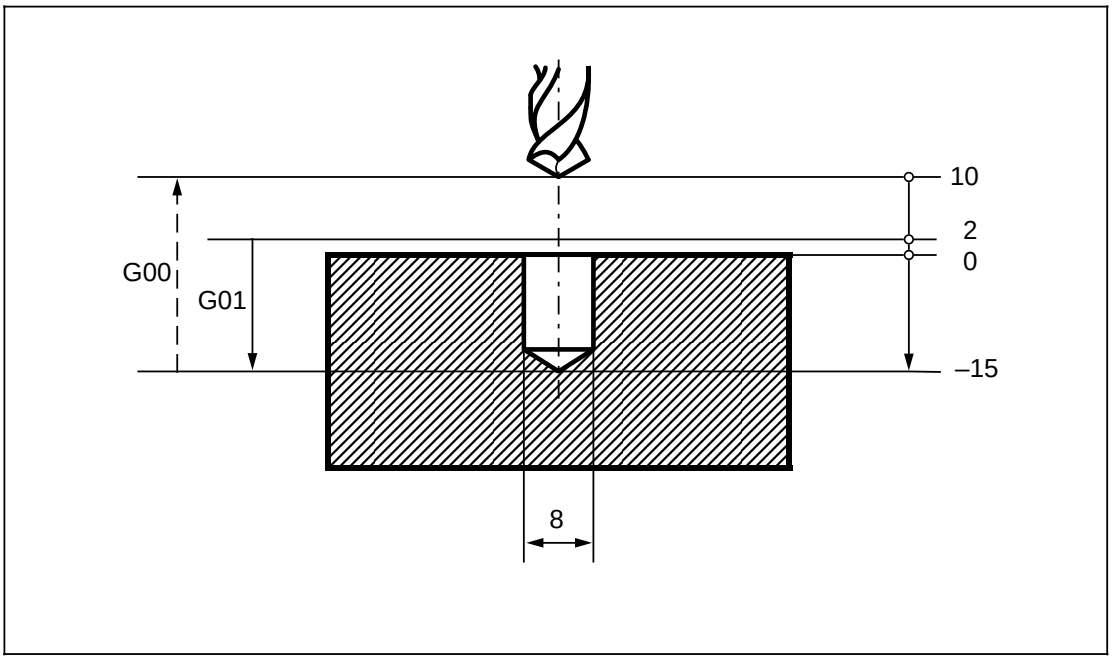
The following values are entered in the menu display or programmed directly as parameter assignments:

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

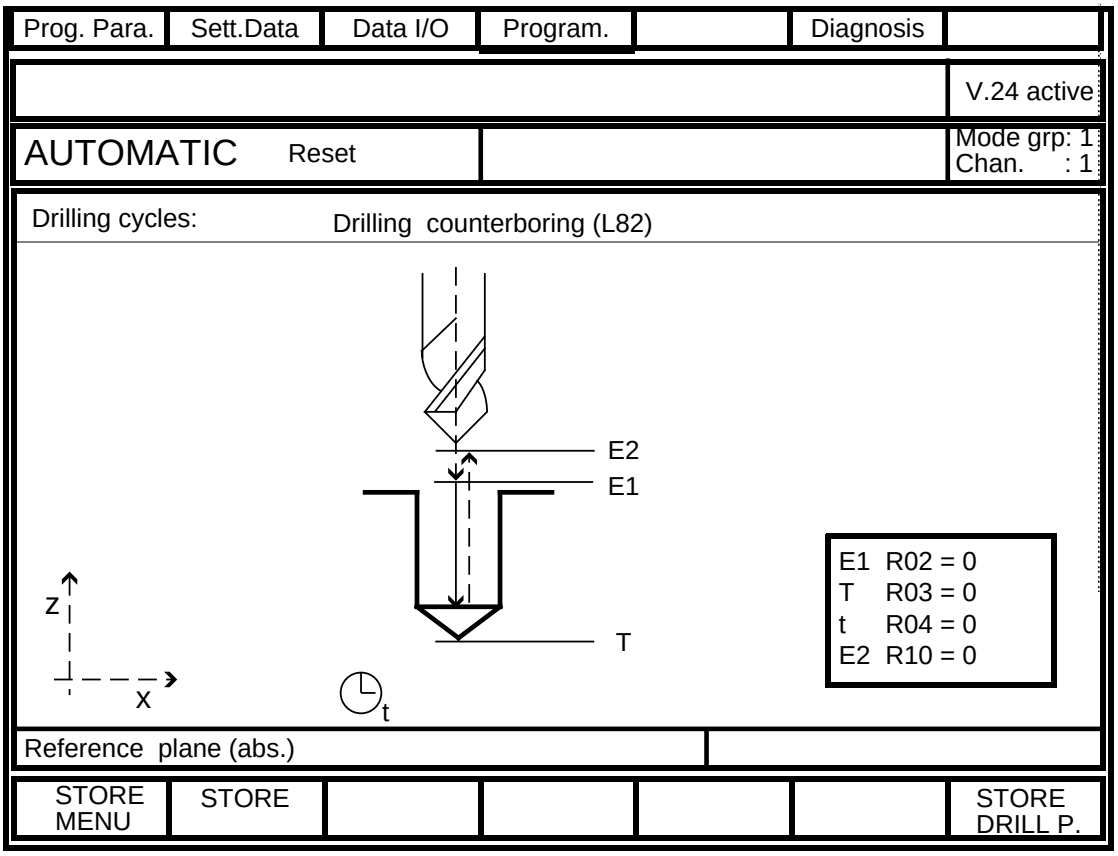
```
%81
N8101  G90  F130  S710  M03  LF
N8102  G00  D01  Z50  T03  LF
N8103  X10  Y15  LF                                  Select 1st drilling position
N8104  G81 R2=2 R3=-15 R10=10  LF                    Call up drilling cycle, 1st hole
N8105  X25  Y60  LF                                  Select 2nd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



Subroutine L82: Drilling, counterboring

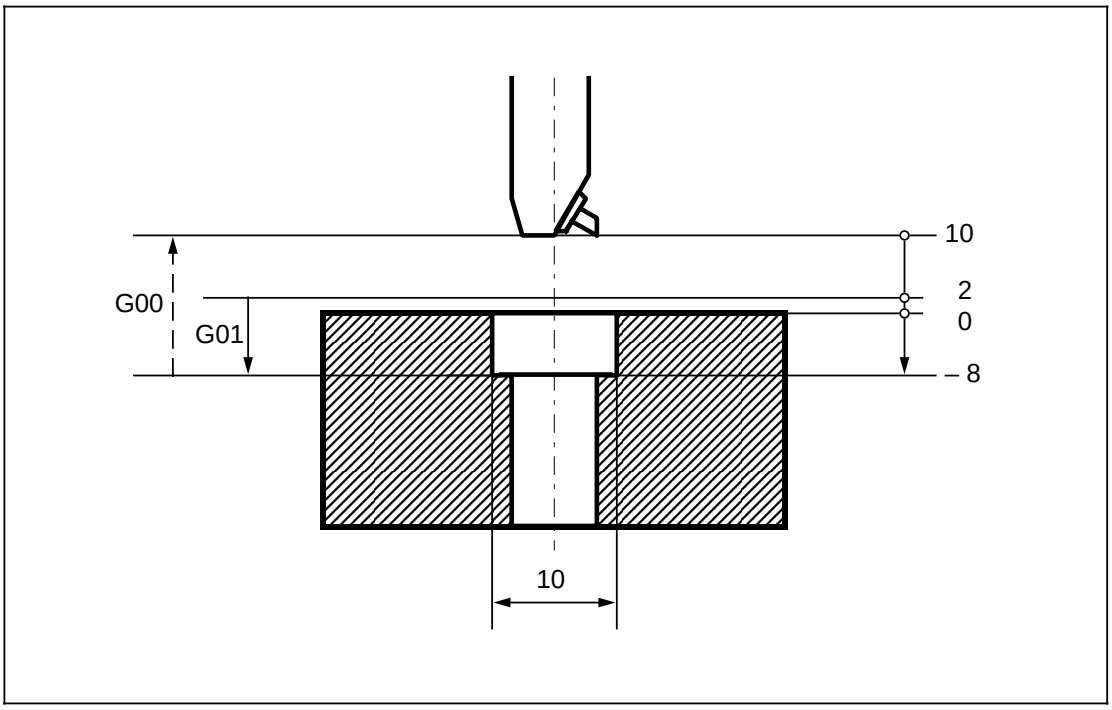
The following values are entered in the menu display or programmed directly as parameter assignments:

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
```

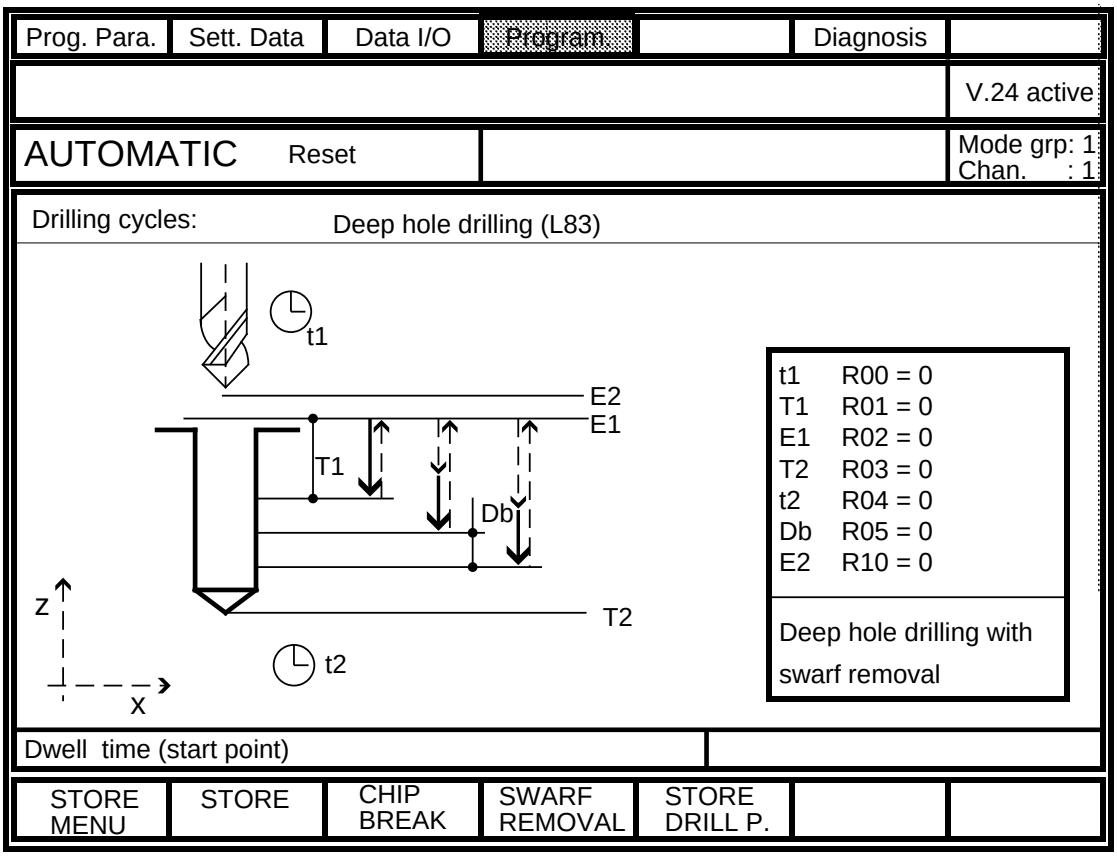


Subroutine L83: Deep hole drilling

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 initial point (for swarf 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 swarf 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 executed in accordance with the programmed R01 (first drilling depth).
2. The second drilling stroke is executed: this results 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 · R05 (a = remaining drilling depth)

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

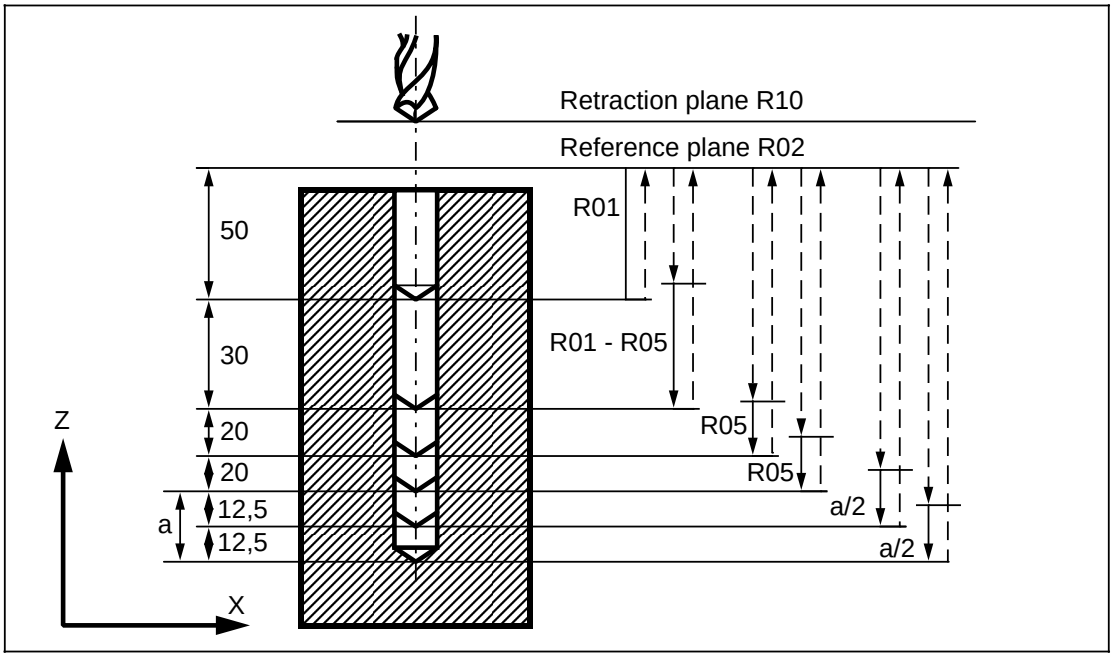
R11: Chip breaking, swarf removal

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 swarf 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. A compensating chuck must be used for tapping without encoder (exception: SINUMERIK 840C, SW 2 and higher).

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 %.

With SINUMERIK 840C (NC SW 2 and higher) it is possible to machine without a compensating chuck when tapping with encoder.

The following values must be 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; 0=without encoder, 1 = with encoder, 2 = without compensating chuck (SINUMERIK 840C, NC SW 2 and higher only)
P	R09	Thread pitch
	R10	Retraction plane (absolute)
	R14	Number of rotary axis (SINUMERIK 840C only)
	R20	Initial point of rotary axis (SINUMERIK 840C only)
	R21	Feedrate (rev/min) (SINUMERIK 840C only)

t R04: Dwell at thread depth

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

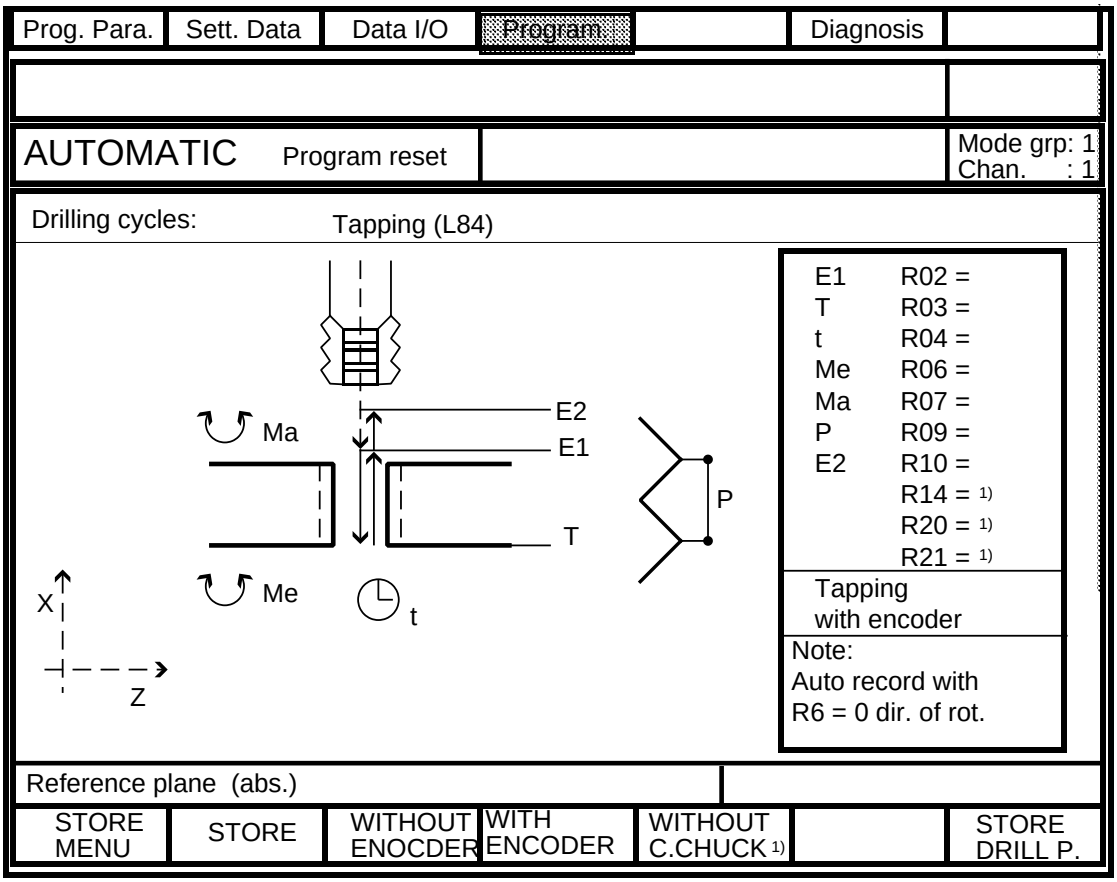
Me R06: Direction of rotation for retraction

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. The direction of rotation for the retraction movement is always determined automatically when tapping without compensating chuck.

Ma R07: Direction of rotation after cycle

This parameter determines the direction of rotation for tapping without compensating chuck, it **must** therefore be programmed.

R07 = 3 right-hand thread
R07 = 4 left-hand thread



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

The following applies only to tapping with compensating chuck:

In order to make the first hole for tapping, the direction of rotation must be written into the part program with M03 or M04 before calling up the cycle. This also applies if the cycle is only called once.

If R06 = 0 is programmed (automatic reversal of spindle direction of rotation), R07 need **not** be assigned.

1) *SINUMERIK 840C only*

R08: Tapping with/without encoder and without compensating chuck

The type of tapping operation is defined in parameter R08.

If the cycle is parameterized in the display, this selection is made with the softkeys.

The cycle automatically determines from MD 5013.1 whether the tapping operation is to be carried out with or without encoder.

MD 5013.1 = 1 : Tapping without encoder
MD 5013.1 = 0 : Tapping with encoder

However, if R08 is parameterized differently, the following applies:

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

2. If MD 5013.1 = 1 without encoder and R08 = 1 with encoder are selected, R08 is ignored.

3. If tapping is to be carried out with encoder and without compensating chuck, R08 = 2 must be set.

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.

R14: No. of rotary axis

This parameter applies only to SINUMERIK 840C, NC SW 2 and higher, when R08 = 2 (tapping without compensating chuck).

Tapping without compensating chuck is implemented on SINUMERIK 840C by interpolation of a rotary axis with a longitudinal axis (the drilling axis).

The number of the axis involved in the operation must be defined for the cycle.

R20: Initial point of the rotary axis

This parameter applies to SINUMERIK 840C, NC SW 2 , only, when R08 = 2 (tapping without compensating chuck).

The starting position of the rotary axis expressed in degrees before tapping is started must be programmed in R20. The end point is then calculated by the control from the programmed pitch and the linear axis.

R21: Feedrate (in rev/min)

This parameter also only applies when R08 = 2.

When tapping without compensating chuck, G98 is triggered by the control, i.e., the programmed feedrate F refers to the rotary axis in rev/min. The required feedrate must therefore be defined in parameter R21 for this cycle and type of operation.

The resulting tapping feedrate is determined by the control.

When the cycle has been completed, G98 is deselected by the G group 12 command active before the cycle was called.

Extension of L84 function

If cycle L84 is used for tapping with compensating chuck, the optional function ”Calculation of brake engagement point” is also available. This function corrects the tap overtravel depending on the spindle speed, actual gear speed and spindle acceleration time constant when tapping with G33.

The brake engagement 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 Tapping too short.
This alarm appears if an overtravel compensation path is calculated which is longer than the tapping.

Remedy:

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

Limitations:

- 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 only works 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.

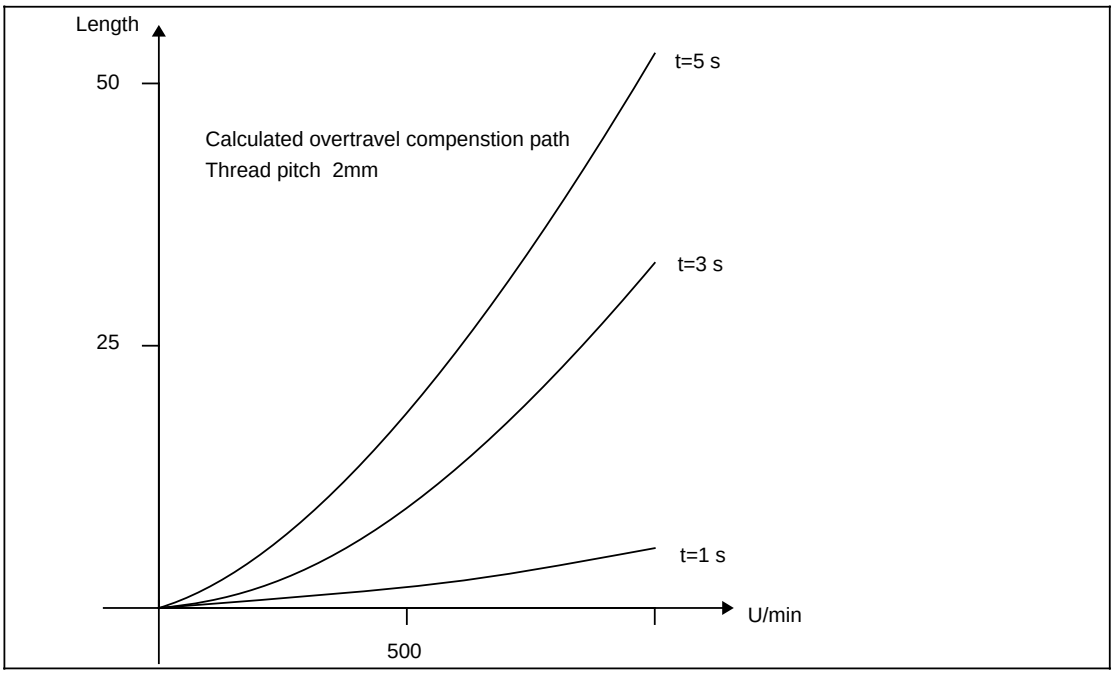


Diagram showing minimum thread length at different speeds and run-up time constants.

Points to note when tapping without compensating chuck

As tapping without a compensating chuck is implemented in SINUMERIK 840C by the interpolation of the rotary and the longitudinal axis, please ensure that the rotary axis involved in the tapping operation is rotating around the longitudinal axis.

Before calling the cycle, define the planes so that the drilling axis is the third axis of the current plane (G16 ... G19) because the cycle automatically calls this as the drilling axis.

Switching from spindle to rotary axis operation is always carried out outside the cycle.

If drilling cycle L84 is called from drilling pattern circle of holes (L900), please observe setting data 6.7 SD 5000.6. In this case it must be assigned with the value 1 (see description of cycle L900).

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

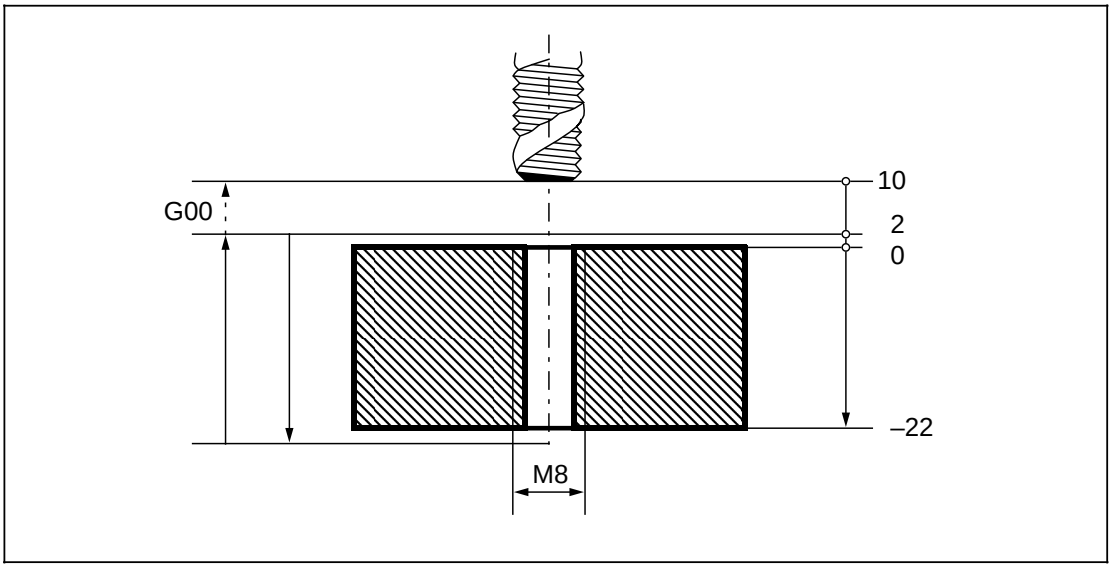
MD 5013.bit 1=0

```
%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
```



Example 3: "Tapping without compensating chuck" machining menu selected via softkey

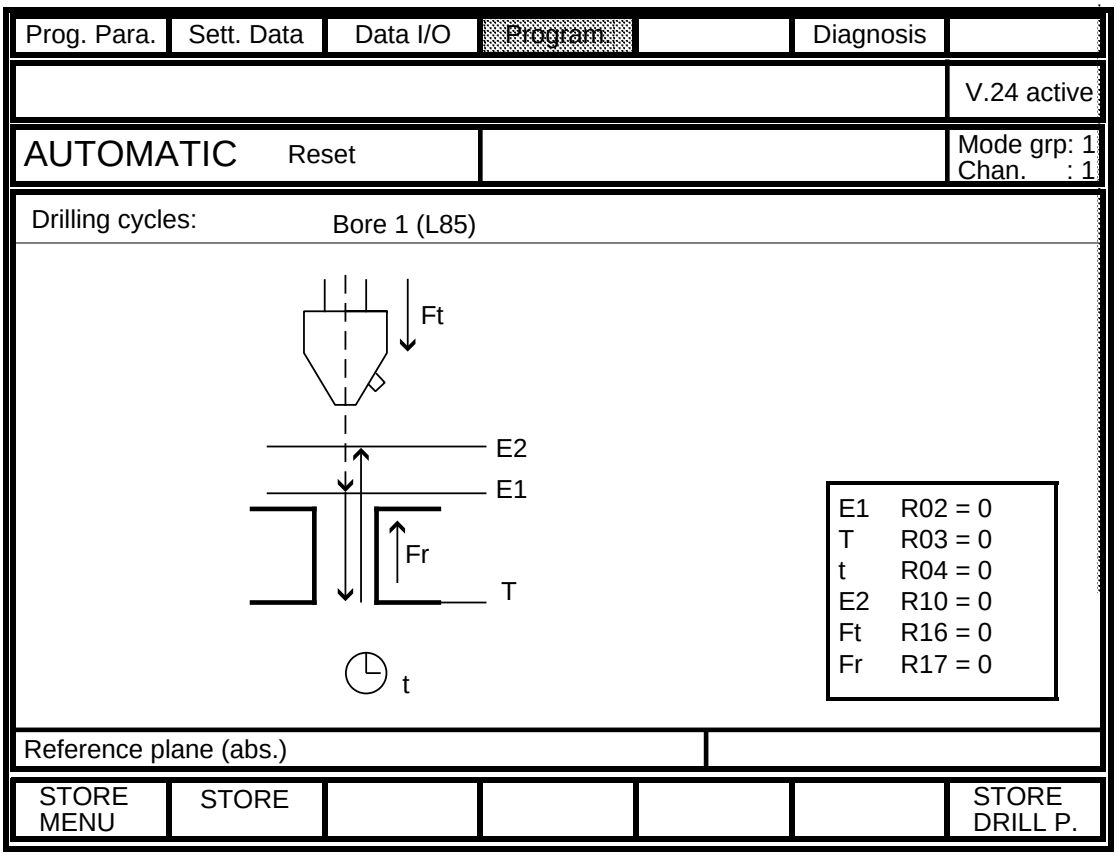
MD 5013.1=0

```
%3
N05   G17  ...                                  Select plane
                                                (Z axis is then selected
                                                internally in the cycle =
                                                3rd axis of the plane as the
                                                drilling axis)
N10   ...                                       Change spindle to axis
                                                mode (with M command)
N15   G90  G0  X20  420  LF                     Select drilling position in
                                                the plane
N20   R02=2  R03=-30  R04=0                     Parameterize cycle
      R06=4  R07=3  R08=2
      R09=5  R10=10  R14=5
      R20=30  R21=1000  RF
N25   L84  P1  LF                               Call drilling cycle
N30   ...                                       End rotary axis operation
N35   M30  LF
```

Subroutine L85: Bore 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 in drilling depth (chip breaking)	
E2	R10	Retraction plane	(absolute)
Ft	R16	Feedrate	
Fr	R17	Retraction feedrate	

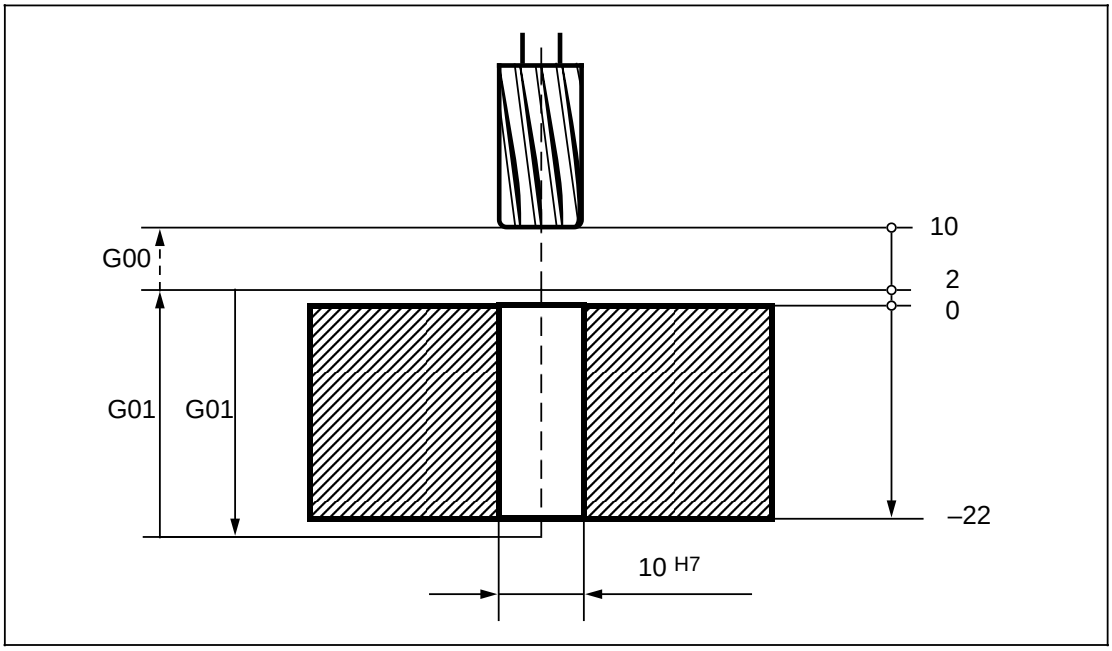


With setting data 5000, bit 0 = 0, a mode compatible with UMS 2 is possible, i.e. programs generated with UMS 2 can be run.

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.

Example: "Bore 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
```

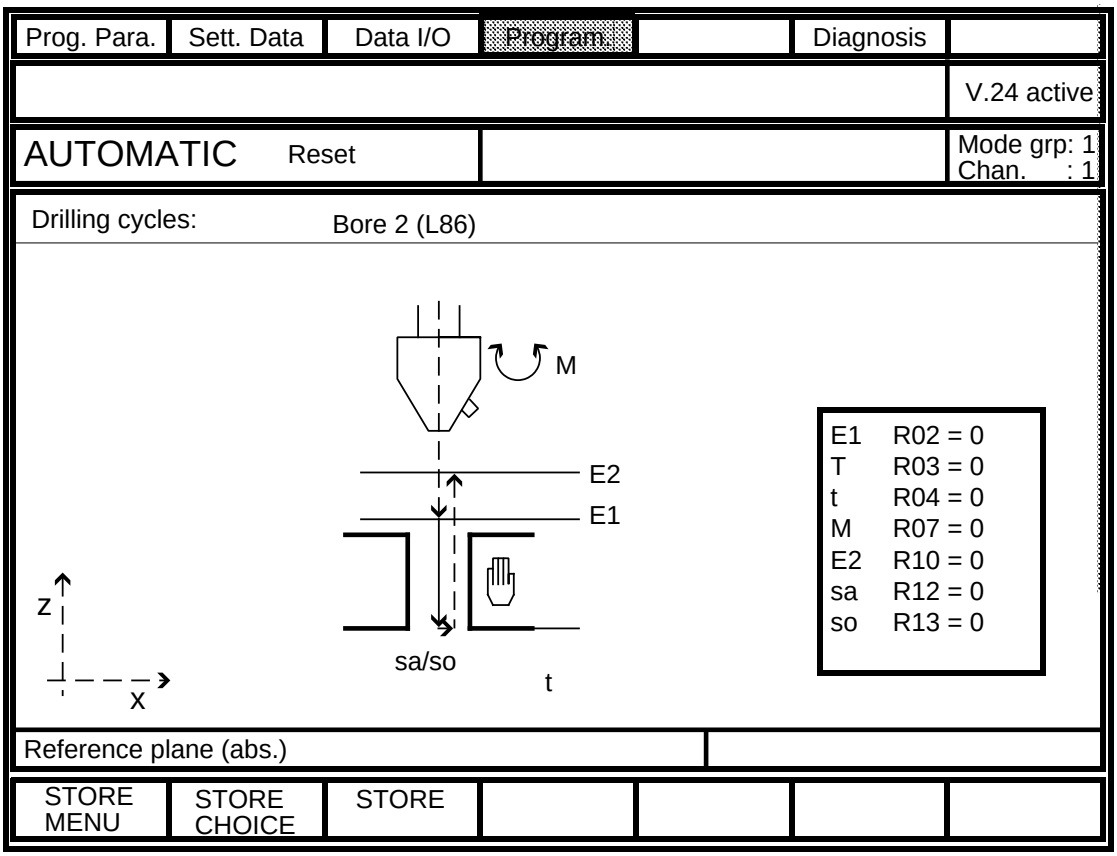


With L85, the inward and outward movements are executed with the feedrates programmed in R parameters R16 and R17.

Subroutine L86: Bore 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)

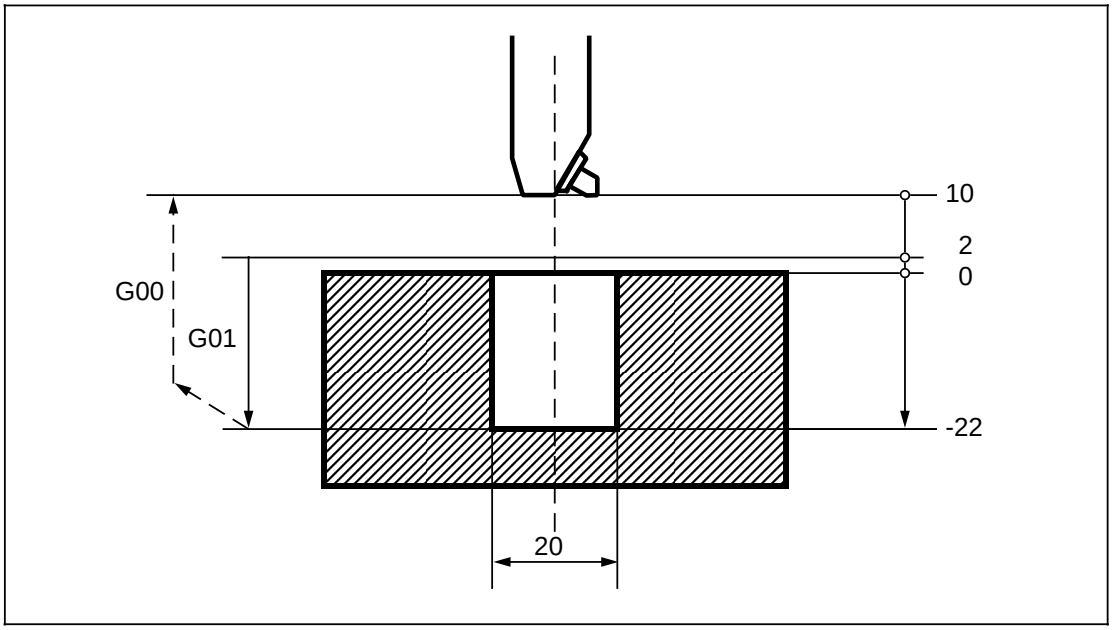


With setting data 5000, bit 0 = 0, a mode compatible with UMS 2 is possible, i.e. programs generated with UMS 2 can be run.

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: "Bore 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.

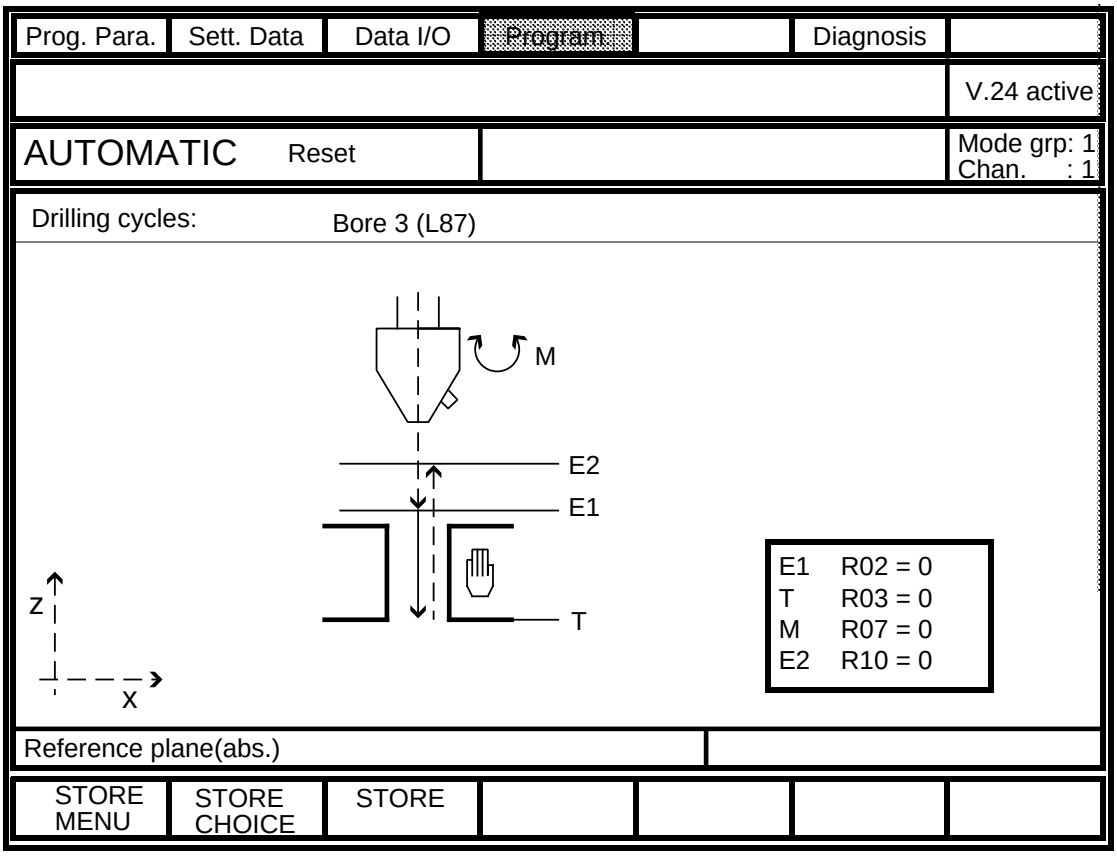
With M19, an oriented stop can be executed with the main spindle. The corresponding angle value is programmed and entered via the operator's panel under "Setting data spindle".

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

Subroutine L87: Bore 3

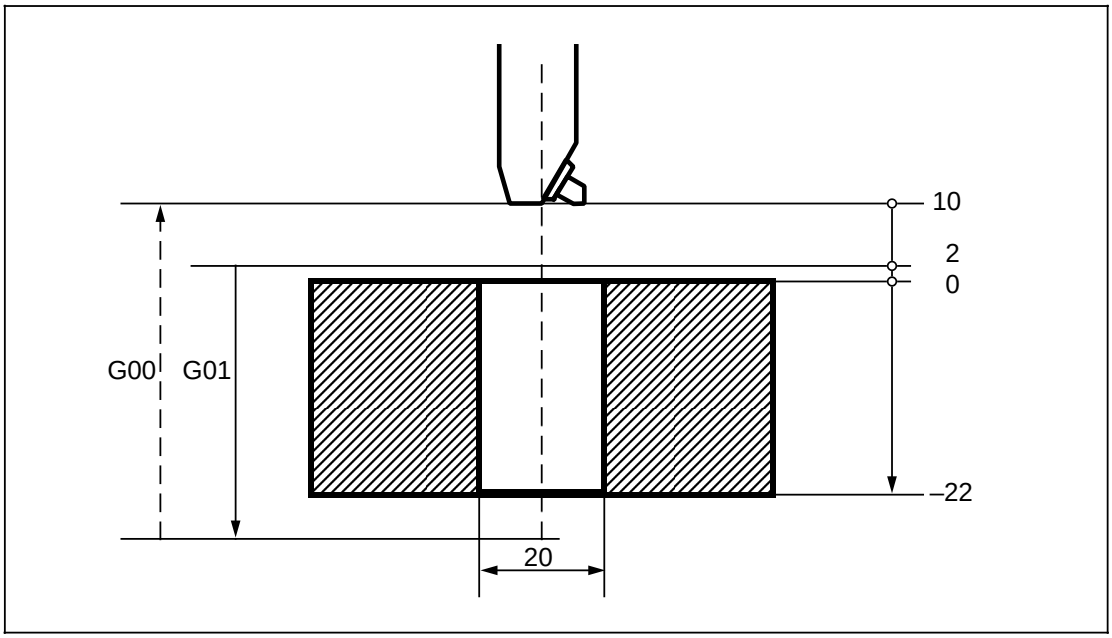
The following values are entered in the menu display or programmed directly as parameter assignments:

Symbol	**P**arameter	**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: "Bore 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  LFF
```

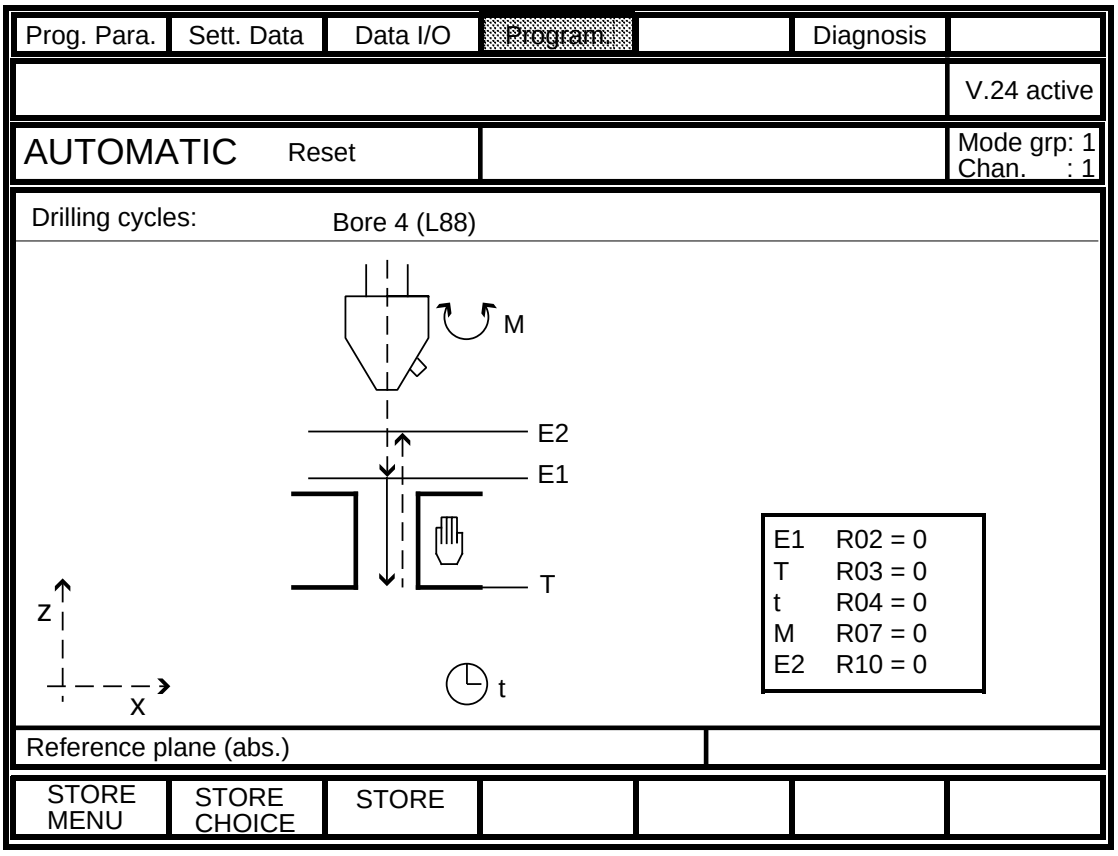


Once the final drilling depth has been reached, an M05 non-oriented spindle stop and an M00 program stop are executed. By pressing the NC start key, the outward movement is continued in rapid traverse as far as the retraction plane.

Subroutine L88: Bore 4

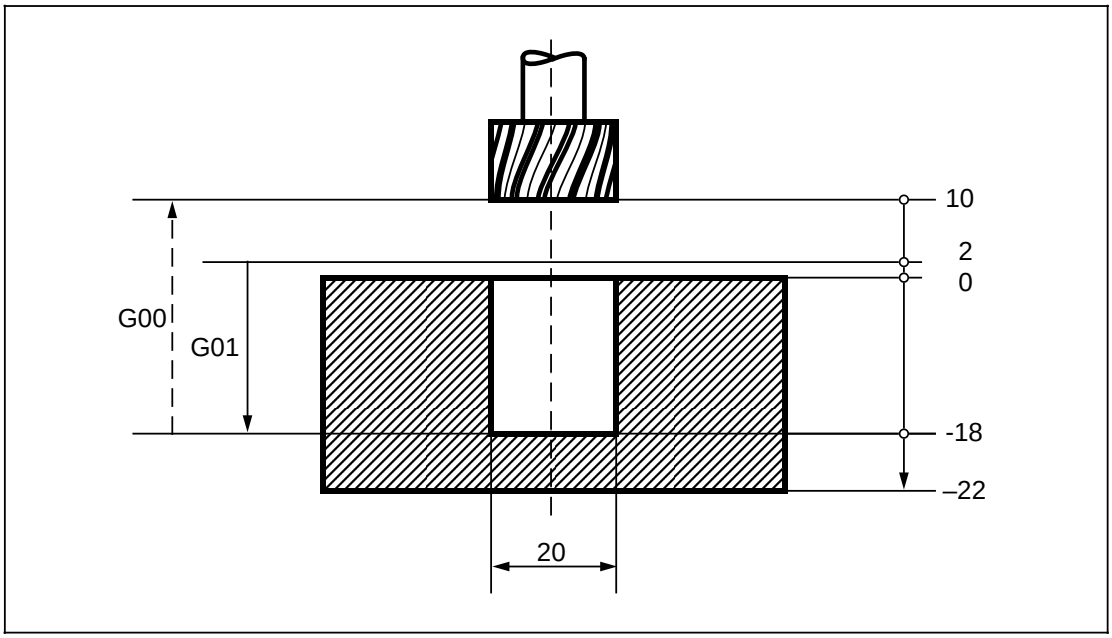
The following values are entered in the menu display or programmed directly as parameter assignments

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 rotation (M03/M04)	
E2	R10	Retraction plane	(absolute)



Example: "Bore 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
```

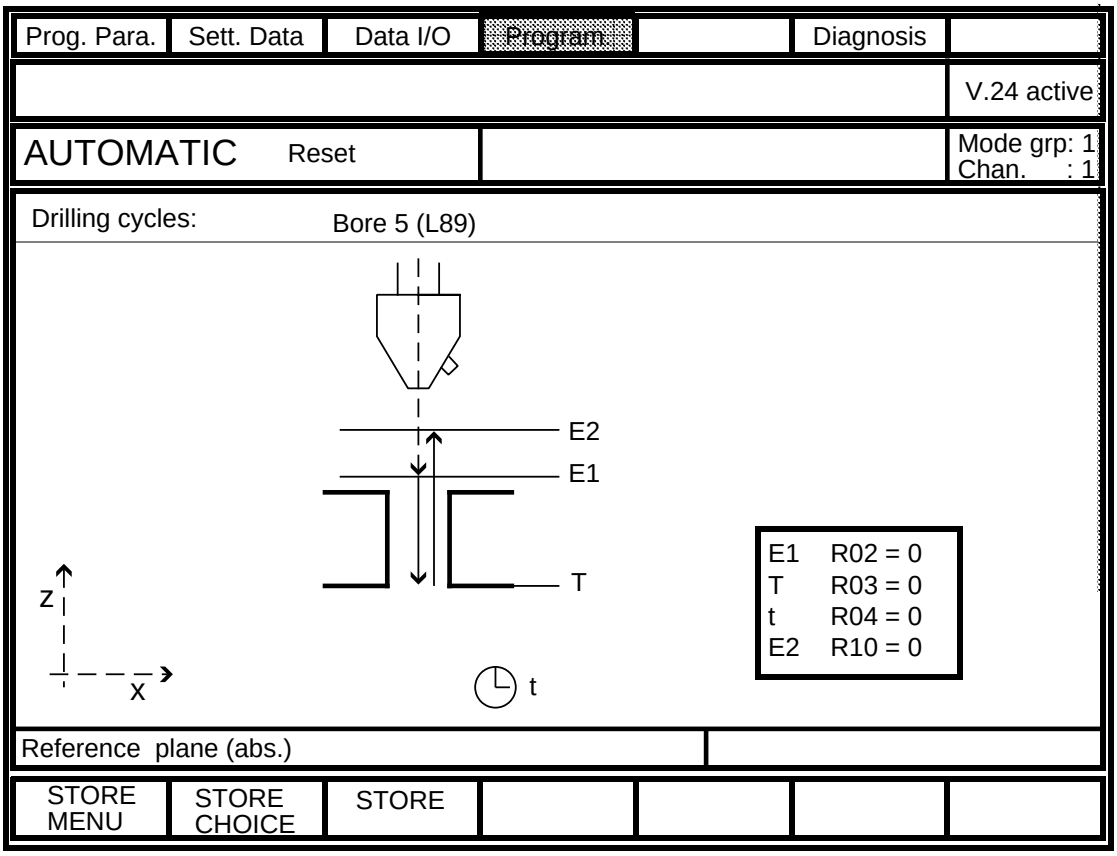


Once the final drilling depth has been reached, an M05 non-oriented spindle stop and an M00 program stop are executed. By pressing the NC start key, the outward movement is continued in rapid traverse as far as the retraction plane. A dwell time can be programmed at the final drilling depth.

Subroutine L89: Bore 5

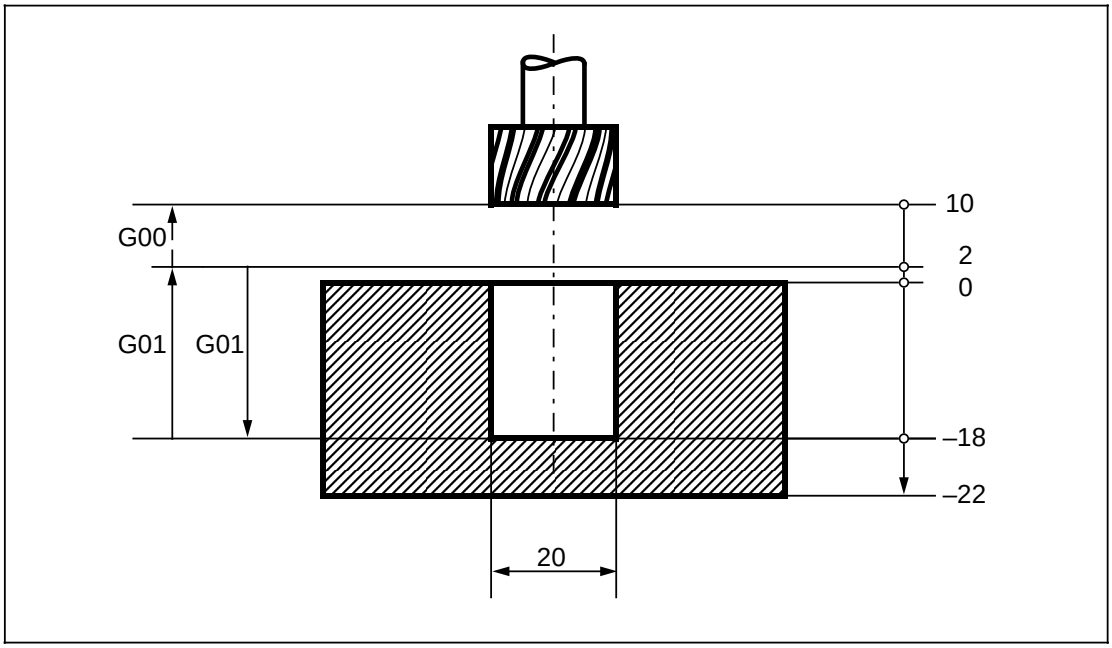
The following values are entered in the menu display or programmed directly as parameter assignments:

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: "Bore 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

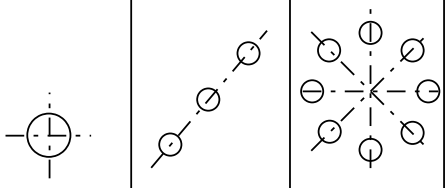
The following parameters are used in cycles L900 to L930:

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)	

* On the SINUMERIK 840, 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).

Prog. Para.	Sett. Data	Data I/O	Program.		Diagnosis	
						V.24 active
AUTOMATIC Reset						Mode grp: 1 Chan. : 1

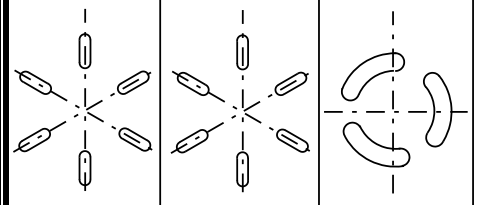
Drill patterns:



SINGLE HOLE	ROW OF HOLES	CIRCLE OF HOLES				

Prog. Para.	Sett. Data	Data I/O	Program.		Diagnosis	
						V.24 active
AUTOMATIC Reset						Mode grp: 1 Chan. : 1

Mill patterns:



SLOT	ELONGA. HOLE	CIRCLE SLOT				

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.

The following should be noted when using L900 together with drilling cycle L84:

An additional safety clearance of 1 mm along the drilling axis is traversed in cycle L900 outside the drilling cycle.

With UMS 48 and higher it is possible to suppress this by setting setting data SD 5000, bit 6.

SD 5000.6 = 0 L900 as before (with safety clearance)
= 1 L900 without safety clearance along the drilling axis

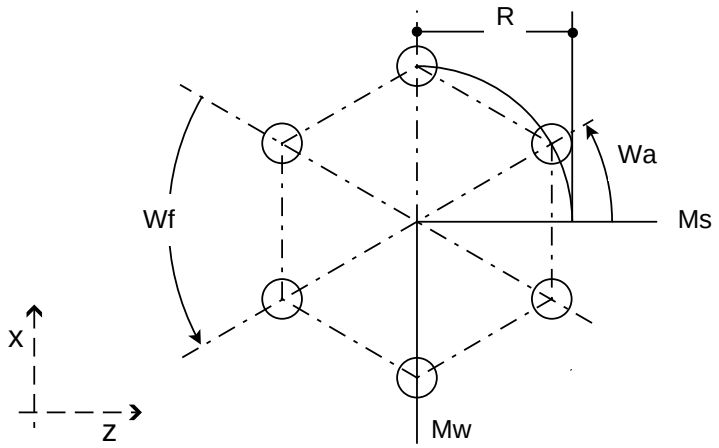
If L84 is used for tapping without compensating chuck, this bit must be set so that alarm 3092 is triggered in cycle L900.

If the cycle is to run as before, a feedrate value must be programmed in the part program before L900 is called.

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	Initial angle (referred to the horizontal axis)
Wf	R26	Indexing angle
	R27	Number of holes
	R28	Number of drilling cycle required (L81 to L89)

Prog. Para.	Sett. Data	Data I/O	Program		Diagnosis	
						V.24 active
AUTOMATIC Reset						Mode grp: 1 Chan. : 1

Drill patterns: Circle of holes



Mw	R22 = 0
Ms	R23 = 0
R	R24 = 0
Wa	R25 = 0
Wf	R26 = 0
	R27 = 0
	R28 = 0

Centre (abs.)

STORE MENU	STORE CHOICE					

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).

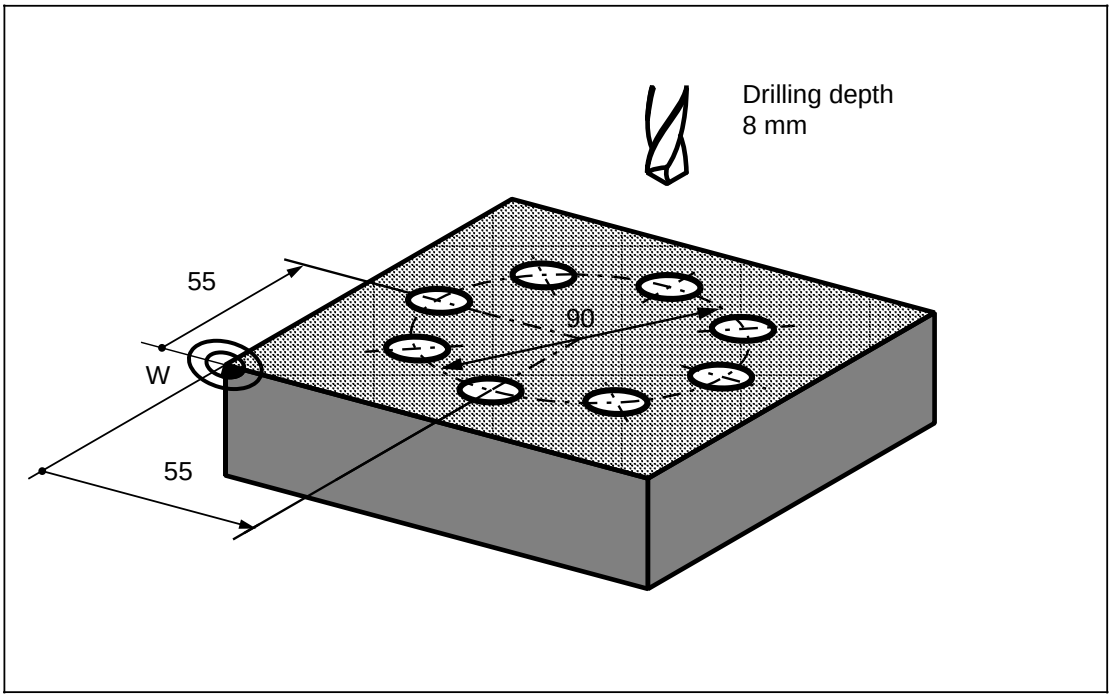
Wf R26: Indexing angle

If 0 is given as the indexing angle, the number of holes is divided up over 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

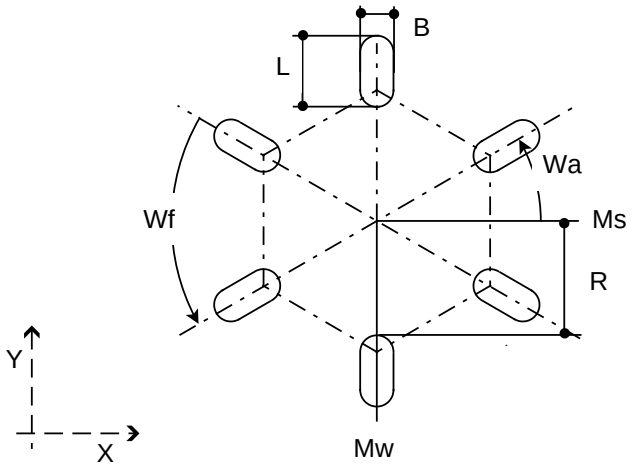
When programming, either the "Slot" menu is selected and the R parameters are entered in the menu displays or the parameters are programmed directly as parameter assignments in the part program: subroutine L901 is active in the current plane.

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	Initial 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.

Prog. Para.	Sett. Data	Data I/O	Program		Diagnosis	
						V.24 active
AUTOMATIC Reset						Mode grp: 1 Chan. : 1

Mill patterns: Slot

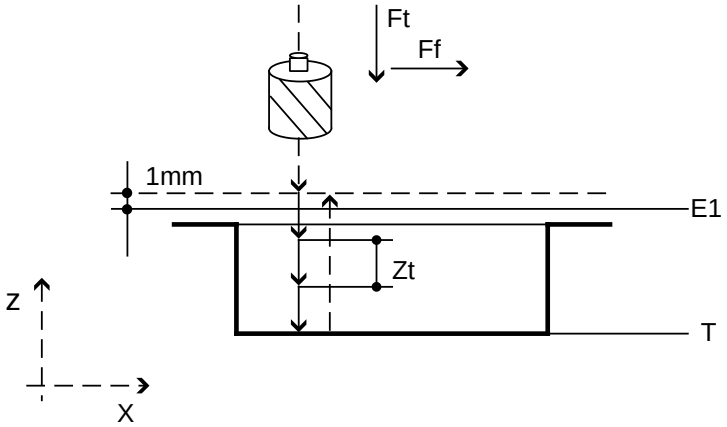


B	R12 = 0
L	R23 = 0
Mw	R22 = 0
Ms	R23 = 0
R	R24 = 0
Wa	R25 = 0
Wf	R26 = 0
	R27 = 0

Slot width						
						DEPTH

Prog. Para.	Sett. Data	Data I/O	Program		Diagnosis	
						V.24 active
AUTOMATIC Reset						Mode grp: 1 Chan. : 1

Mill patterns: Slot



Zt	R01 = 0
E1	R02 = 0
T	R03 = 0
Ff	R15 = 0
Ft	R16 = 0

Infeed depth (incr.)						
STORE MENU	STORE CHOICE	STORE				

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 given as the indexing angle, the number of slots is divided up over 360°.

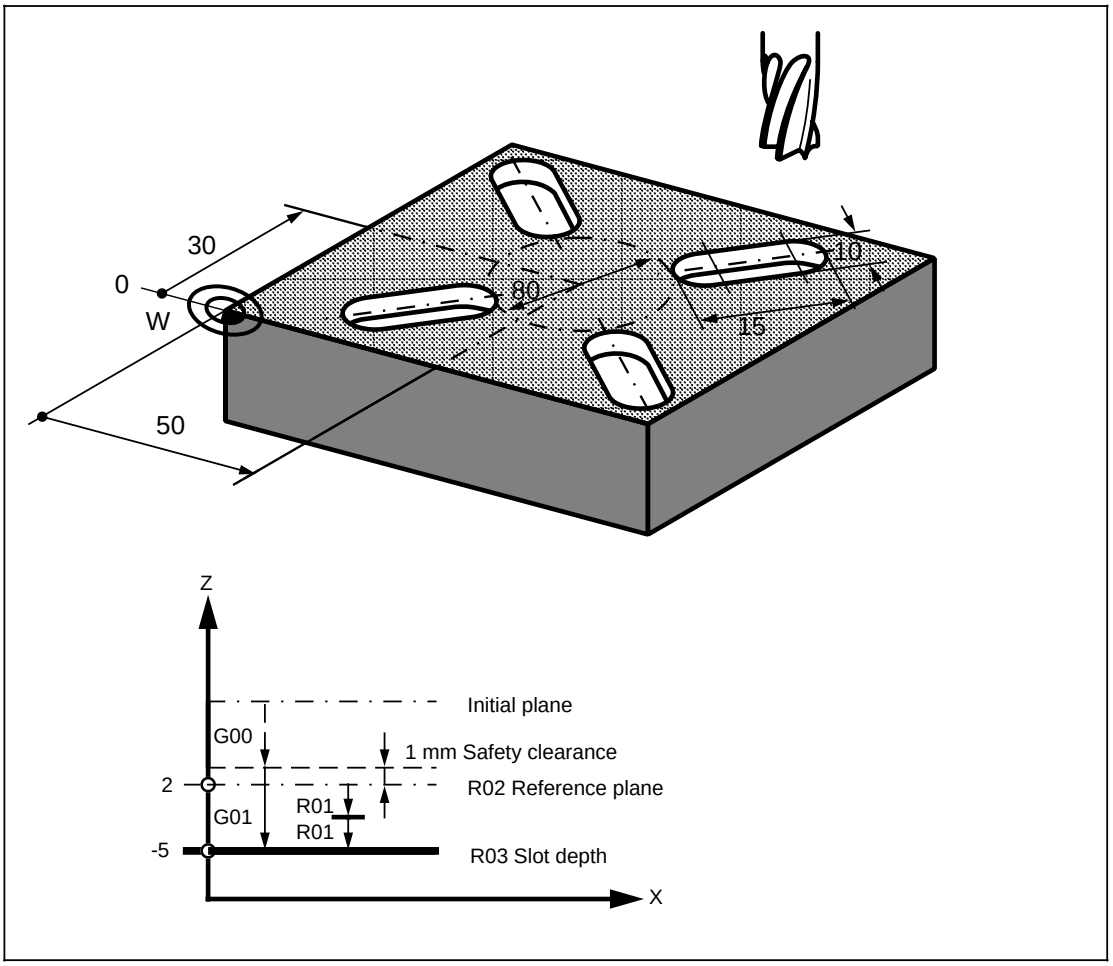
B R12: Slot width

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

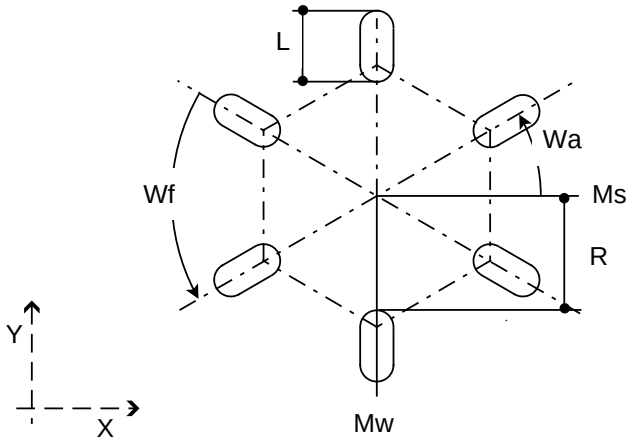
When programming, either the "Elongated hole" menu is selected and the R parameters are entered in the menu displays, or these values are directly programmed as parameter assignments in the part program. Subroutine L902 is active in the current plane.

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	Initial 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.

Prog. Para.	Sett. Data	Data I/O	Program		Diagnosis	
						V.24 active
AUTOMATIC Reset						Mode grp: 1 Chan. : 1

Mill patterns: Elongated hole



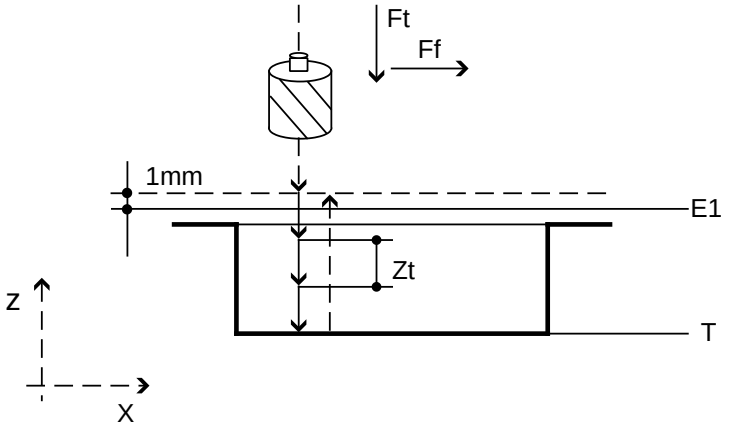
L	R23 = 0
Mw	R22 = 0
Ms	R23 = 0
R	R24 = 0
Wa	R25 = 0
Wf	R26 = 0
	R27 = 0

Elongated hole length

						TIEFE

Prog. Para.	Sett. Data	Data I/O	Program		Diagnosis	
						V.24 active
AUTOMATIC Reset						Mode grp: 1 Chan. : 1

Mill patterns: Elongated hole



Zt	R01 = 0
E1	R02 = 0
T	R03 = 0
Ff	R15 = 0
Ft	R16 = 0

Infeed depth (incr.)

STORE MENU	STORE CHOICE	STORE				

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.

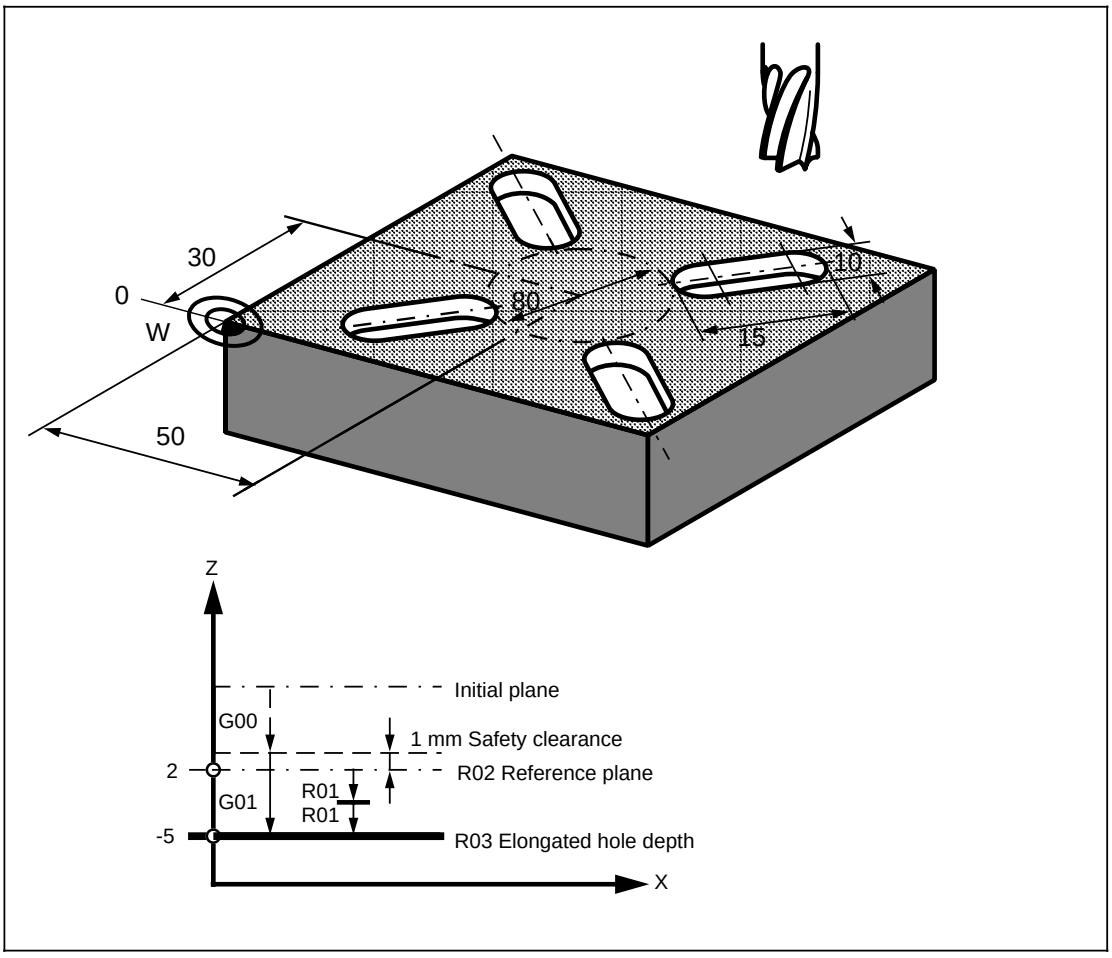
Wf R26: Indexing angle

If 0 is given as the indexing angle, the number of slots is divided up over 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

When programming, either the "Milling 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	

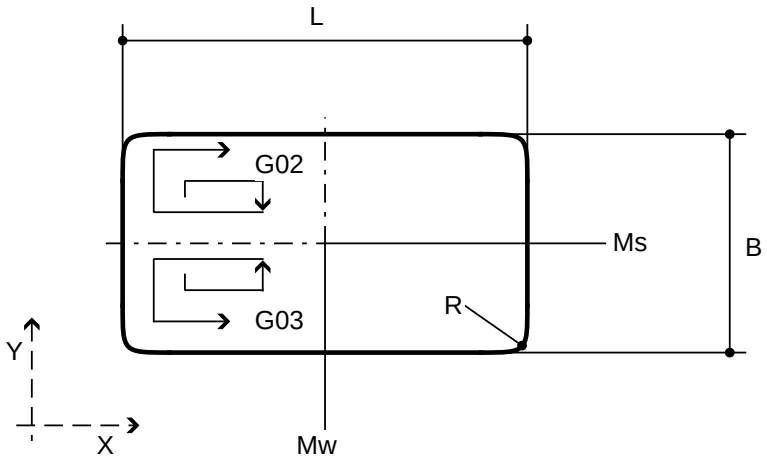
In cycle L903, cutter radius compensation is deselected (G40). The milling cutter radius is automatically taken into account and this must be stored in the tool offset memory.

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.

Prog. Para.	Sett. Data	Data I/O	Program		Diagnosis	
						V.24 active
AUTOMATIC Reset						Mode grp: 1 Chan. : 1

Mill cycles: Rectangular pocket



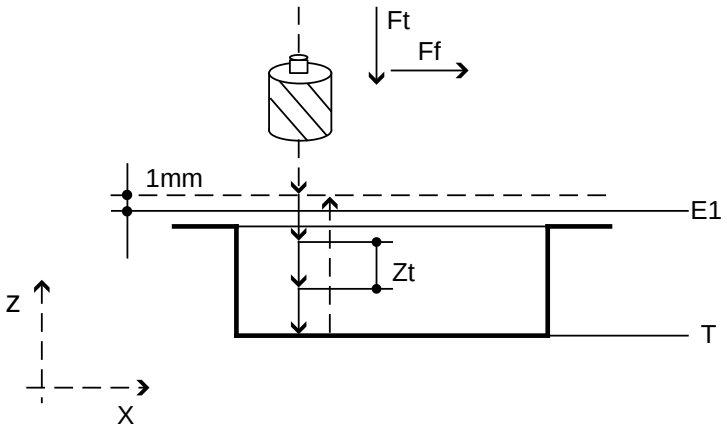
G	R06 = 0
L	R12 = 0
B	R13 = 0
Mw	R22 = 0
Ms	R23 = 0
R	R24 = 0

Cutter direction

						DEPTH

Prog. Para.	Sett. Data	Data I/O	Program		Diagnosis	
						V.24 active
AUTOMATIC Reset						Mode grp: 1 Chan. : 1

Mill cycles: Rectangular pocket



Zt	R01 = 0
E1	R02 = 0
T	R03 = 0
Ff	R15 = 0
Ft	R16 = 0

Infeed depth (incr.)

STORE MENU	STORE CHOICE	STORE				

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.

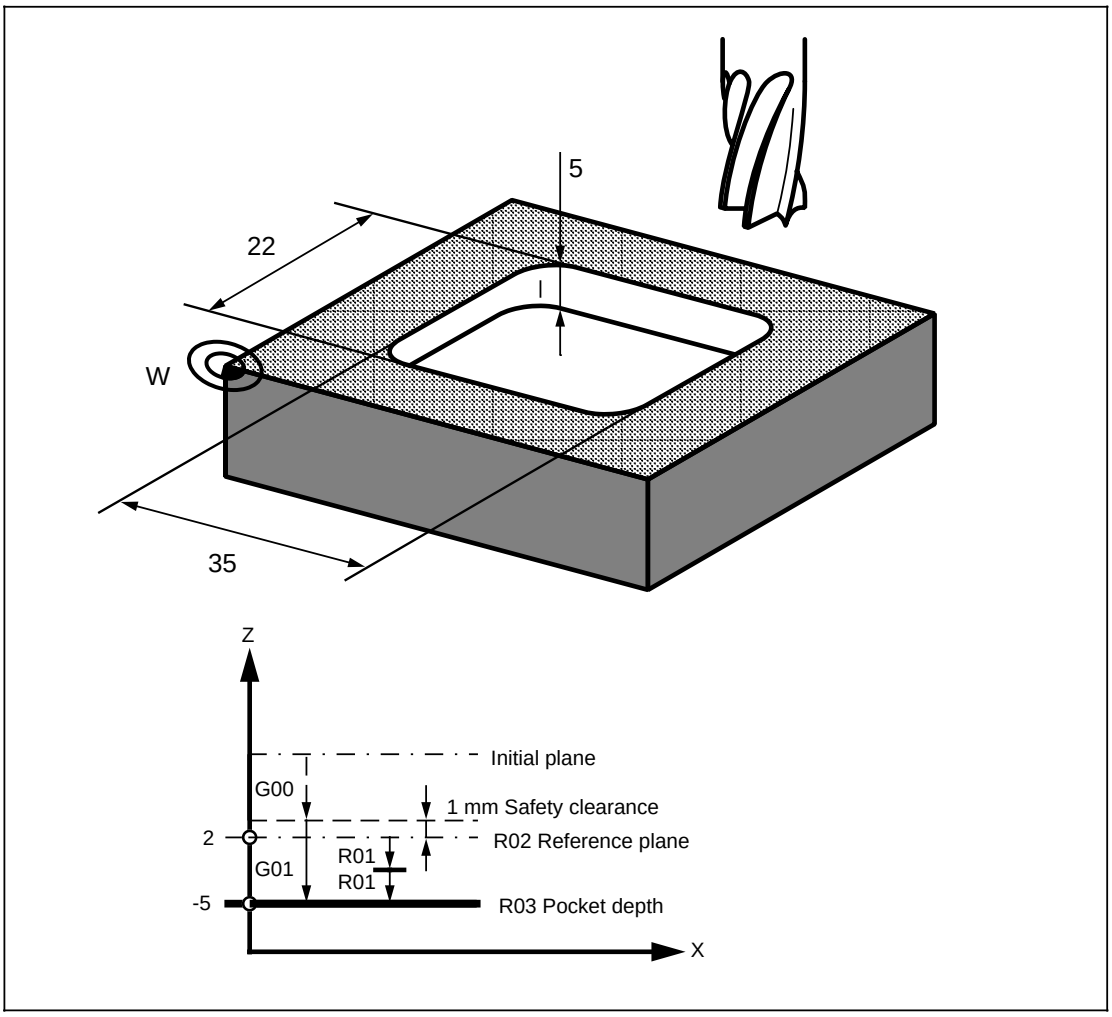
R R24: Corner radius

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

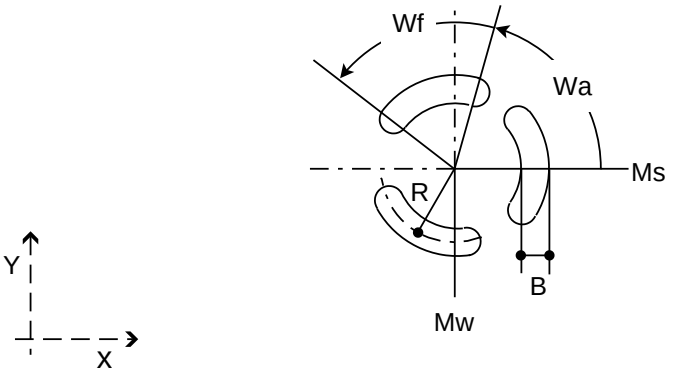
When programming, either the "Circular slot" menu is selected and the R parameters are entered in the menu displays, or these values are programmed directly as parameter assignments in the part program. Subroutine 904 is active in the current plane.

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	Initial angle (referred to horizontal axis)	
	R27	Number of slots	

Prog. Para.	Sett. Data	Data I/O	Program		Diagnosis	
						V.24 active
AUTOMATIC Reset						Mode grp: 1 Chan. : 1

Mill patterns: Circular slots



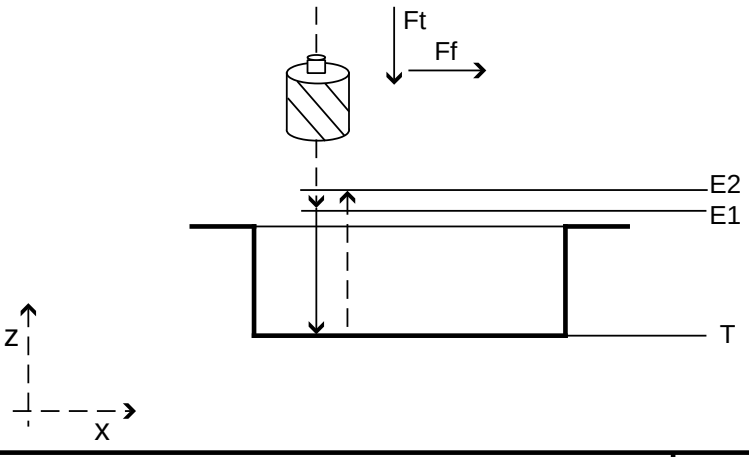
B	R12 = 0
Wf	R13 = 0
Mw	R22 = 0
Ms	R23 = 0
R	R24 = 0
Wa	R25 = 0
	R27 = 0

Slot width

						DEPTH

Prog. Para.	Sett. Data	Data I/O	Program		Diagnose	
						V.24 active
AUTOMATIC Reset						Mode grp: 1 Chan. : 1

Mill patterns: Circular slots



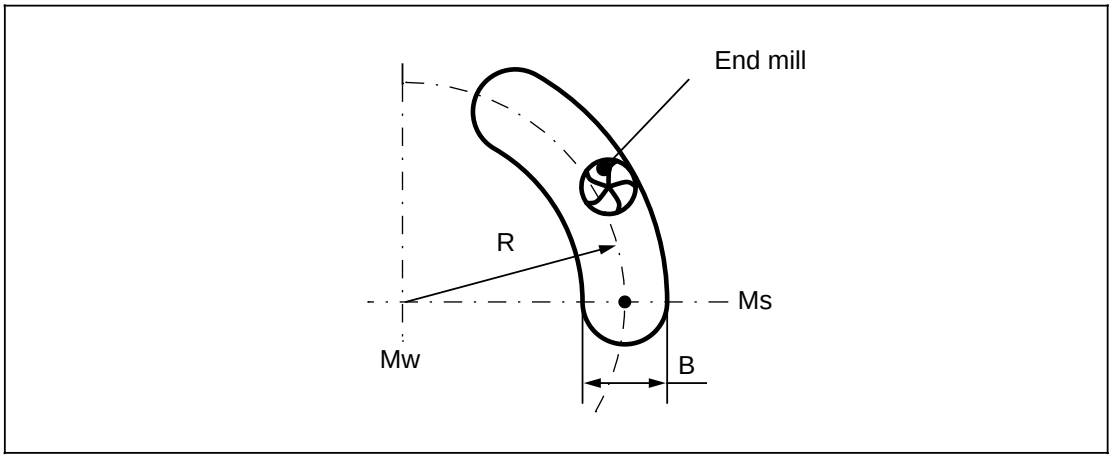
E1	R02 = 0
T	R03 = 0
E2	R10 = 0
Ff	R15 = 0
Ft	R16 = 0

Reference plane(abs.)

STORE MENU	STORE CHOICE	STORE				

Wa R25: Initial angle

The initial 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.

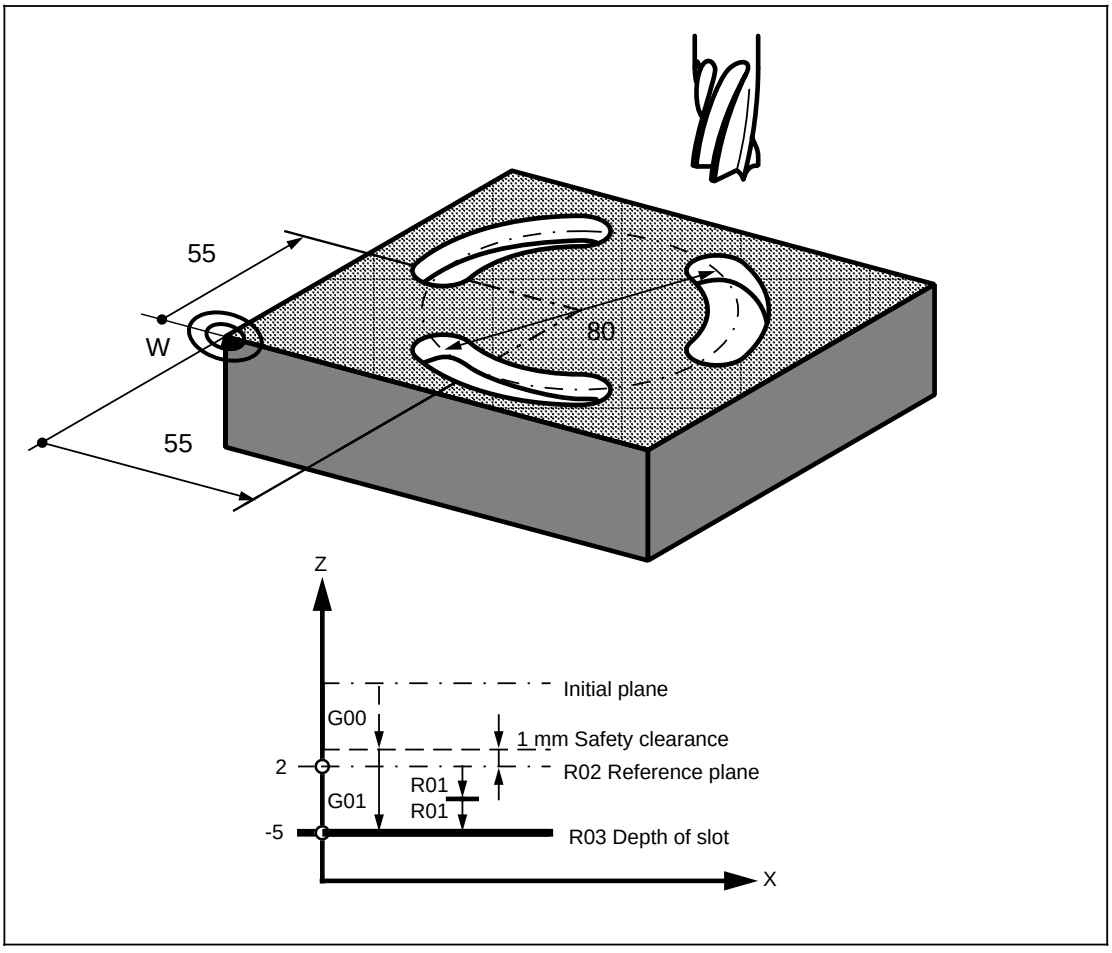
B R12: Slot width

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).

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
```

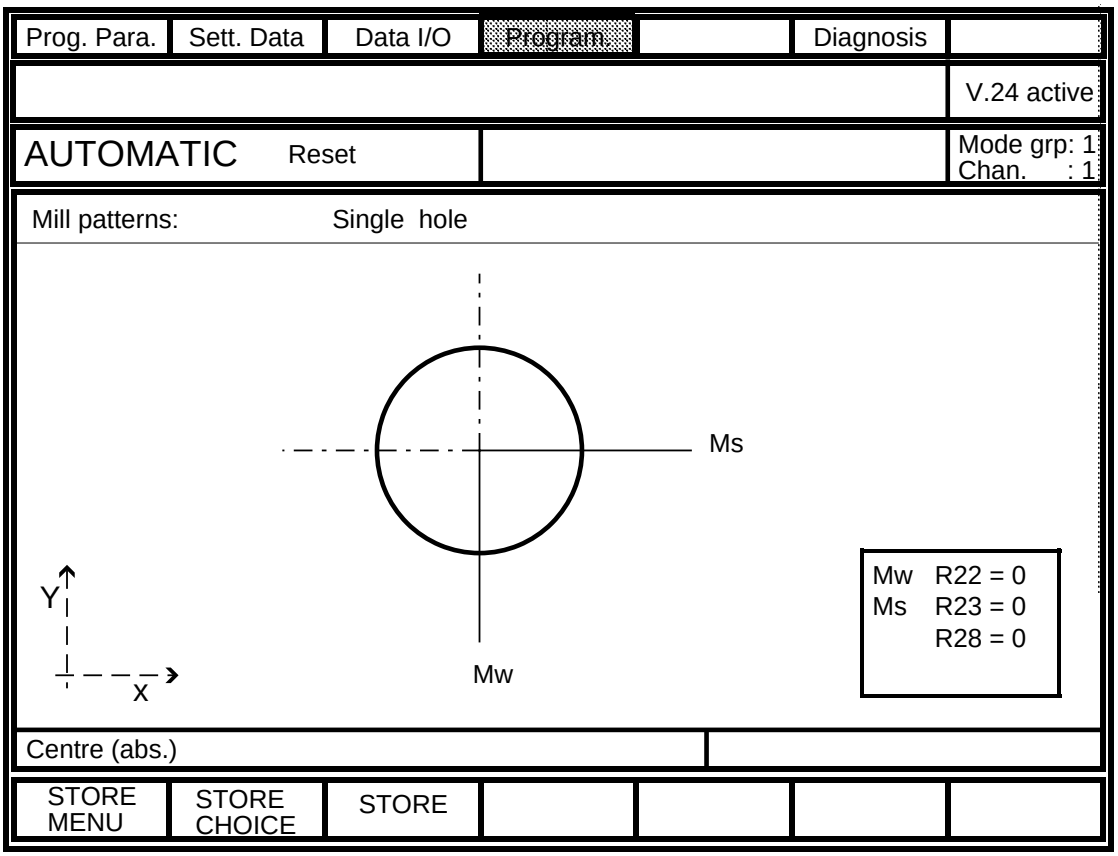
The parameters are assigned in two menu displays.



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) (absolute) (referred to workpiece zero)
	R28	Number of required drilling cycle (L81 to L89)



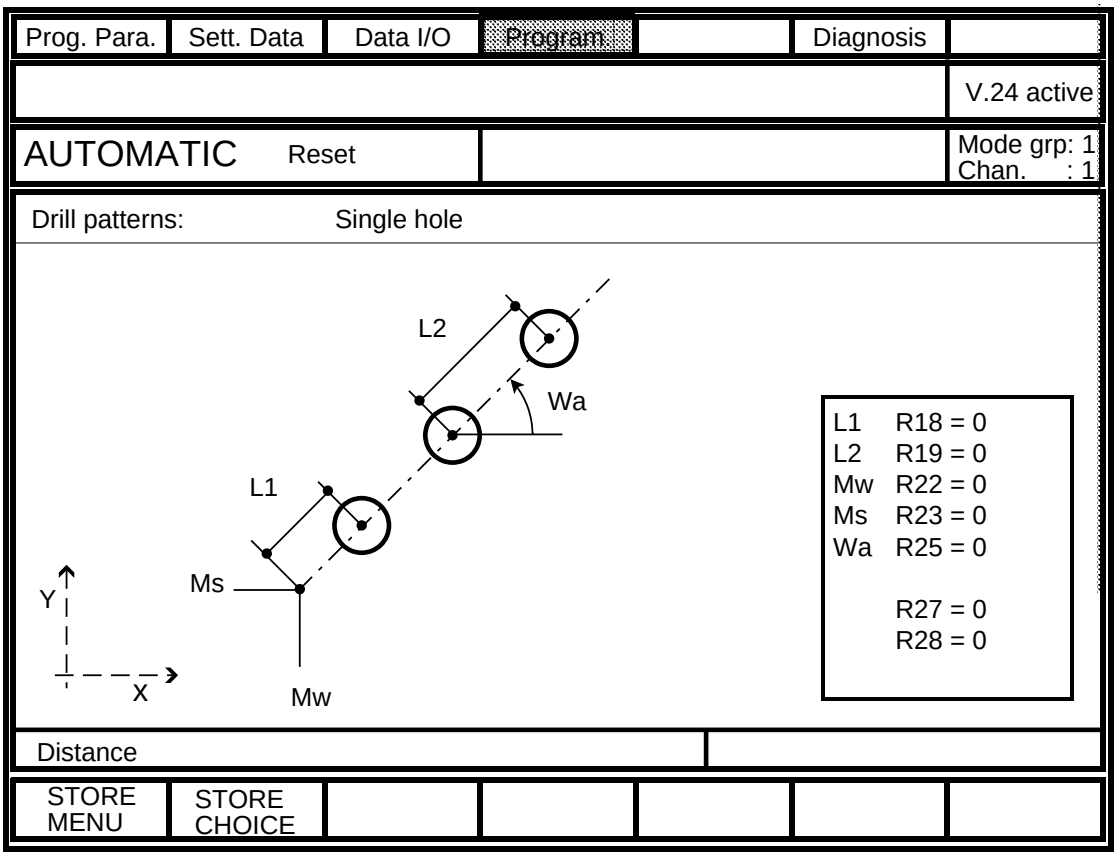
R28: Number of required drilling cycle (L81 to L89)

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

2.2.2.7 L906 "Row of holes" drilling pattern

When 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	Initial 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)

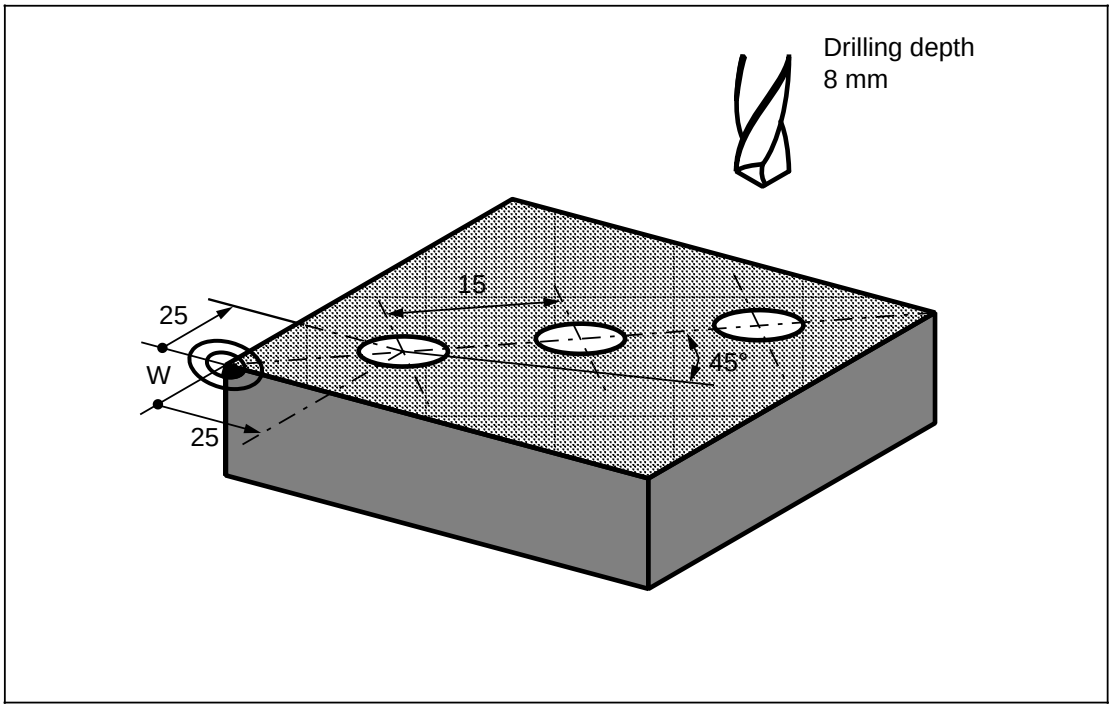
R19 is the hole spacing and must be entered as an incremental value.

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" machining 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
```



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)

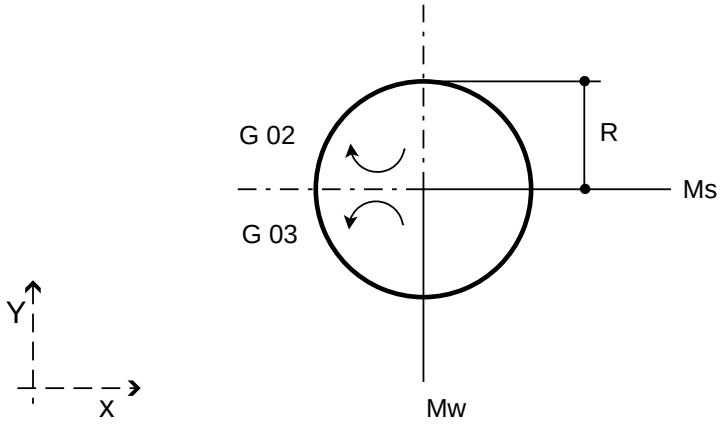
If the infeed depth is assigned with R1 = 0, the infeed takes place immediately to pocket depth at the feedrate. If the pocket cannot be milled with a single infeed, an infeed depth must be entered. The milling process is repeated until the pocket depth is reached. If a residual infeed depth of less 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.

Prog. Para.	Sett. Data	Data I/O	Program		Diagnosis	
						V.24 active
AUTOMATIC Reset						

Mill. cycles: Circular pocket



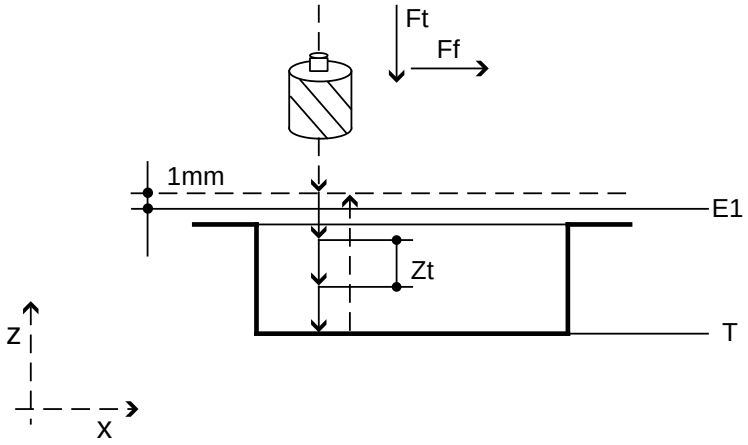
G	R06 = 0
Mw	R22 = 0
Ms	R23 = 0
R	R24 = 0

Milling direction

						DEPTH

Prog. Para.	Sett. Data	Data I/O	Program		Diagnosis	
						V.24 active
AUTOMATIC Reset						

Mill cycles: Circular pocket



Zt	R01 = 0
E1	R02 = 0
T	R03 = 0
Ff	R15 = 0
Ft	R16 = 0

Infeed depth (incr.)

STORE MENU	STORE CHOICE	STORE				

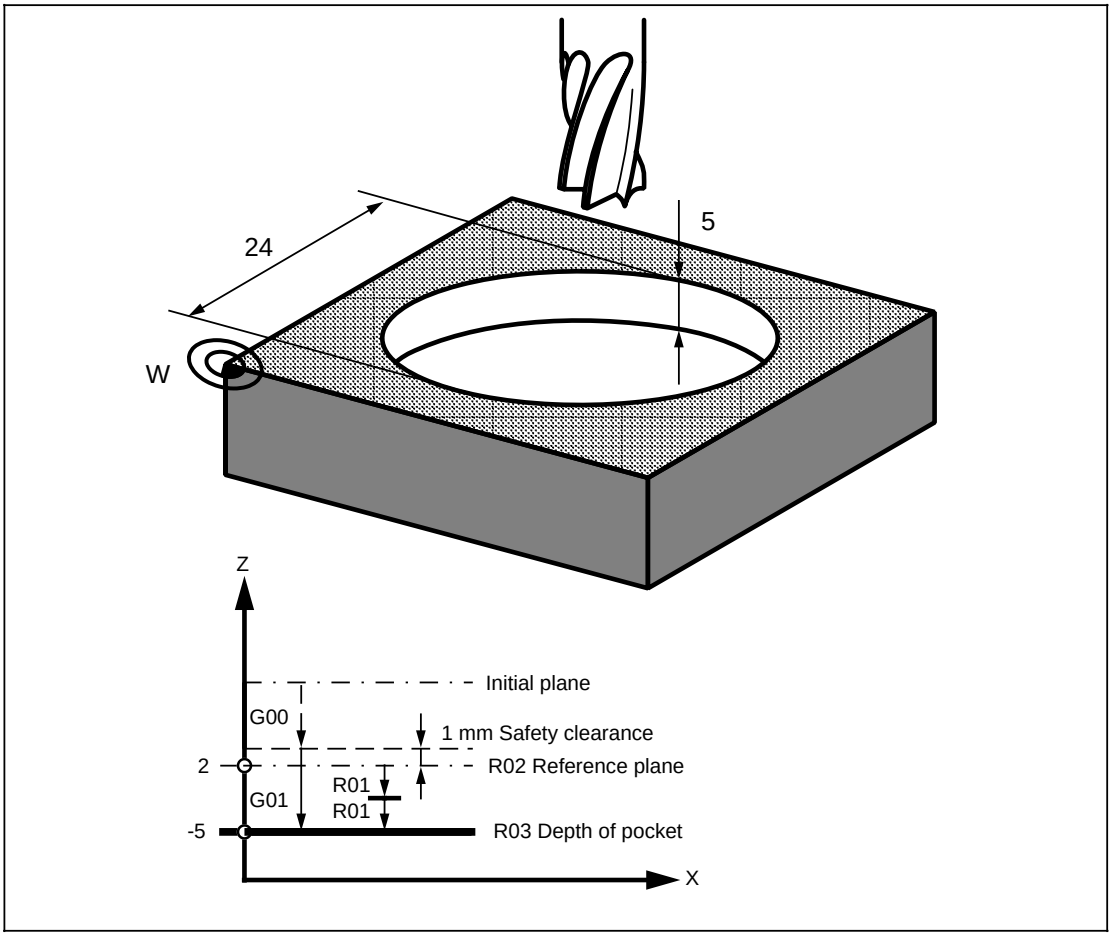
R R24: Pocket radius

If the miller radius is equal to or greater than the pocket radius, error 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.

The buffer memory can be cleared by the program by calling subroutine L999.

The program L999 must 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
                                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  LF
```

2.4 L960 Transfer of zero offset groups

The cycles used to transfer zero offset groups are sold separately and must therefore be ordered as a separate item.

L960_EZS
With this program it is possible to store up to 10 zero offset data blocks with 5 axes in the SINUMERIK 840C/840 and to exchange them with the data in the zero offset memory.

The cycle transfers the input buffer parameters (MIB 200 to MIB 399) to the settable zero offset memory coarse (G54 to G57) and vice versa.

The desired data block is selected with parameter R0. The contents of R0 are automatically stored in machine data 18 by L960. The zero offset data can be entered in the MIB parameters via the program or manually via the input displays. If the transfer of zero offset memory to MIB parameters is selected, parameter R0 must be assigned with 12344.321. The target of the transfer is determined by the current MD 18.

L960_RPA
A maximum of 3 zero offset data blocks with 5 axes can be transferred to R parameters (R240 - R299) and vice versa with this cycle. The parameters are assigned via the system menu display.

Conditions:

These two cycles are not executed in SIMULATION.

Points to watch when using L960 with the SIEMENS measuring cycles are described in the relevant documentation.

The following parameters must be defined before calling L960_EZS:

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

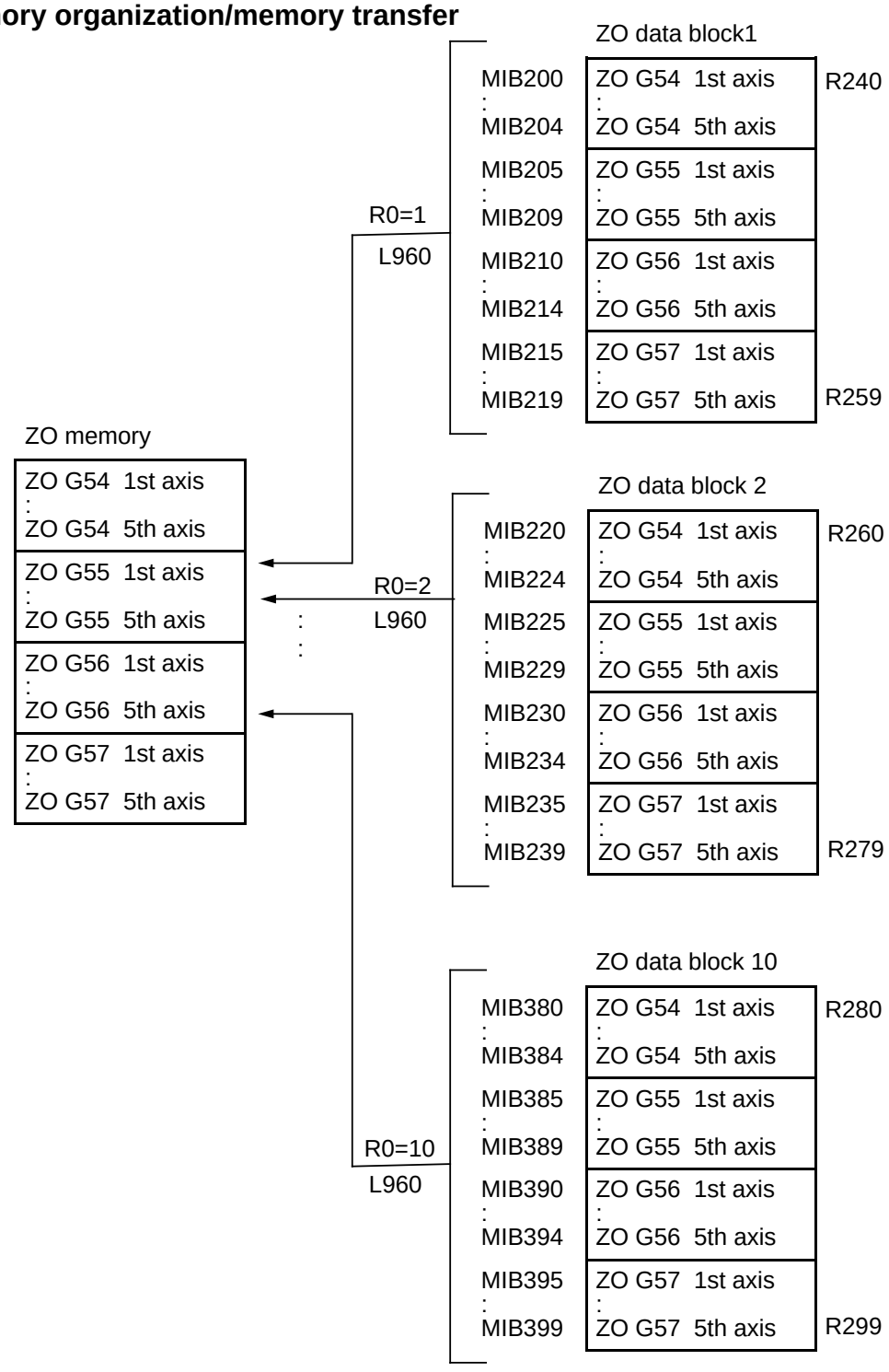
R0=12344.231 Transfer zero offset memory MIB 380 to MIB 399
The current zero offset block is defined in NC MD 18.

The MIB parameters are then reserved for this purpose only.

Program example:

```
%MPF123
N5 R0=2 L960 P1      Transfer 2nd zero offset data block MIB 220 to MIB 239 in zero offset
                     memory
:
:
N55 R0=12344.321 L960 P1      Transfer zero offset memory MIB 220 to MIB 239
:
:
N2000 M30
```

Memory organization/memory transfer



2.4.1 Creating the UMS

Each project contains

- the user menu tree
- a link list L960LIST.LBD
- the cycle in both forms
  - L960_EZS.ZPL Transfer of 10 ZO groups with MIB parameters
  - L960_RPA.ZPL Transfer of 3 ZO groups with R parameters

The following steps must be carried out before a UMS file can be linked:

- Configure the exit from the system menu tree going into the user menu tree. It may be necessary to modify the return to the system menu tree.
- The link list contains the cycle in the MIB parameter version. Enter the required cycle version and add the modified system menu tree.
- The alarm text for alarm no. 4200 "Check definition R (Nxxxx)" must be entered in an alarm text file, this is already available when measuring cycles are used.

END OF SECTION