Typ3 osa

Universal Milling Cycles

Edition

101

Typ3 osa

Universal Milling Cycles

1070 073 795-101 (99.07) GB

© 1999

by Robert Bosch GmbH, Erbach / Germany
All rights reserved, including applications for protective rights.
Reproduction or handing over to third parties are subject to our written permission.

Discretionary charge 10.00 DM

Table of Contents

1 Safety Instructions

Before you start working with the “Universal milling cycles” we recommend that you thoroughly familiarize yourself with the contents of this manual. Store this manual in a place to which all users have access at any time.

1.1 Proper use

This documentation contains information required for the proper use of the control unit. For reasons of clarity, however, it cannot contain all detail about all possible combinations of functions. Likewise, as the control unit is usually part of a larger installation or system, it is impossible to consider every conceivable case of integration, programming or operation.

The Typ3 osa is used to

- activate feed drives, spindles and auxiliary axes of a machine tool via SERCOS interface for the purpose of guiding a processing tool along a programmed path to machine a workpiece (CNC). Furthermore, a PLC is required with appropriate I/O components which – in communication with the actual CNC – controls the machine processing cycles holistically and acts as a technical safety monitor.
- program contours and the machining technology (path feedrate, spindle speed, tool change) of a workpiece.

Any other application is deemed improper use!

The product described

- has been developed, manufactured, tested and documented in compliance with the safety standards. This product poses no danger to persons or property if it is used in accordance with the handling regulations and safety notes prescribed for its configuration, installation and proper operation.
- complies with the requirements of
  - the EMC Directives (89/336/EEC, 93/68/EEC and 93/44/EEC)
  - the Low-Voltage Directive (73/23/EEC)
  - the harmonized standards EN 50081-2 and EN 50082-2
- is designed for operation in industrial environments (emission class A), i.e.
  - no direct connection to public low-voltage power supply,
  - connection to the medium- or high-voltage system via a transformer.

  In residential environments, in trade and commerce as well as small enterprises class A equipment may only be used if it does not inadmissibly interfere with other equipment.

☞ **This is a class A device which may cause radio interference in residential environments. In this case, the operator may be required to take suitable countermeasures and to bear the cost of the same.**

The faultless, safe functioning of the product presupposes proper transport, storage, erection and installation as well as careful operation.

CAUTION
This product as well as our operating, programming, visualization and control software have been examined for ”Year 2000 compliance” and have passed the required tests.

Please note that the ”Year 2000 compliance” may be lost if additional software packages are installed, and that any such event would be the customer’s responsibility.

Examples for the possible loss of the ”Year 2000 compliance” known to us include:

1. **If other software packages are installed, some of the software components supplied by us may be overwritten (e.g. DLL’s).**
2. **The additionally installed software may read the date directly from the BIOS or even from the time module. To correct the date after the turn of the millennium it will be sufficient to restart the unit.**
3. **Some components of Microsoft products (e.g. the File Manager and the Explorer) have limited ”Year 2000 compliance” capabilities. More information is available on the World Wide Web pages of Microsoft at http://www.eu.microsoft.com/ithome/topics/year2k/product/product.htm .**

1.2 Qualified personnel

The requirements as to qualified personnel depend on the qualification profiles described by ZVEI (central association of the electrical industry) and VDMA (association of German machine and plant builders) in:
Weiterbildung in der Automatisierungstechnik
edited by: ZVEI and VDMA
MaschinenbauVerlag
Postfach 71 08 64
D-60498 Frankfurt.

The present manual is designed for NC programming personnel and NC project engineers.

Programming, start and operation as well as the modification of programs or program parameters may only be performed by properly trained personnel! This personnel must be able to judge potential hazards arising from programming, program changes and in general from the mechanical, electrical, or electronic equipment.

The programming of the control unit may enable you to influence axis movements (e.g. positions, etc.), machining technology (e.g. feed, speed, etc.) and the machining process (e.g. tool change, compensations, issuing of auxiliary functions, etc.).
All this presupposes firstly a general knowledge of programming (e.g. for the logical program sequence, etc.) and, secondly, knowledge of the technology of the process to be performed.

Unauthorized interference in the program code or ignoring warning signs attached to the components, or non-compliance with the warning notes given in this manual may result in serious bodily injury or material damage.

Only technicians who are familiar with the contents of this manual may compile programs or adapt existing programs.

Such personnel are

- those who, being well trained and experienced in their field and familiar with the relevant standards, are able to analyze the work to be carried out and recognize any hazards.
- those who have acquired the same amount of expert knowledge through years of experience that would normally be acquired through formal technical training.

Please note our comprehensive range of training courses.
Our training center will be pleased to provide you with further information, telephone: +49 (0) 6062 78-258.

1.3 Safety markings on products

Warning of dangerous electrical voltage!

Warning of danger caused by batteries!

Components sensitive to electrostatic discharge!

Disconnect from mains before opening!

Pin for connecting PE conductor only!

Connection of shield conductor only

1.4 Safety instructions in this manual

DANGER
This symbol is used wherever an insufficient or lacking compliance with instructions can result in **personal injury**.

CAUTION
This symbol is used wherever an insufficient or lacking compliance with instructions can result in **damage to equipment or data files**.

☞ This symbol is used to inform the user of special features.

★ This symbol is used if user activities are required.

1.5 Safety instructions concerning the product described

DANGER
Danger of death due to inadequate EMERGENCY-STOP devices!
EMERGENCY-STOP devices must be active and within reach in all system modes. Releasing an EMERGENCY-STOP device must not result in an uncontrolled restart of the system!
First check the EMERGENCY-STOP circuit, then switch the system on!

DANGER
Incorrect or undesired axis movement!
First, new programs should be tested carefully without axis movement! For this purpose, the control unit offers the possibility of inhibiting axis movements and/or auxiliary function outputs by appropriate softkeys in the “automatic“ group operating mode.

DANGER
Tool or axis movements!
Feed and spindle motors generate very powerful mechanical forces and can accelerate very quickly due to their high dynamics.

- **Always stay outside the danger area of the machine when it is running!**
- **Do not ever deactivate the safety-relevant functions of the unit!**
- **Report any malfunction of the unit to your servicing and repairs department immediately!**

CAUTION
Only spare parts approved by Bosch may be used!

CAUTION
Danger to the module!
All ESD protection measures must be observed when using the module! Prevent electrostatic discharges!

The following protective measures must be observed for modules and components sensitive to electrostatic discharge (ESD)!

- The personnel resposible for storage, transport, and handling must have been trained for ESD protection.
- ESD-sensitive components must be stored and transported in their prescribed protective packaging.
- ESD-sensitive components may only be handled at special ESD-workplaces.

- Personnel, working surfaces, as well as all equipment and tools which get in contact with ESD-sensitive components must have the same potential (e.g., by grounding).
- Wear an approved grounding bracelet. The grounding bracelet must be connected with the working surface through a cable with an integrated resistor of 1 MΩ.
- ESD-sensitive components must by no means get in contact with chargeable objects, including most plastic materials.
- When ESD-sensitive components are installed in or removed from equipment, the equipment must be de-energized.

1.6 Dokumentation, Version und Warenzeichen

Documentation

The present manual provides information about programming “Universal Milling Cycles”.

Overview of available documentation	Part no.	
	German	English
Interface conditions for project engineering and maintenance	1070 073 704	1070 073 736
Operating instructions Standard operator interface	1070 073 726	1070 073 739
Operating instructions – Diagnostics Tools	1070 073 779	1070 073 780
DIN programming instructions for programming to DIN 66025	1070 073 725	1070 073 738
CPL programming instructions	1070 073 727	1070 073 740
ICL700 system description, program structure of the integrated PLC	1070 073 706	1070 073 737
ICL700 project planning manual, software interfaces and CNC interface signals of the integrated PLC	1070 073 728	1070 073 741
MACODA operation and configuration of the machine parameters	1070 073 705	1070 073 742
Tool Management Parameter Setting	1070 073 782	–
Software PLC Development Environment for Windows NT	1070 073 783	1070 073 792
Measuring cycles for touch-trigger switching probes	1070 073 788	1070 073 789
Universal Milling Cycles	–	1070 073 795

Version

☞ **This manual refers to the following product version:**
Software: V4.3.16

☞ **The current release number of the individual software modules can be viewed by selecting the ”Control-Diagnostics” softkey in the ”Diagnostics” group operating mode.**

☞ **The software version of Windows 95 or Windows NT may be displayed as follows:**
1. **Click with right mouse key on the ”My Computer” icon on your desktop**
2. **Select menu item ”Properties”.**

Modifications

Modifications in the present manual as compared to the previous edition are marked by black vertical bars in the margin.

This symbol is used in illustrations to call your attention to any modifications made.

Trademarks

All trademarks of software installed on Bosch products upon delivery are the property of the respective manufacturer.

Upon delivery, all installed software is copyright-protected. The software may only be reproduced with the approval of Bosch or in accordance with the license agreement of the respective manufacturer.

MS-DOS® and Windows™ are registered trademarks of Microsoft® Corporation.

2 Special Features

Material is removed in the pocket cycles by following contours. For prema-chined workpieces, a blank overmeasure can be defined so that you can ma-chine without empty movements by beginning at a certain distance from the pocket centre rather than from the pocket centre itself.

The pocket can be machined clockwise or counter-clockwise (direction of rotation of the tool) in the pocket cycles. This means that both down milling and up milling are possible, depending on the direction of rotation of the spindle.

The direction of rotation of the tool can be chosen differently for roughing and for finishing.

The pocket cycles can be used as separate or combined roughing/finishing cycles, controlled by an execution parameter. Different finishing overmea-sures can be specified for the wall and base.

The milling cycles are invoked without active tool path compensation be-cause the active tool radius is offset internally in the cycles.

All infeed movements in rapid traverse maintain a safety clearance of 1mm. This value appears on a solid background in the cycles, and can be changed by modifying the value assigned to the #CCVS parameter in the P999999100 subroutine.

When base finishing, the finishing tool – for technological reasons – does not touch the wall that has already been roughed. The value for this wall clear-ance (0.01mm) appears on a solid background in the cycles, and can be changed by modifying the value assigned to the #CCVW parameter in the P999999100 subroutine.

After base finishing, i.e. before wall finishing, the finishing tool is retracted for technological reasons. The value for this base clearance (0.01mm) appears on a solid background in the cycles, and can be changed by modifying the value assigned to the #CCVB parameter in the P999999100 subroutine.

An overlap of at least 10% is maintained in the pocket cycles. This value ap-pears on a solid background in the cycles, and can be changed by modifying the value assigned to the #CCVL parameter in the P999999101 subroutine.

Deactivation conditions:

At the end of the cycle, the following conditions apply for all milling cycles:

- The tool is located, with spindle rotating, at safety clearance above the infeed plane.
- Rapid traverse GO is active.
- The last feed programmed in the milling cycle is active.
- Depending on the situation before the cycle is invoked, either absolute dimension (G90) or incremental dimension (G91) is active.
- The polar plane corresponds to the milling plane, with the pole in the centre of the pocket.

Variable assignment:

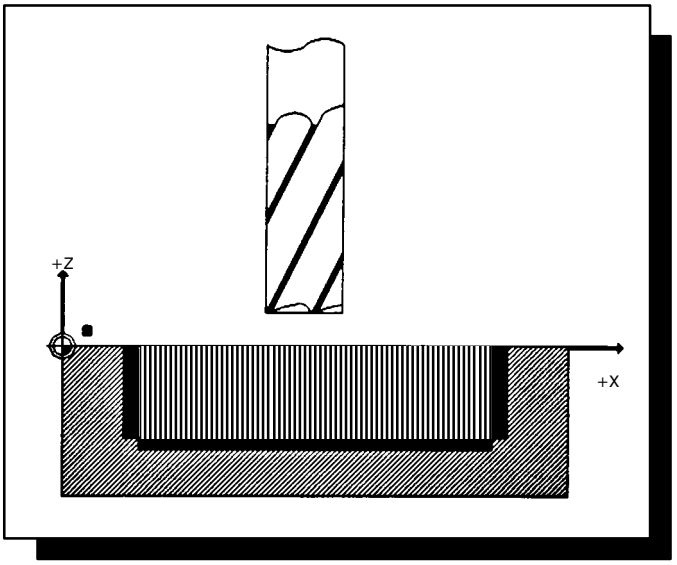
All global variables used in the cycles begin with the letters “#CC”.

Do not use this combination of letters to begin variables in your own CPL programs.

3 Definitions

3.1 Roughing/Finishing

3.1.1 Combined



Combined means:

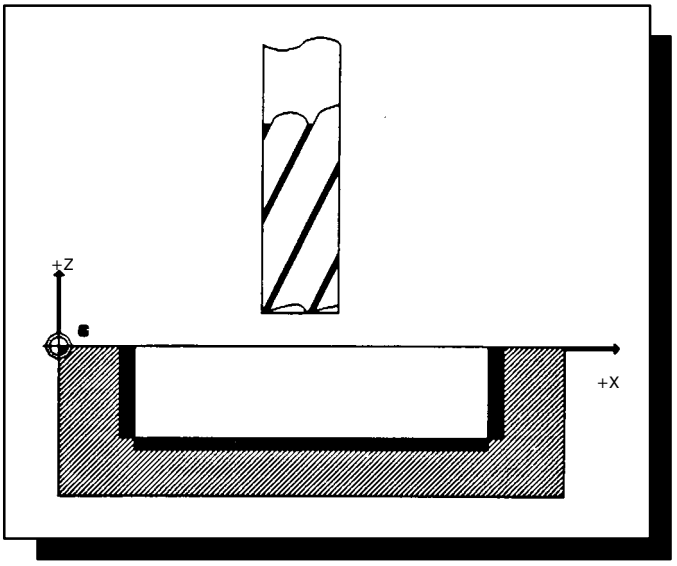
One single tool does the following in succession:

- roughing
- base finishing and
- wall finishing.

This parameter is only active in the following cycles:

- rectangular pocket
- circular pocket and
- groove.

3.1.2 Roughing



Roughing means:

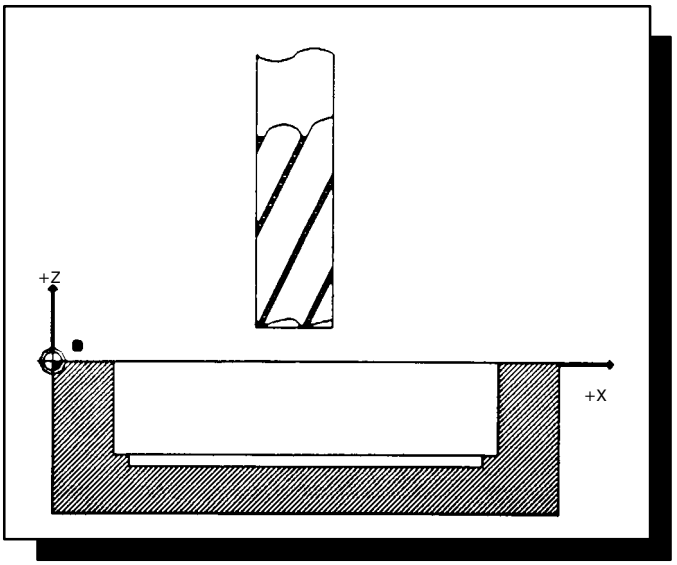
The tool performs roughing only.

This parameter value lets you rough and finish in separate steps using different tools.

This parameter is only active in the following cycles:

- rectangular pocket
- circular pocket and
- groove.

3.1.3 Finishing



Finishing means:

The tool performs finishing only.

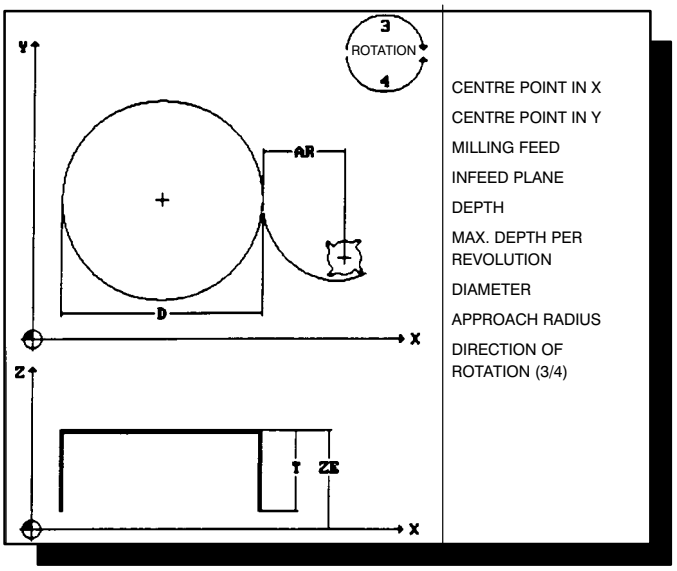
This parameter value lets you rough and finish in separate steps using different tools.

This parameter is only active in the following cycles:

- rectangular pocket
- circular pocket and
- groove.

3.2 Centre Point

3.2.1 Plane G17



Milling plane: X/Y (centre point)

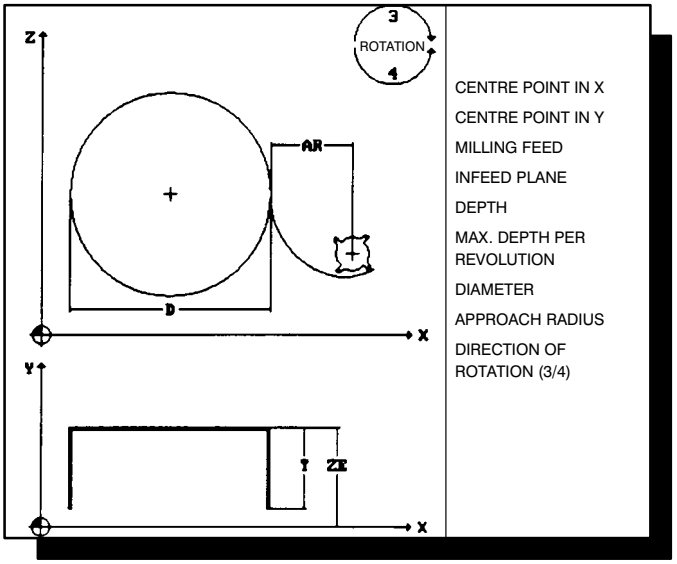
Infeed plane: Z (drilling axis)

The centre point dimensions specify, for each axis, the distance between the workpiece centre and the axis datum.

The dialogue screen automatically switches to the associated infeed plane when you choose the infeed axis.

The parameter values are entered as absolute dimensions (in mm).

3.2.2 Plane G18



Milling plane: Z/X (centre point)

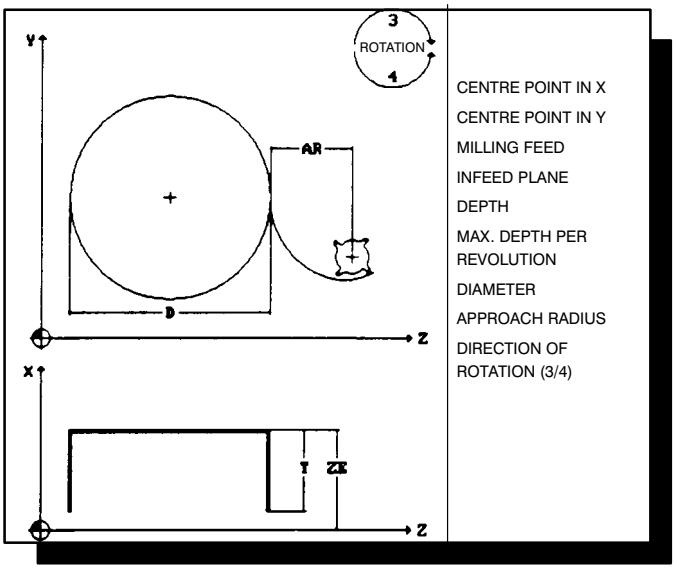
Infeed plane: Y (drilling axis)

The centre point dimensions specify, for each axis, the distance between the workpiece centre and the axis datum.

The dialogue screen automatically switches to the associated infeed plane when you choose the infeed axis.

The parameter values are entered as absolute dimensions (in mm).

3.2.3 Plane G19



Milling plane: Y/Z (centre point)

Infeed plane: X (drilling axis)

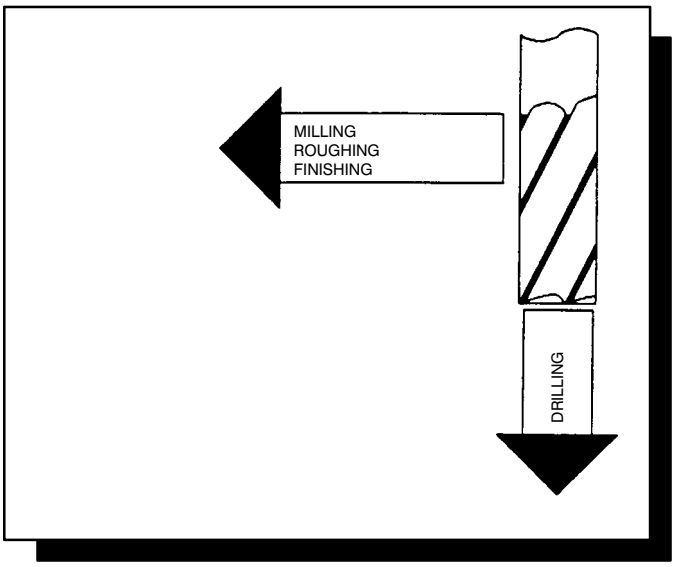
The centre point dimensions specify, for each axis, the distance between the workpiece centre and the axis datum.

The dialogue screen automatically switches to the associated infeed plane when you choose the infeed axis.

The parameter values are entered as absolute dimensions (in mm).

3.3 Feed

3.3.1 Drilling Feed



The drilling feed acts along the drilling axis.

The drilling feed is only active in the following cycles:

- rectangular pocket
- circular pocket
- groove and
- end groove.

Input in mm/min.

3.3.2 Roughing Feed

The roughing feed acts on the milling plane.

The roughing feed is only active during roughing in the following cycles:

- rectangular pocket
- circular pocket and
- groove.

Input in mm/min.

3.3.3 Finishing Feed

The finishing feed acts on the milling plane.

The finishing feed is only active during finishing in the following cycles:

- rectangular pocket
- circular pocket and
- groove.

Input in mm/min.

3.3.4 Milling Feed

The milling feed acts on the milling plane.

The milling feed is only active during milling in the following cycles:

- end groove
- borehole
- inside frame
- outside frame
- tenon
- inside circle segment and
- outside circle segment.

Input in mm/min.

3.4 Infeed Plane

The Infeed Plane parameter corresponds to the top edge of the workpiece.

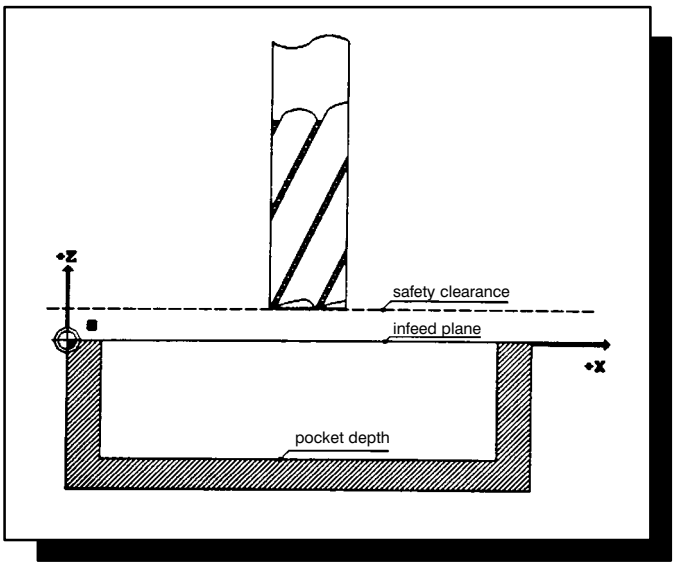
The variable value is entered as an absolute dimension (in mm).

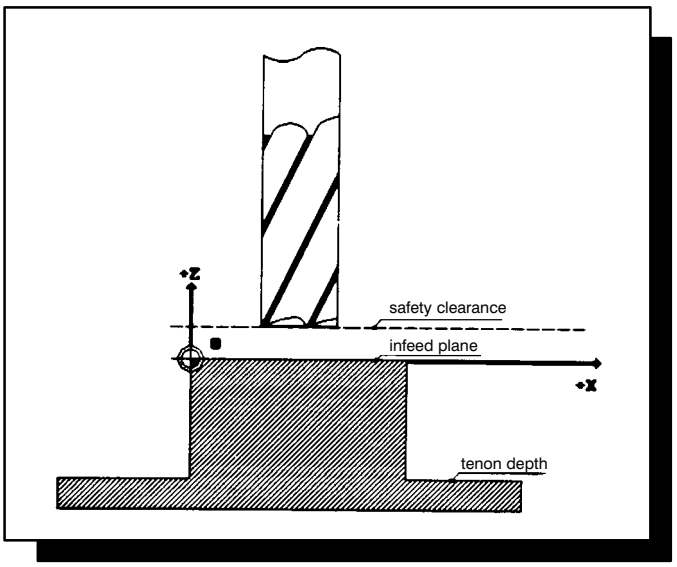
Positive and negative values are allowed.

When approaching the infeed plane, a safety clearance is automatically maintained.

(defined as #CCVS in the P999999100 cycle – see TECHNOLOGY PARAMETERS)

The infeed axis switches from rapid traverse to drilling feed when it is the safety clearance away from the infeed plane.





3.5 Depth

The Depth parameter specifies the final dimension of the pocket base in the cycles:

- rectangular pocket
- circular pocket and
- groove.

The depth parameter specifies the height in the cycles:

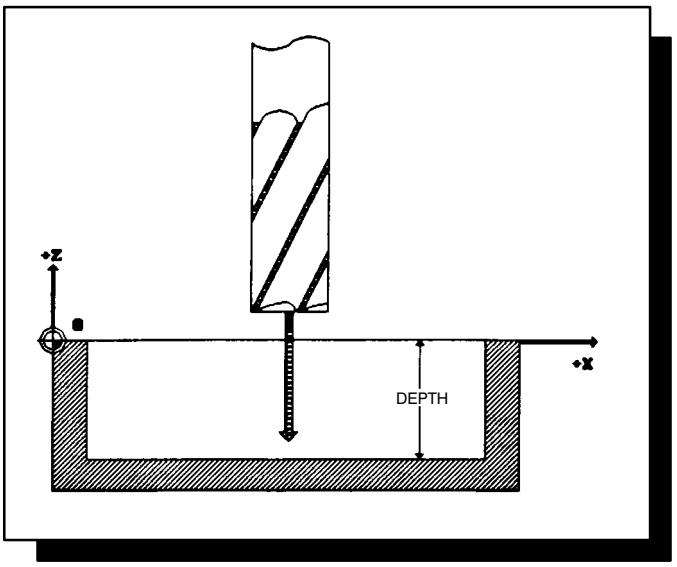
- end groove
- borehole
- inside frame and
- inside circle segment.

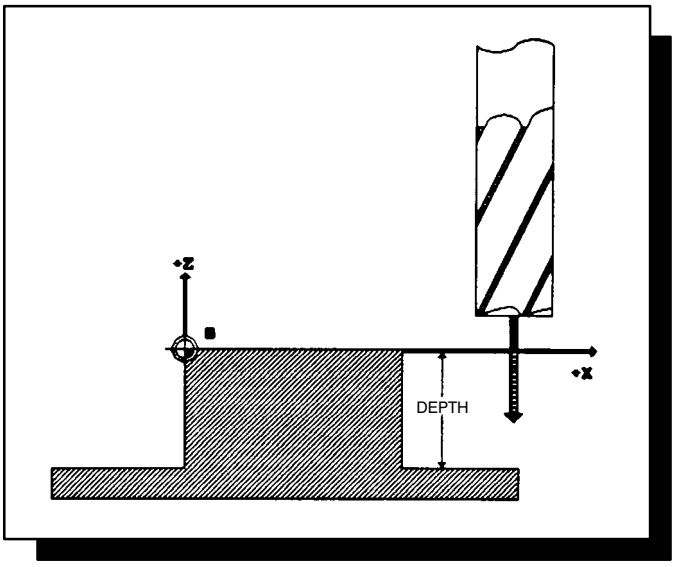
The depth is counted in negative direction starting from the infeed plane.

The parameter value is entered as an incremental dimension (in mm). Only positive values are allowed.

The infeed axis and infeed direction are switched over using G78.

The milling plane is selected using G17 or G18, or G19 or G220.
(The X/Y level appears on a solid background by default.)





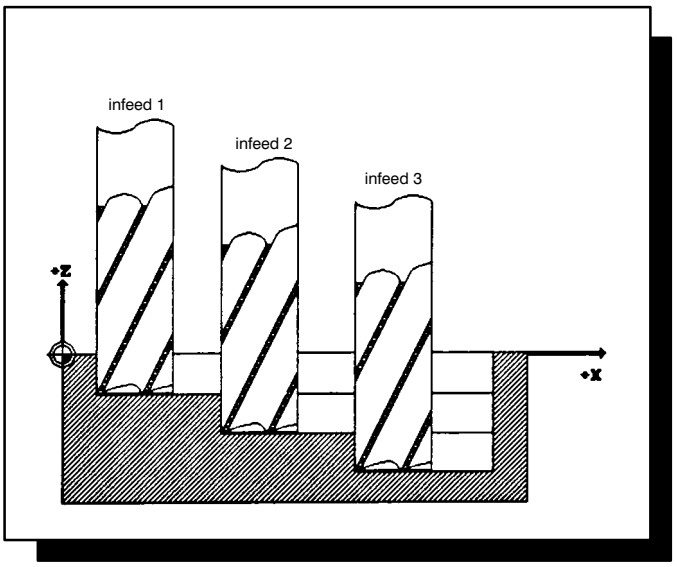
3.6 Max. Depth per Revolution

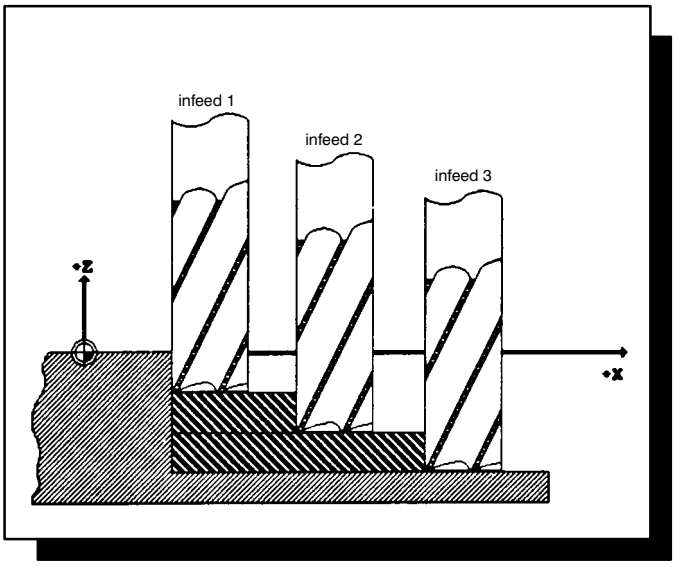
The Max. Depth per Revolution parameter specifies the maximum depth infeed that is allowed in any one cutter revolution.

The effective depth infeed is the result of even cut distribution combined with maximum infeed depth.

Only positive parameter values (in mm) are allowed.

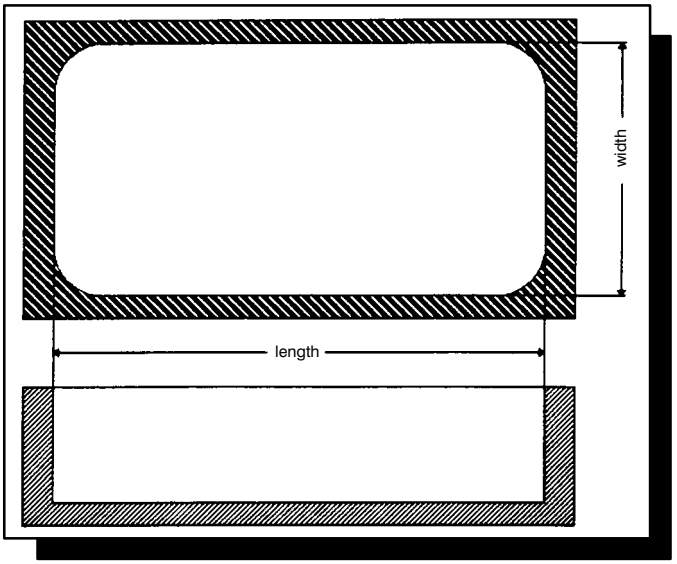
The parameter is active in all cycles, both roughing and finishing.





3.7 Length/Width

3.7.1 Rectangular Pocket/ Inside Frame



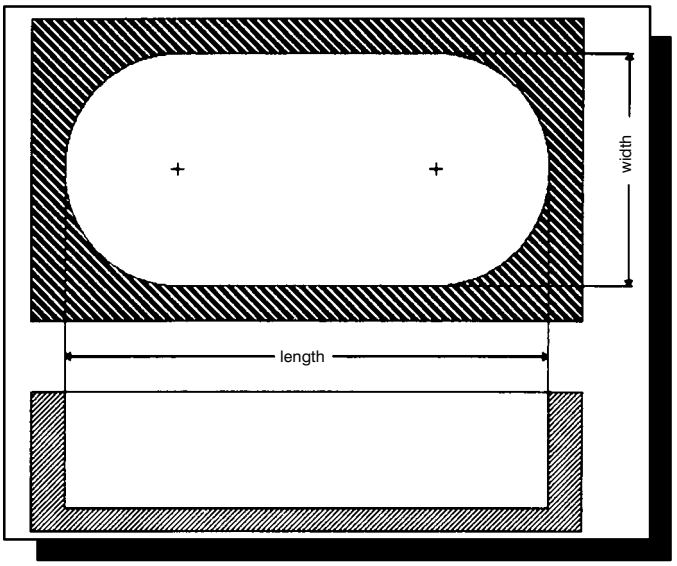
These parameters determine the length and width of the rectangular pocket and inside frame.

(Finished dimension)

The cycle monitors to determine if the width is greater than the length.

The parameter values are entered as incremental dimensions (in mm). Only positive values are allowed.

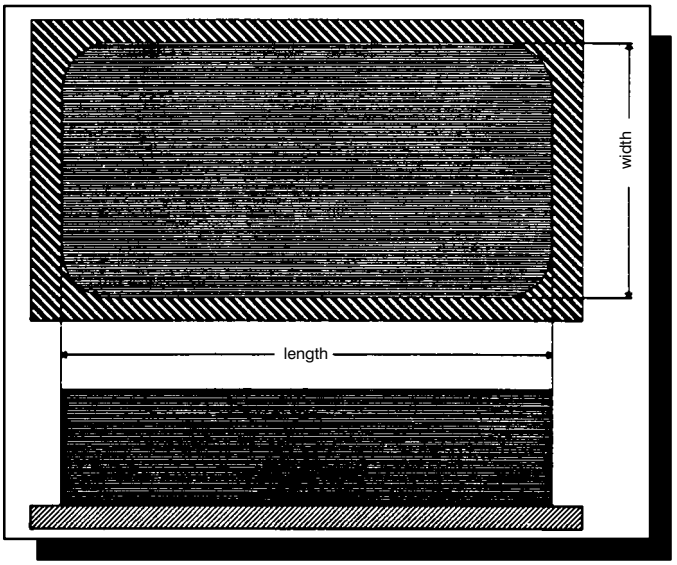
3.7.2 Groove



These parameters determine the length and width of the groove.
(Finished dimension)

The parameter values are entered as incremental dimensions (in mm). Only positive values are allowed.

3.7.3 Outside Frame



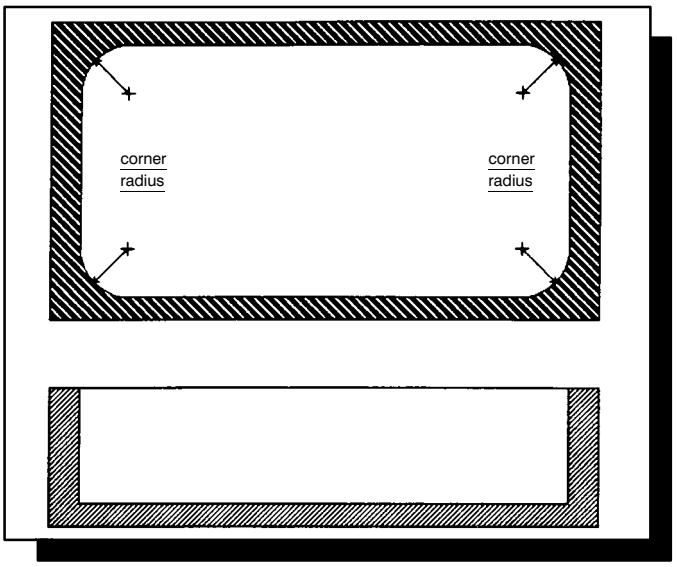
These parameters determine the length and width of the outside frame.
(Finished dimension)

The cycle monitors to determine if the width is greater than the length.

The parameter values are entered as incremental dimensions (in mm). Only positive values are allowed.

3.8 Corner Radius

3.8.1 Rectangular Pocket/ Inside Frame

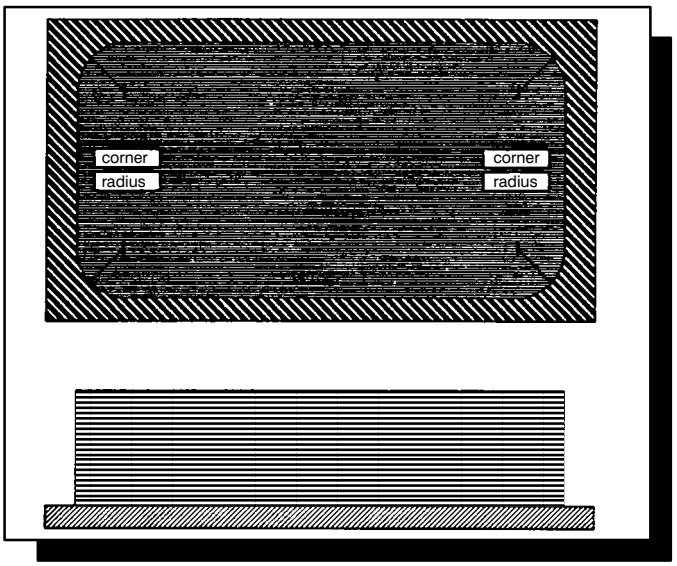


These parameters determine the length and width of the rectangular pocket and inside frame.
(Finished dimension)

The cycle monitors to determine if the cutter radius of the selected tool is smaller than the corner radius minus finishing overmeasure.

The parameter values are entered as incremental dimensions (in mm). Only positive values are allowed.

3.8.2 Rectangular Pocket/ Outside Frame



This parameter determines the corner radius of the outside frame.
(Finished dimension)

The parameter value is entered as an incremental dimension (in mm). Only positive values are allowed.

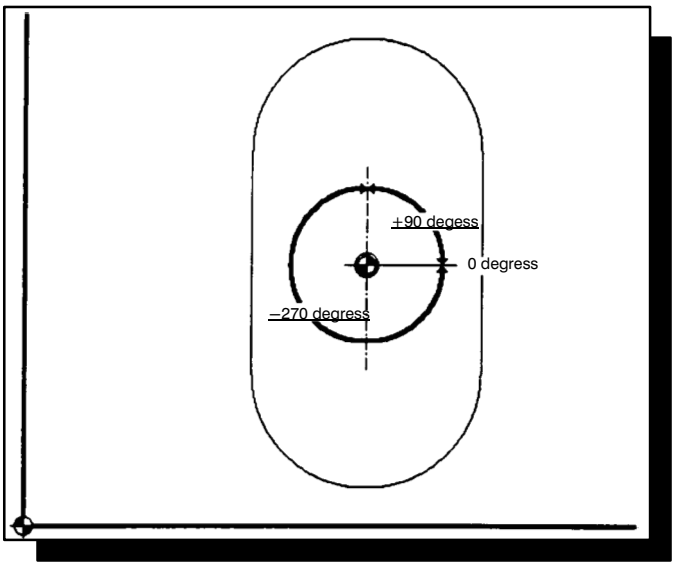
3.9 Angle

3.9.1 Groove

This parameter is used to rotate the cycle geometry on a plane.

The angle is specified as an absolute value in degrees. (Mathematically positive direction of rotation)

Values can be positive, negative or zero.

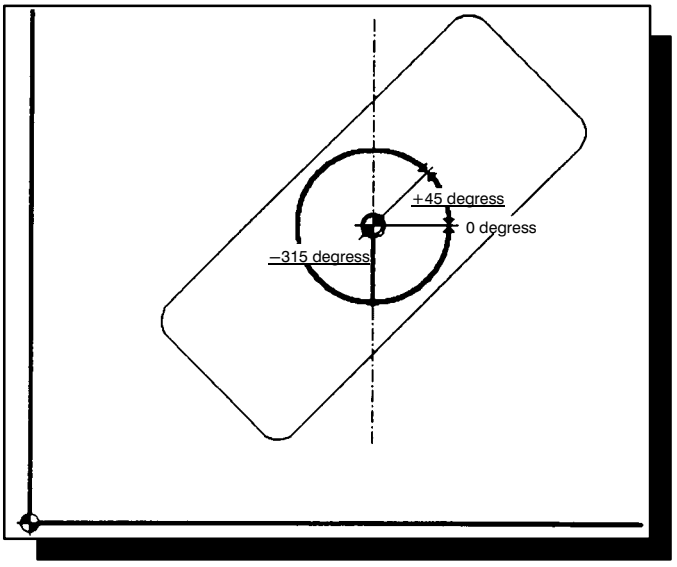


3.9.2 Rectangular Pocket Inside Frame Outside Frame

These parameters are used to rotate the cycle geometry on a plane.

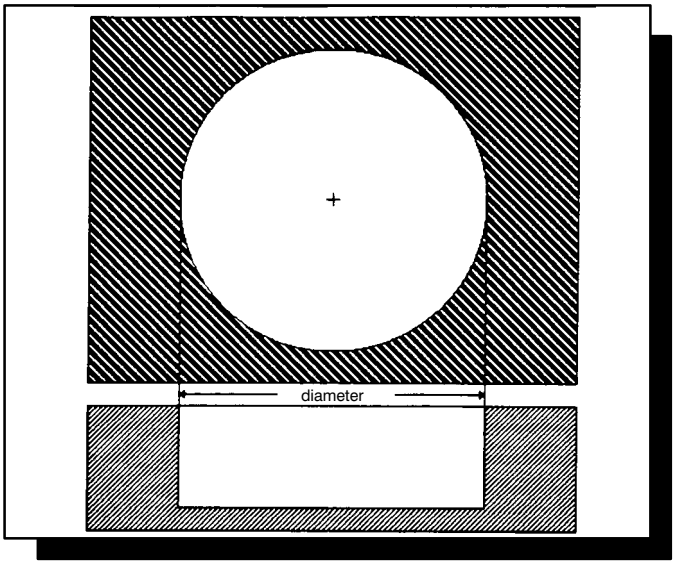
The angle is specified as an absolute value in degrees. (Mathematically positive direction of rotation)

Values can be positive, negative or zero.



3.10 Diameter

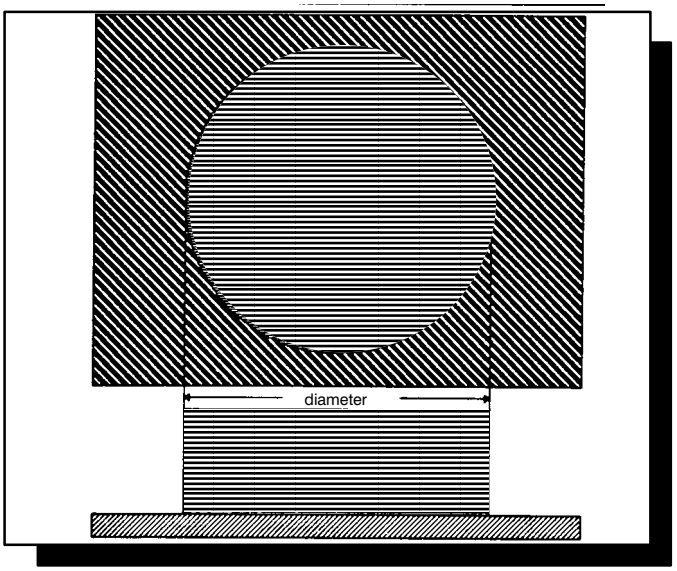
3.10.1 Circular Pocket/Borehole



These parameters determine the diameter of the circular pocket and borehole.
(Finished dimension)

The parameter value is entered as an incremental dimension (in mm). Only positive values are allowed.

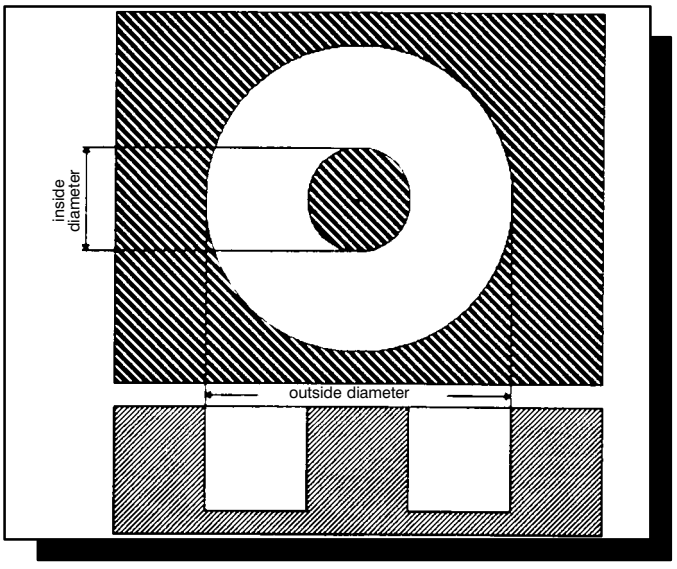
3.10.2 Tenon



This parameter determines the diameter of the tenon.
(Finished dimension)

The parameter value is entered as an incremental dimension (in mm). Only positive values are allowed.

3.10.3 End Groove/ Inside Diameter Outside Diameter



These parameters determine the diameter of the end groove.
(Finished dimension)

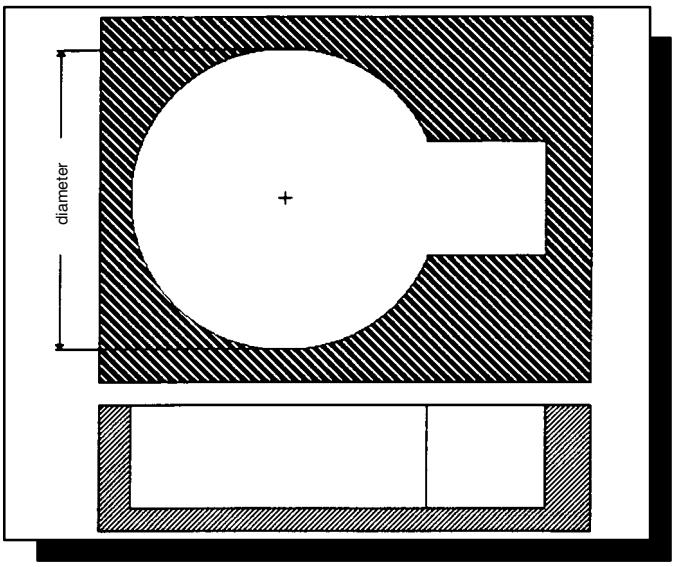
The groove width is half of the diameter difference.

The parameter value is entered as an incremental dimension (in mm). Only positive values are allowed.

3.10.4 Inside Circle Segment

This parameter determines the diameter of the inside circle segment.
(Finished dimension)

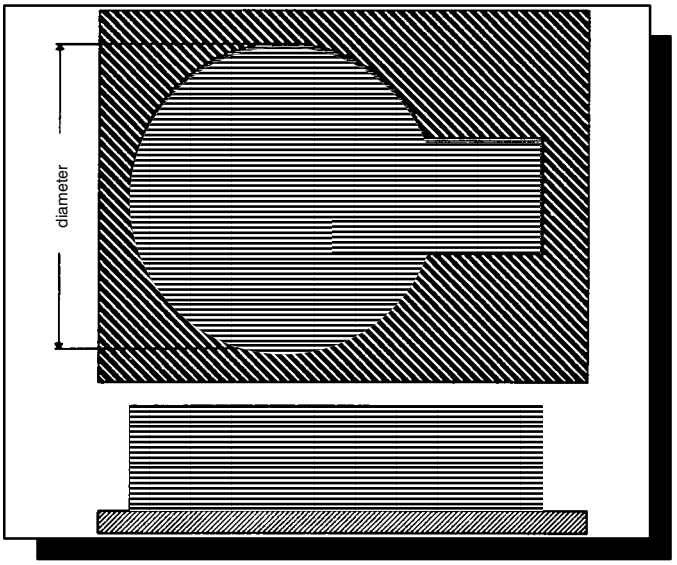
The parameter value is entered as an incremental dimension (in mm). Only positive values are allowed.



3.10.5 Outside Circle Segment

This parameters determines the diameter of the outside circle segment.
(Finished dimension)

The parameter value is entered as an incremental dimension (in mm). Only positive values are allowed.



3.11 Direction of Rotation

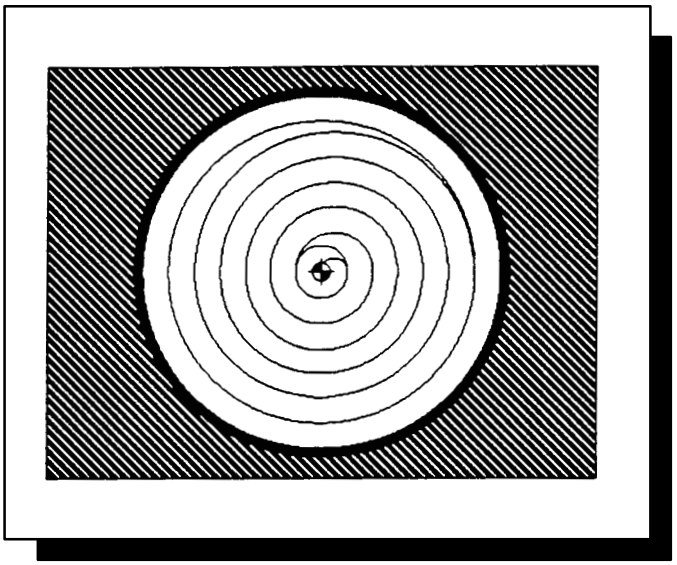
Direction of Rotation=3
Roughing Rotation =3
Finishing Rotation =3

This parameter determines the cutting direction clockwise during machining.

In the cycles:

- rectangular pocket
- circular pocket and
- groove,

a different direction of rotation can be chosen for roughing and finishing. In all other cycles there is only one direction of rotation.



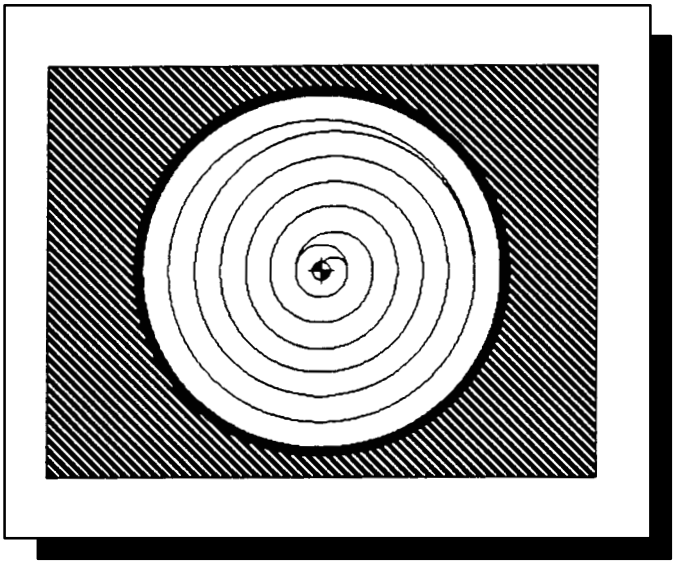
Direction of Rotation=4
Roughing Rotation =4
Finishing Rotation =4

This parameter determines the cutting direction counter-clockwise during machining.

In the cycles:

- rectangular pocket
- circular pocket and
- groove,

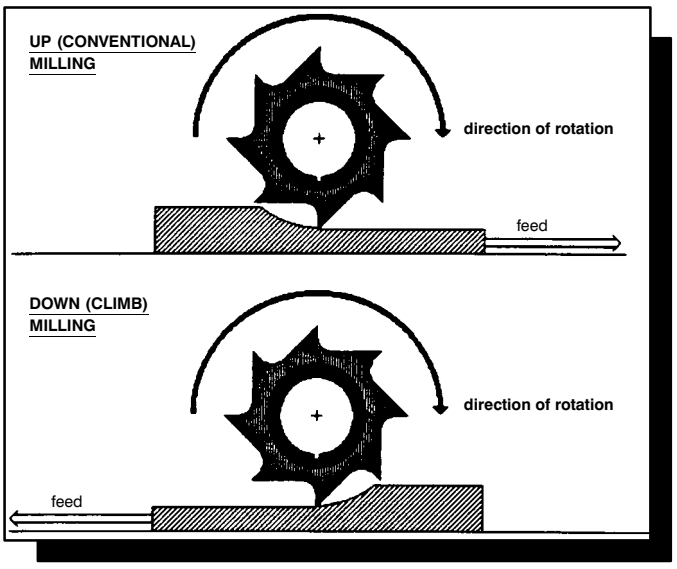
a different direction of rotation can be chosen for roughing and finishing. In all other cycles there is only one direction of rotation.



3.11.1 Down/Up Milling

Corresponding to the chosen direction of rotation and the chosen spindle speed, milling is either

- up (conventional) or
- down (climb).

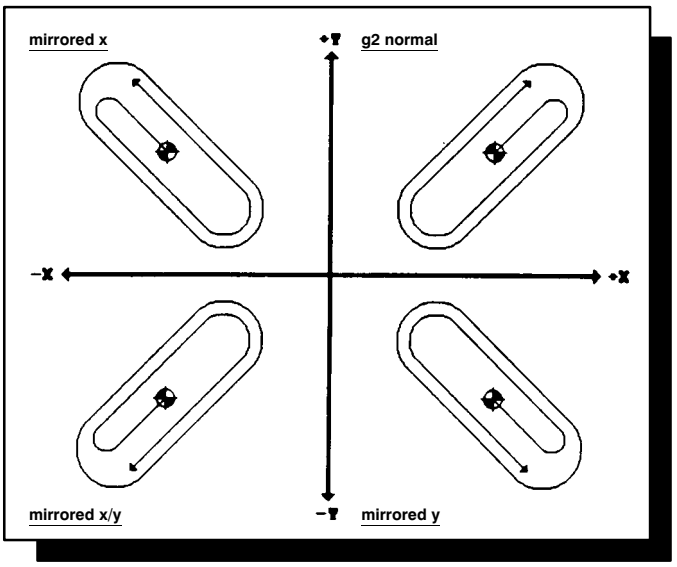


3.11.2 Direction of Rotation during Mirroring

The direction of rotation during mirroring is not adapted to the mirrored geometry.

When mirroring about just one axis, up milling becomes down milling and vice versa.

The up or down milling only remains the same when mirroring about two axes.



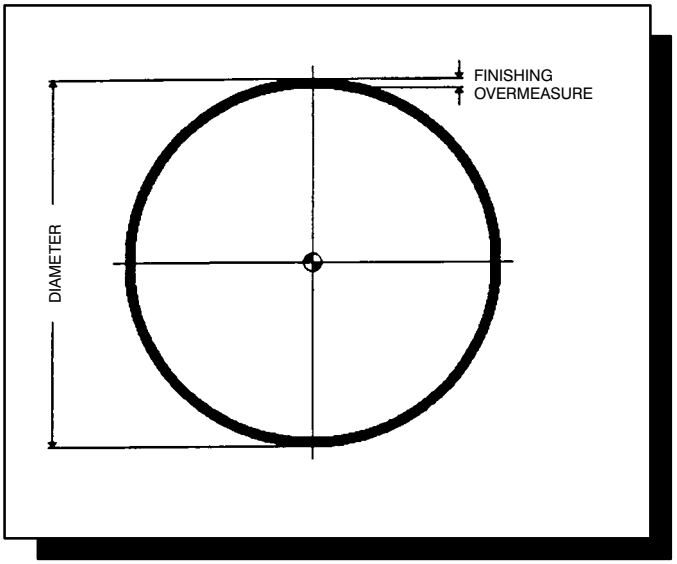
3.12 Finishing Overmeasure/Wall

3.12.1 Circular Pocket

The parameter value is entered as an incremental dimension (in mm). Only positive values or zero are allowed.

If you enter zero, finishing continues to the finished dimension, i.e. the wall is not finished.

This also applies when the cycle is invoked with the parameter Roughing/Finishing = 2.

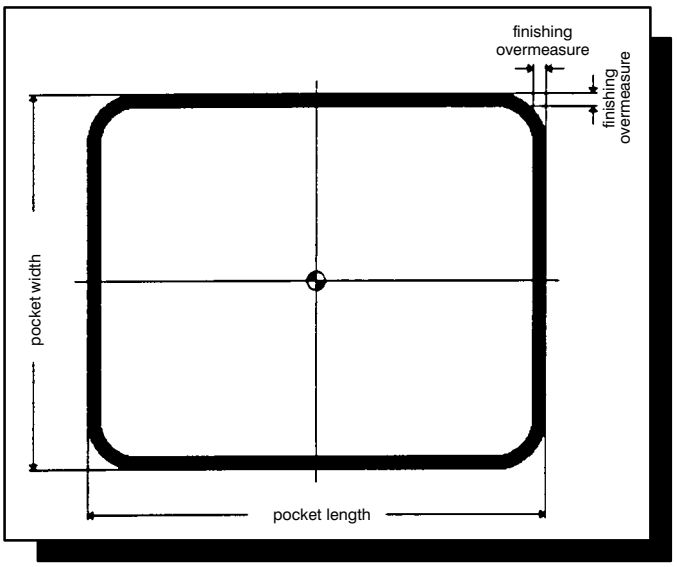


3.12.2 Rectangular Pocket

The parameter value is entered as an incremental dimension (in mm). Only positive values or zero are allowed.

If you enter zero, finishing continues to the finished dimension, i.e. the wall is not finished.

This also applies when the cycle is invoked with the parameter Roughing/Finishing = 2.

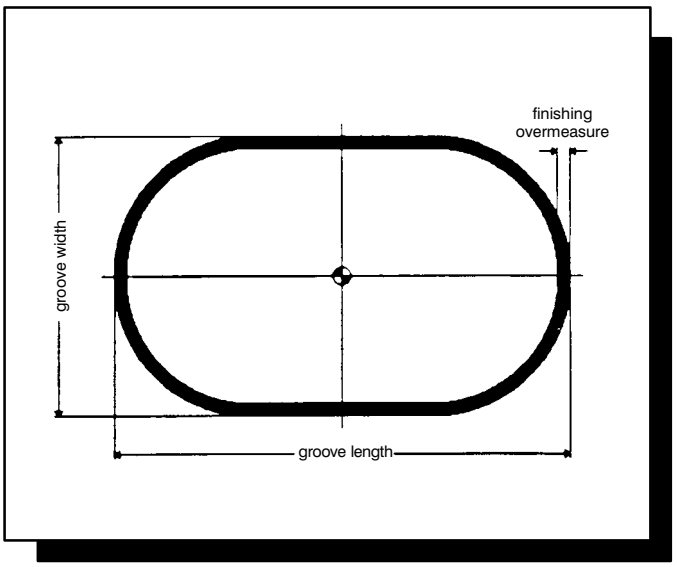


3.12.3 Groove

The parameter value is entered as an incremental dimension (in mm). Only positive values or zero are allowed.

If you enter zero, finishing continues to the finished dimension, i.e. the wall is not finished.

This also applies when the cycle is invoked with the parameter Roughing/Finishing = 2.

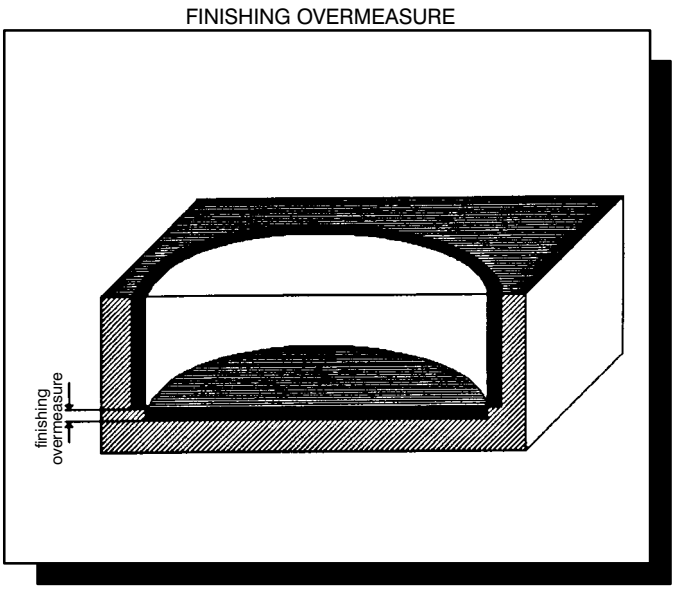


3.13 Finishing Overmeasure/Base

3.13.1 Circular Pocket

The parameter value is entered as an incremental dimension (in mm). Only positive values or zero are allowed.

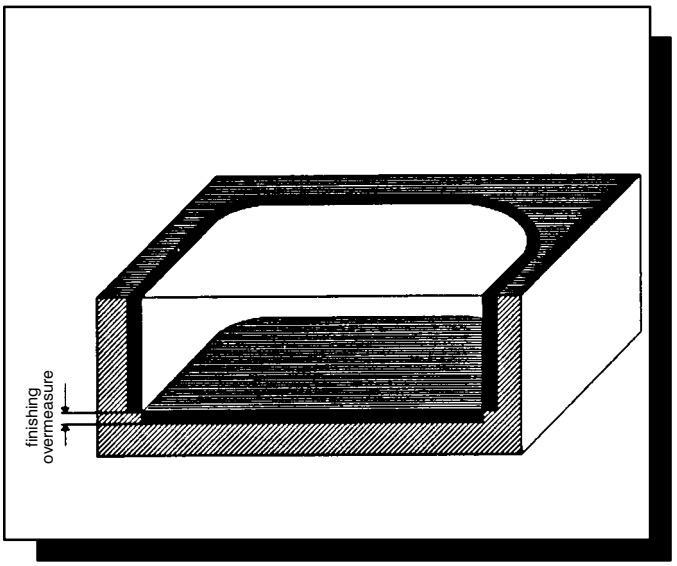
If you enter zero, depth finishing continues to the finished dimension, i.e. the base is not finished.



3.13.2 Rectangular Pocket

The parameter value is entered as an incremental dimension (in mm). Only positive values or zero are allowed.

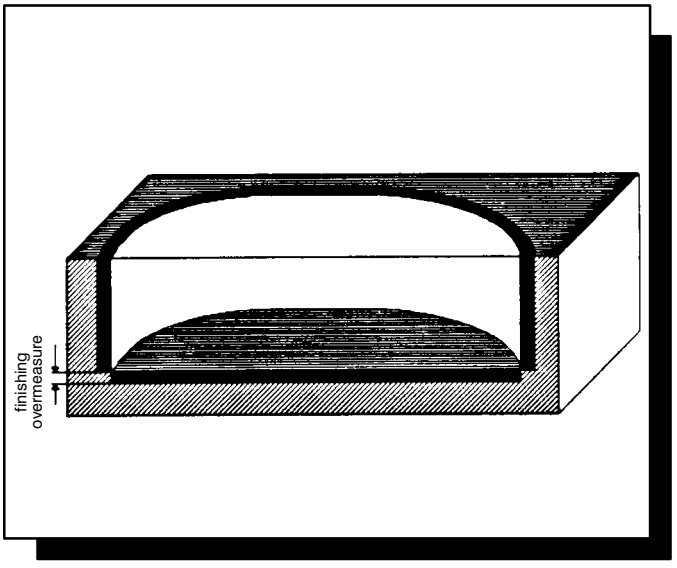
If you enter zero, depth finishing continues to the finished dimension, i.e. the base is not finished.



3.13.3 Groove

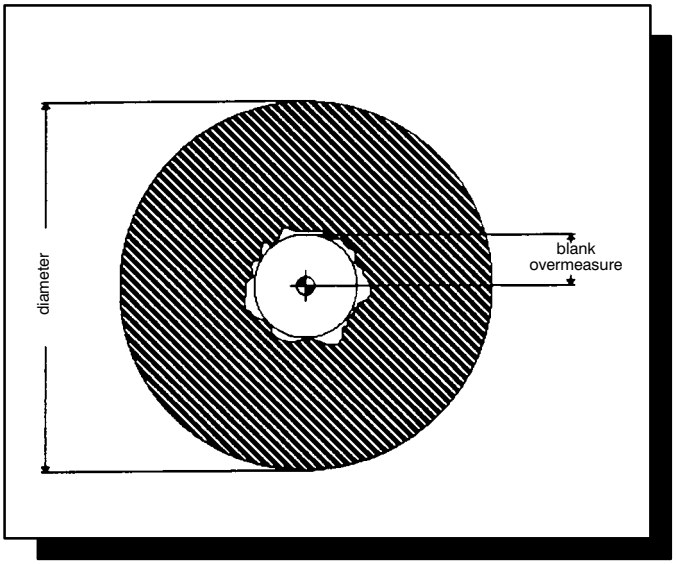
The parameter value is entered as an incremental dimension (in mm). Only positive values or zero are allowed.

If you enter zero, depth finishing continues to the finished dimension, i.e. the base is not finished.



3.14 Blank Overmeasure

3.14.1 Circular Pocket



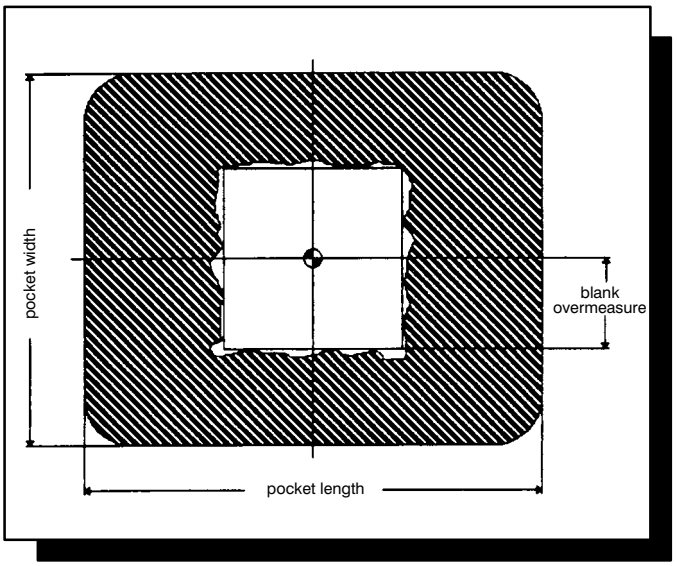
This parameter is active during both roughing and finishing.

The parameter value is entered as an incremental dimension (in mm) and calculated starting at the centre point. Only positive values and zero are allowed.

Entering zero means machining starts at the centre point.

Maximum value
1/2 diameter minus (finishing overmeasure/wall plus cutter radius).

3.14.2 Rectangular Pocket



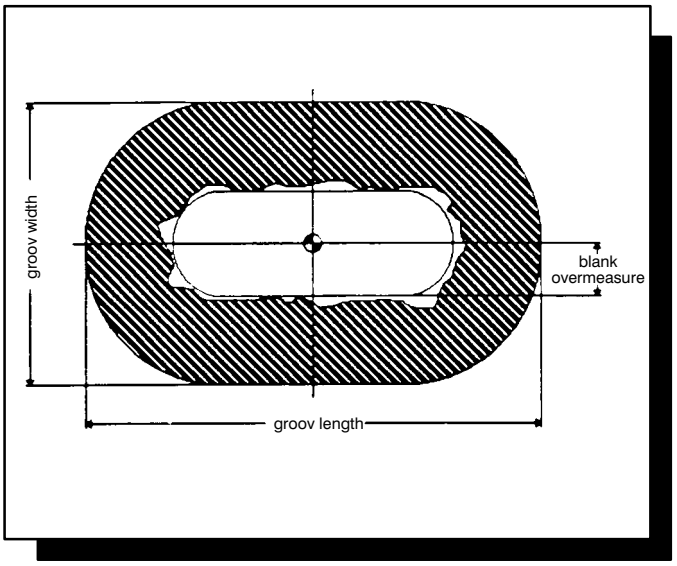
This parameter is active during both roughing and finishing.

The parameter value is entered as an incremental dimension (in mm) and calculated starting at the centre point. Only positive values and zero are allowed.

Entering zero means machining starts at the centre point.

Maximum value
1/2 pocket width minus (finishing overmeasure/wall plus cutter radius).

3.14.3 Groove



This parameter is active during both roughing and finishing.

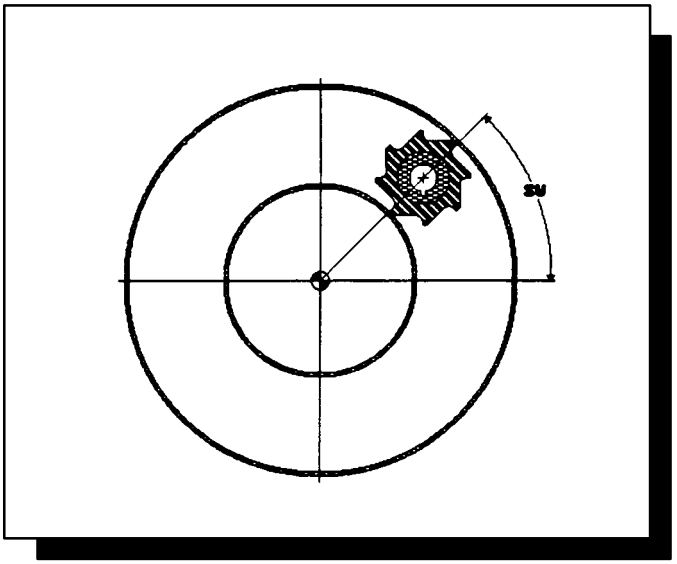
The parameter value is entered as an incremental dimension (in mm) and calculated starting at the centre point. Only positive values and zero are allowed.

Entering zero means machining starts at the centre point.

Maximum value
1/2 groove width minus (finishing overmeasure/wall plus cutter radius).

3.15 Starting Angle

3.15.1 End Groove



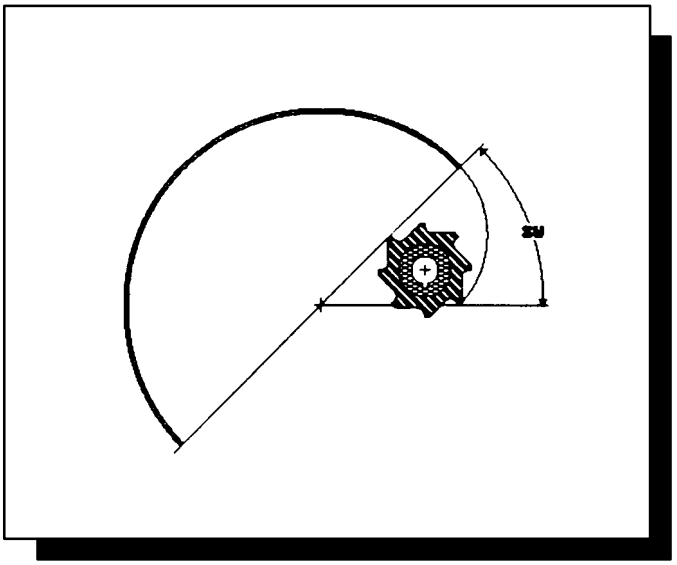
This parameter determines the position of the starting point for machining.

The starting point is calculated from the inside diameter, half the groove width (computed value) and the starting angle.

The angle is entered as an absolute value in degrees. (Mathematically positive direction of rotation)

Values can be positive, negative or zero.

3.15.2 Inside Circle Segment



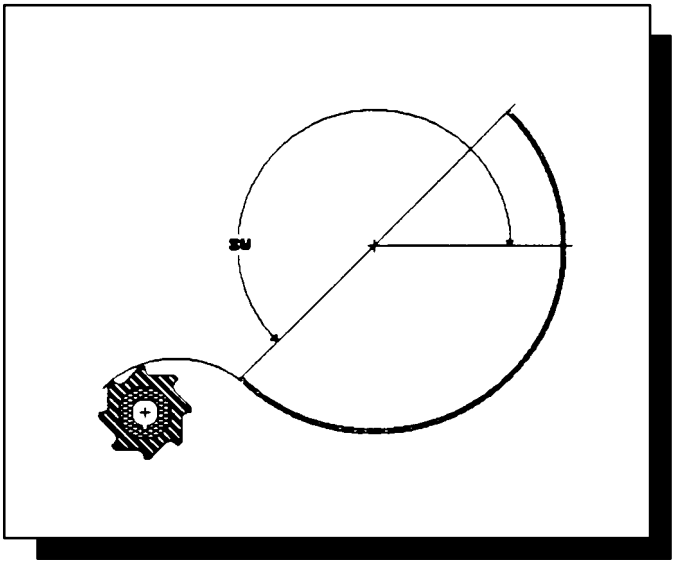
This parameter determines the contour start and the position of the starting point for machining.

The starting point is calculated from the diameter, the approach radius, the cutter radius and the starting angle.

The angle is entered as an absolute value in degrees. (Mathematically positive direction of rotation)

Values can be positive, negative or zero.

3.15.3 Outside Circle Segment



This parameter determines the contour start and the position of the starting point for machining.

The starting point is calculated from the diameter, the approach radius, the cutter radius and the starting angle.

The angle is entered as an absolute value in degrees. (Mathematically positive direction of rotation)

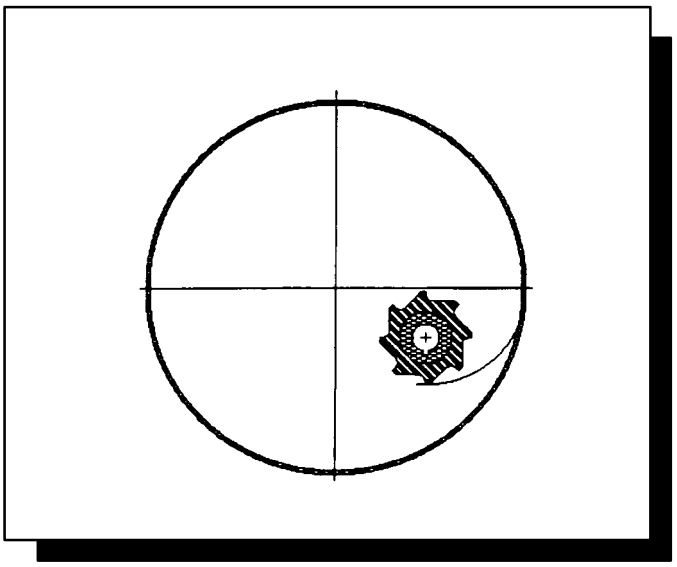
Values can be positive, negative or zero.

3.15.4 Borehole

A starting angle of zero degrees is automatically specified in this cycle.

The starting point is calculated from the diameter, the starting angle of zero degrees and the approach radius.

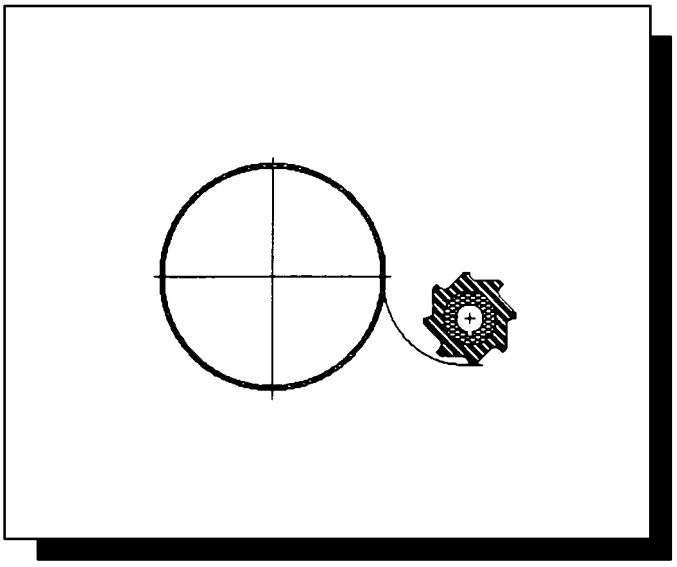
The start is always made from the centre point.



3.15.5 Tenon

In this cycle, a starting angle of zero degrees is automatically specified.

The starting point is calculated from the centre point, the diameter, the starting angle of zero degrees and the approach radius.

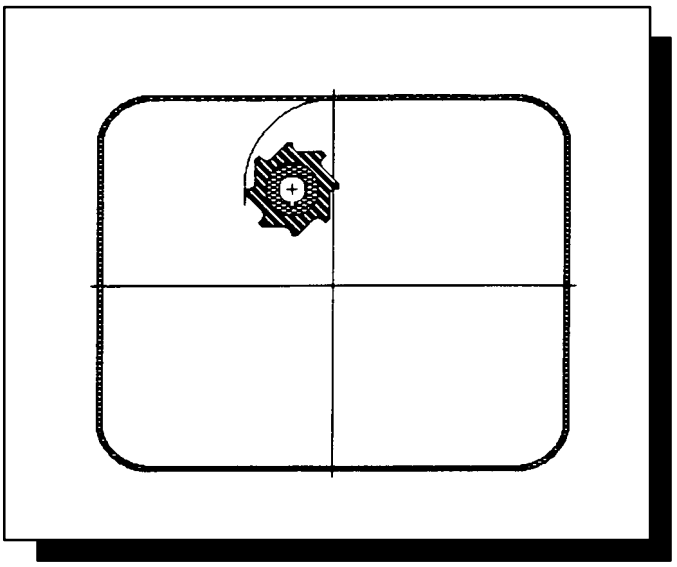


3.15.6 Inside Frame

A starting angle of zero degrees is automatically specified in this cycle.

The starting point is calculated from the centre point, the frame width, the starting angle of zero degrees and the approach radius.

The start is always made from the centre point.

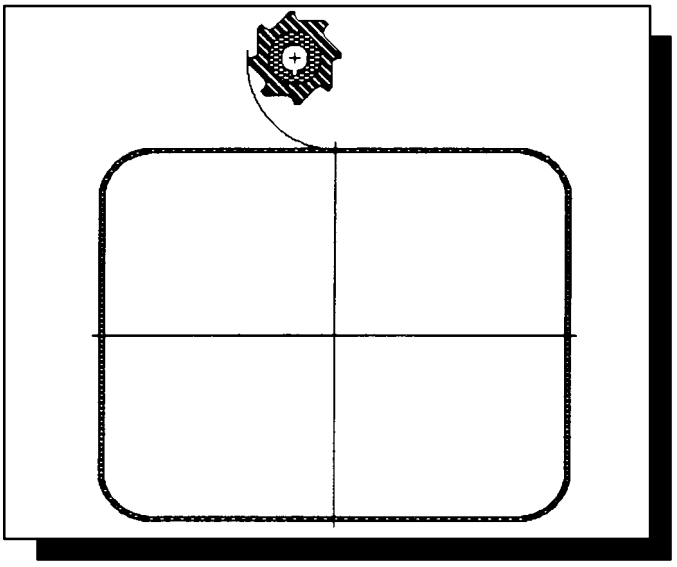


3.15.7 Tenon (Frame)

A starting angle of zero degrees is automatically specified in this cycle.

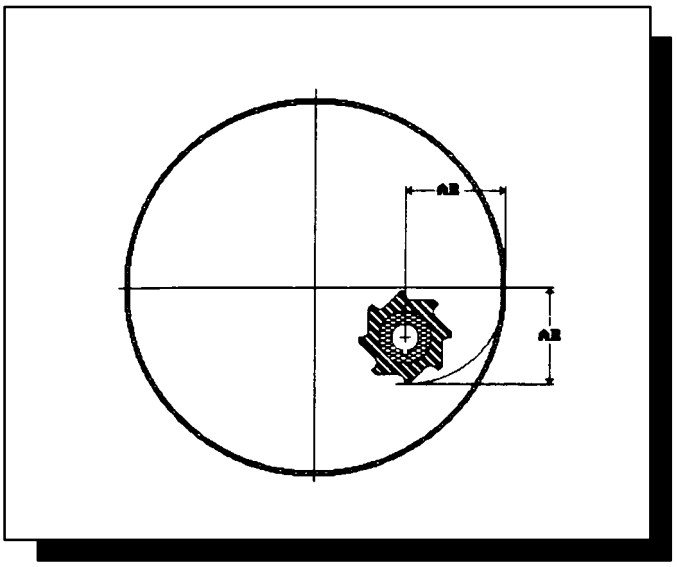
The starting point is calculated from the centre point, the frame width, the starting angle of zero degrees and the approach radius.

The start can be made from any position; the contour is always approached at a tangent.



3.16 Approach Radius

3.16.1 Borehole



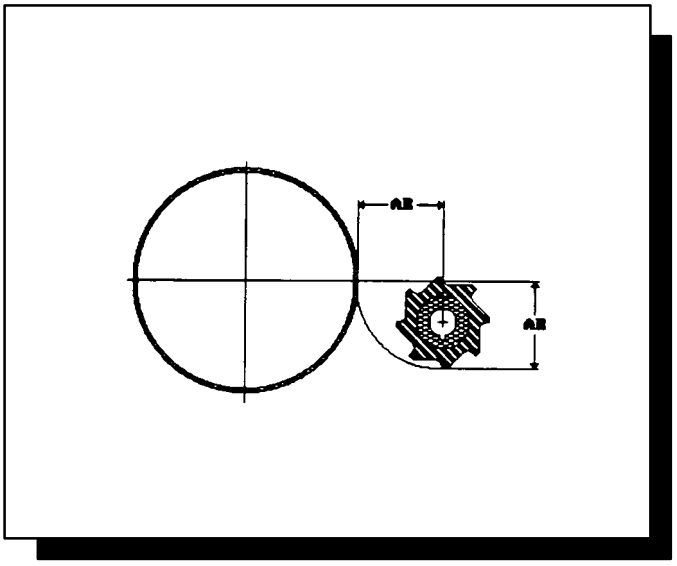
This parameter determines the starting point for machining.

The parameter value is entered as an incremental value (in mm). Only positive values are allowed.

The starting point is calculated from the centre point, the diameter, the starting angle of zero degrees and the approach radius.

The start is always made from the centre point.

3.16.2 Tenon

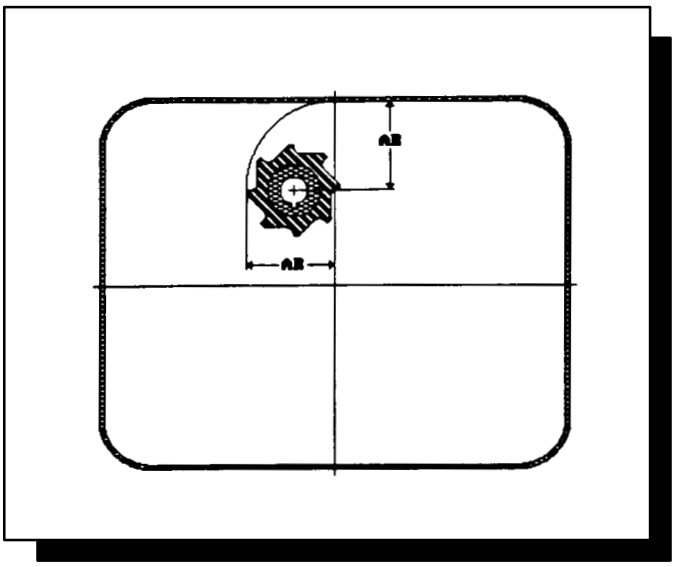


This parameter determines the starting point for machining.

The parameter value is entered as an incremental value (in mm). Only positive values are allowed.

The starting point is calculated from the centre point, the diameter, the starting angle of zero degrees and the approach radius.

3.16.3 Inside Frame



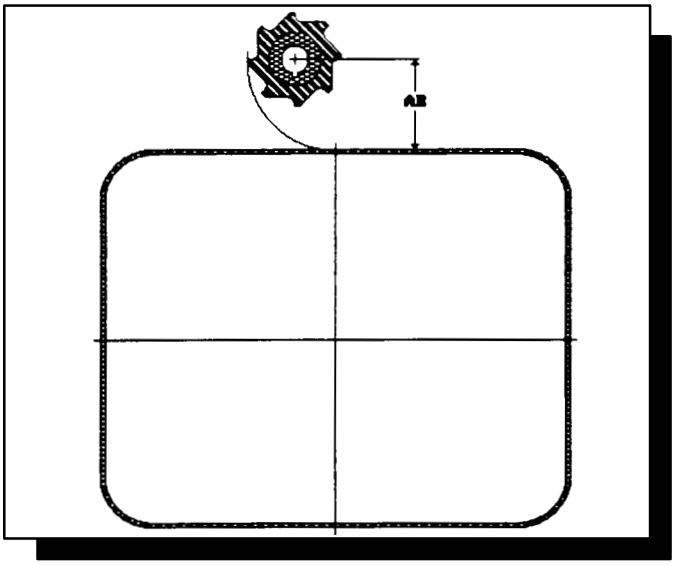
This parameter determines the starting point for machining.

The parameter value is entered as an incremental value (in mm). Only positive values are allowed.

The starting point is calculated from the centre point, the frame width, the starting angle of zero degrees and the approach radius.

The start is always made from the centre point.

3.16.4 Outside Frame



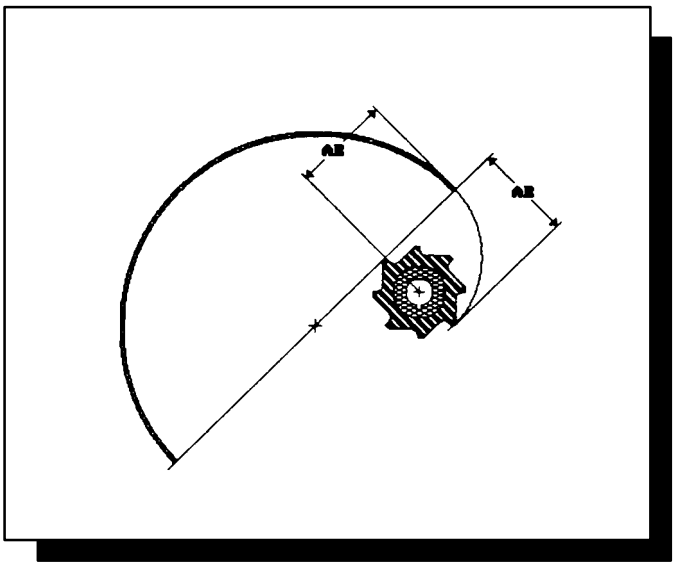
This parameter determines the starting point for machining.

The parameter value is entered as an incremental value (in mm). Only positive values are allowed.

The starting point is calculated from the centre point, the frame width, the starting angle of zero degrees and the approach radius.

The start can be made from any position; the contour is always approached at a tangent.

3.16.5 Inside Circle Segment

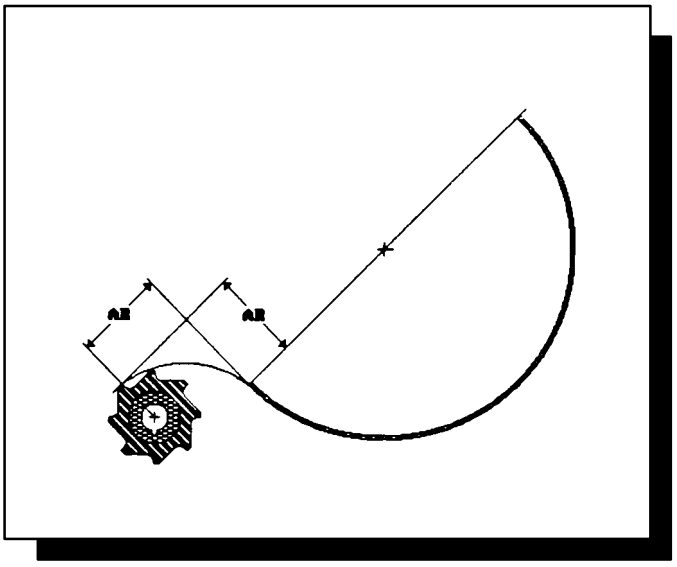


This parameter determines the starting point for machining.

The parameter value is entered as an incremental value (in mm). Only positive values are allowed.

The starting point is calculated from the diameter, the approach radius, the cutter radius and the starting angle.

3.16.6 Outside Circle Segment



This parameter determines the starting point for machining.

The parameter value is entered as an incremental value (in mm). Only positive values are allowed.

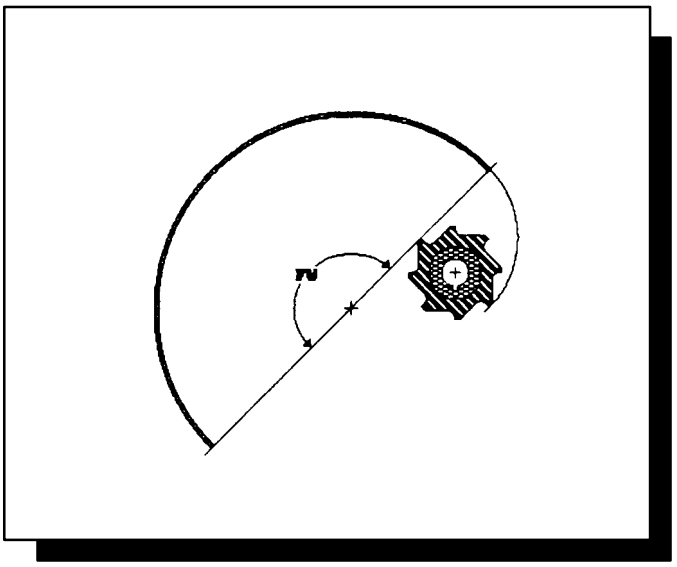
The starting point is calculated from the diameter, the approach radius, the cutter radius and the starting angle.

3.17 Milling Angle

3.17.1 Inside Circle Segment

This parameter determines the cutting length at the inside circle.

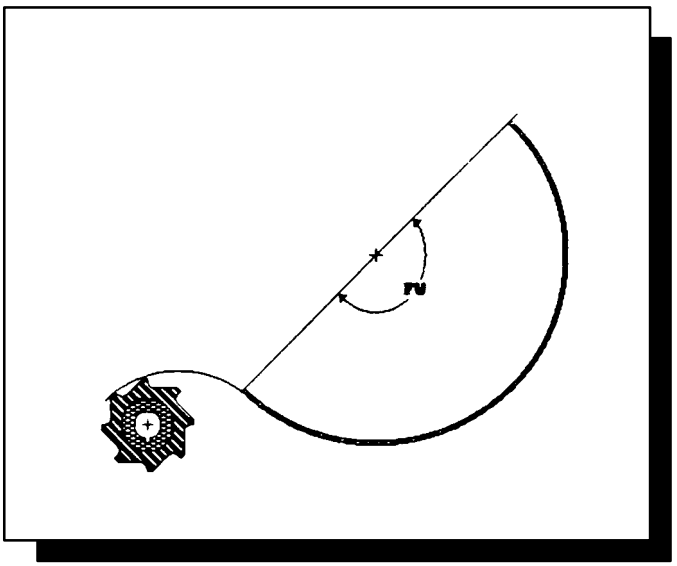
The parameter value is entered as an incremental dimension (in degrees). Only positive values are allowed.



3.17.2 Outside Circle Segment

This parameter determines the cutting length at the outside circle.

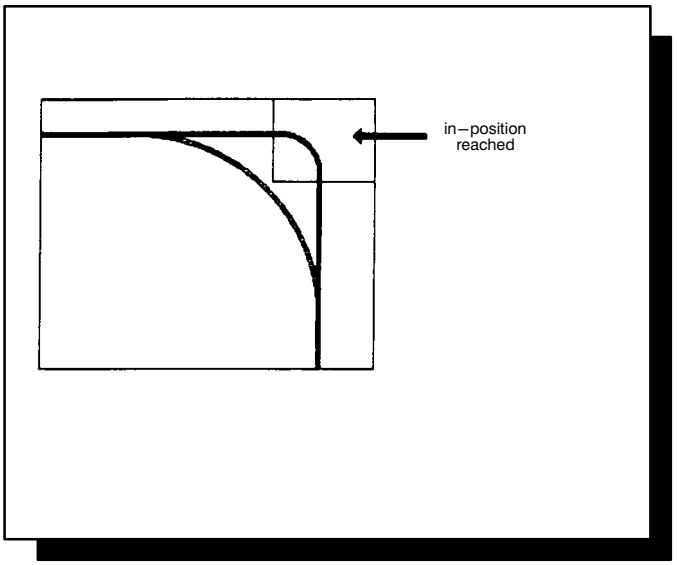
The parameter value is entered as an incremental dimension (in degrees). Only positive values are allowed.



3.18 In-position Programming

In-position = 1:

This parameter value causes the axes to be positioned accurately in their end position.
The position area is defined by machine parameters.



In-position = 0

This parameter value extends the positioning area (changing the block “on the fly”).

This parameter is only active in the cycles:

- inside frame and
  outside frame

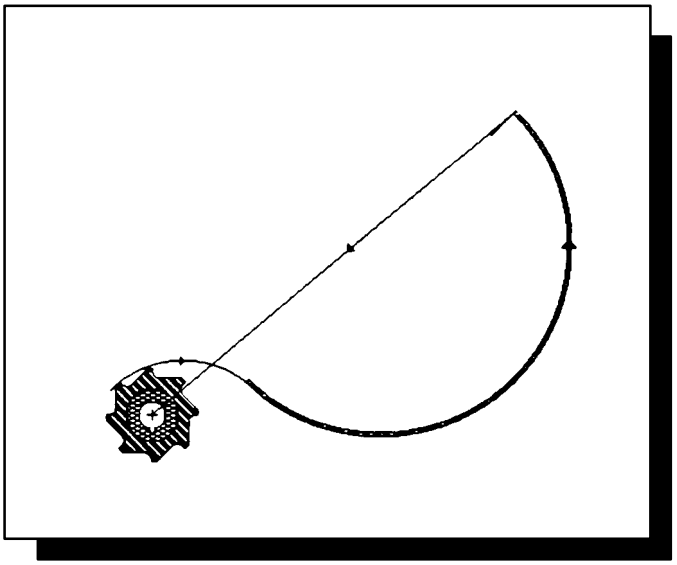
3.19 Return to Starting Point

Return to Starting Point = 0
(In G0 Mode):

This parameter value causes the cutter, after completing machining, to retract to the approach plane plus safety clearance and then return in rapid traverse to the starting point.

This parameter is only active in the cycles:

- inside circle segment and
  outside circle segment.

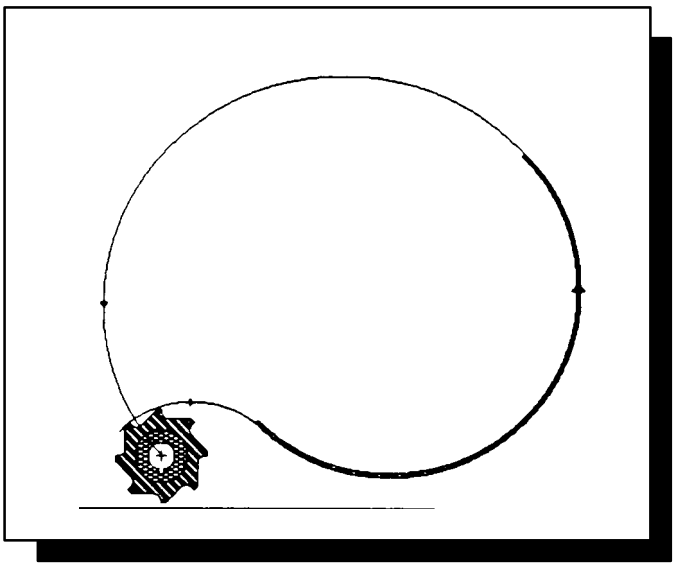


Return to Starting Point = 1
(In G2/G3):

This parameter value causes the cutter, after completing machining, to retract to the approach plane plus safety clearance and then return in a circular movement to the starting point.

This parameter is only active in the cycles:

- inside circle segment and
  outside circle segment.



4 Program Preconditions

4.1 Tool

The tool must be selected before the cycle is invoked.

4.2 Cutter Radius

The active cutter radius must be defined either by programming the appropriate D geometric compensation group or by activating the compensation values (G145–G845) provided by the PC600.

4.3 Cutter Compensation

No cutter compensation G41 or G42 must be active. The cycles calculate the cutter path themselves.

4.4 Spindle Speed

The spindle speed must be programmed before the cycle is invoked.

4.5 Spindle Direction of Rotation

The spindle direction of rotation must be programmed before the cycle is invoked.

4.6 Scale Factor

No scale factor must be programmed.

4.7 Milling Plane

The plane must be selected using G17, G18, G19 or G20.

4.8 Infeed Axis and Direction

The infeed axis and direction with the length compensation offset must be switched over with G78. By default, infeed is in negative direction on the Z-axis (corresponds to machine parameter P6507).

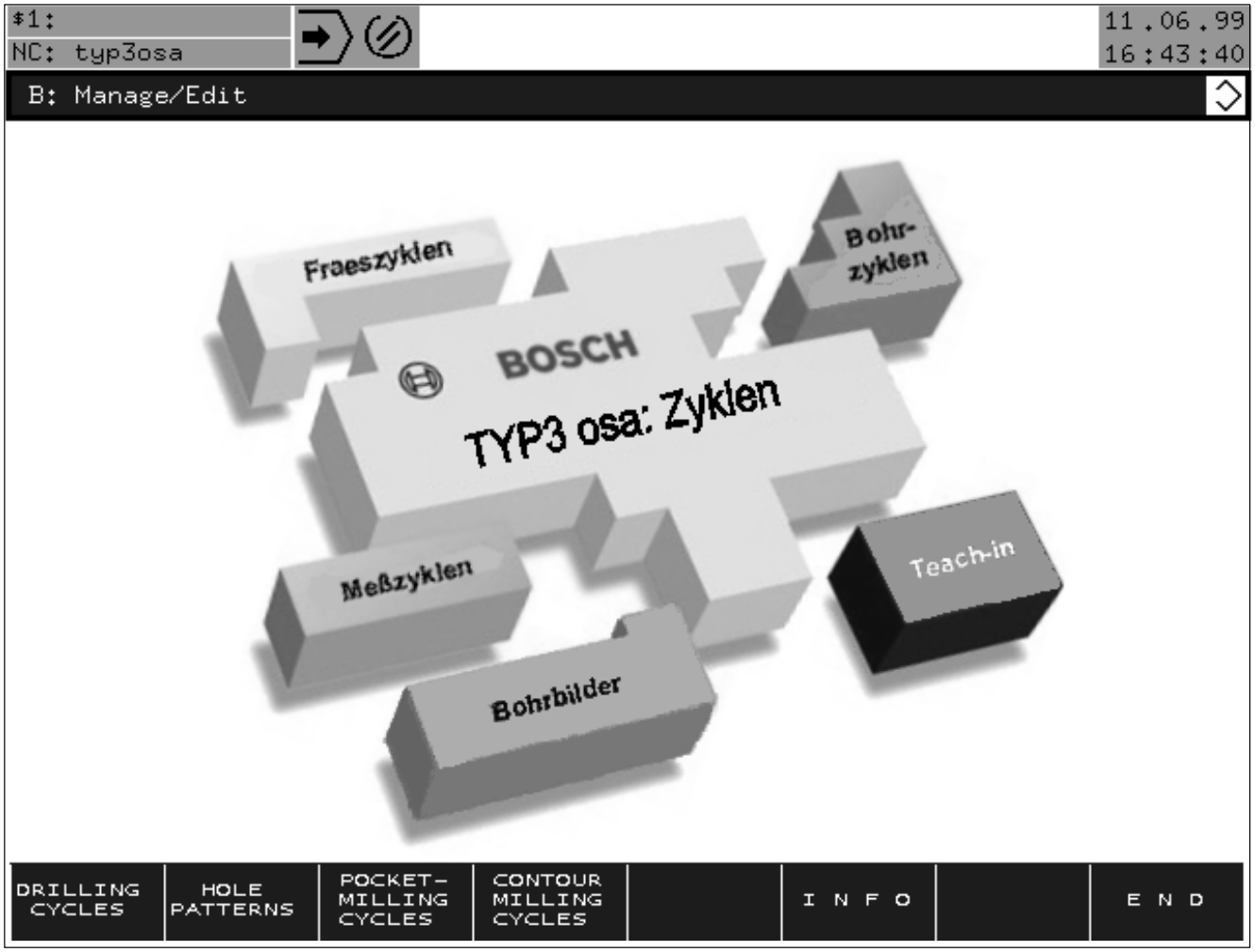
Notes:

5 Description of cycles

5.1 Menu Overview

5.1.1 Main Menu

The following figure shows the main menu. You can press softkeys to select.



Drilling Cycles	• means: CALL ———>	menu of **drilling cycles** (see Typ3 DIN-Programming documentation)
Hole Patterns	• means: CALL ———>	menu of **hole patterns** (see Typ3 DIN-Programming documentation)
Pocket–Milling Cycles	• means: CALL ———>	menu of **pocket milling cycles** see kap. 5.2

Contour–Milling Cycles

- means: CALL ———> menu of **contour milling cycles** (see kap. 5.3)

Info

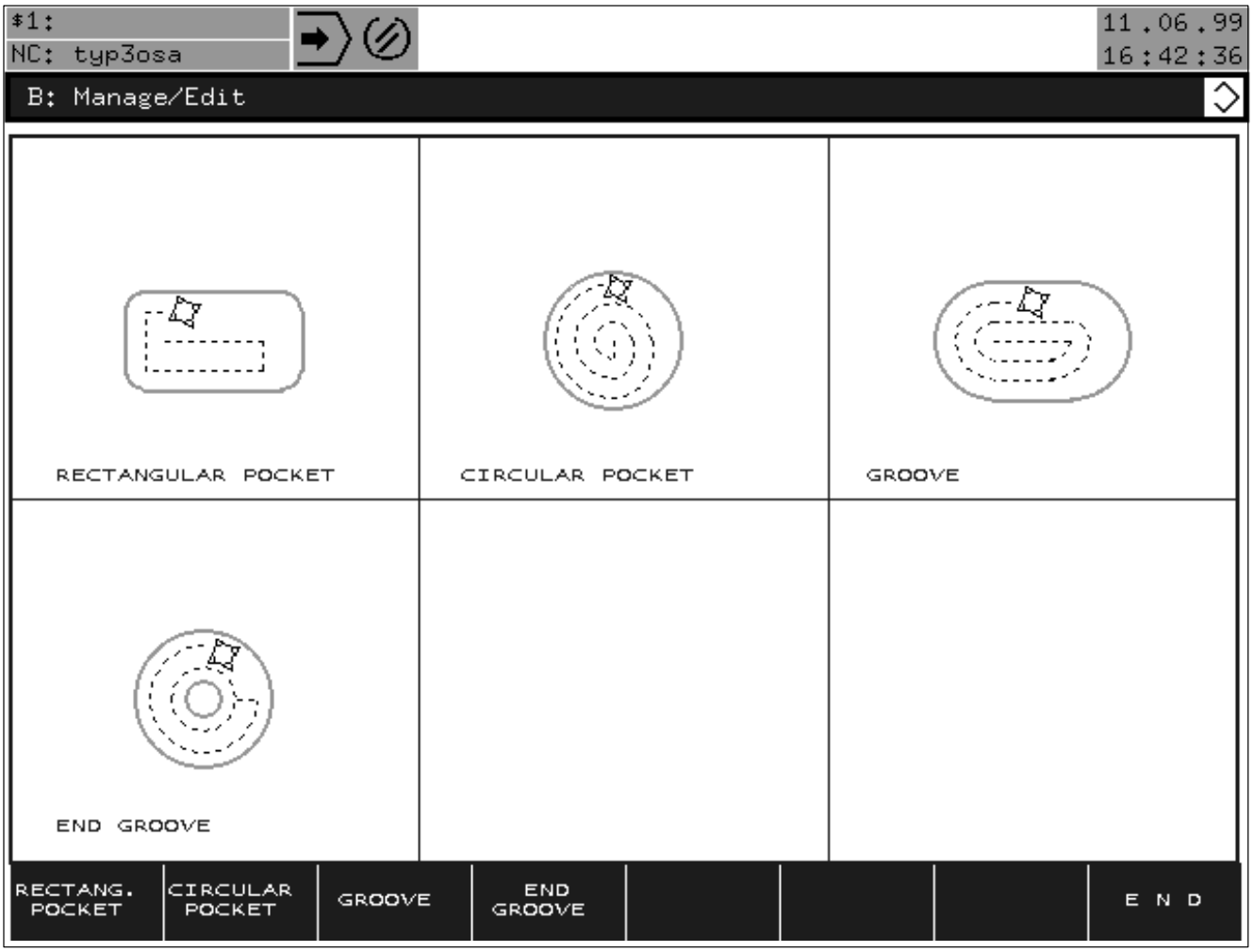
- means: CALL ———> CPL-program information

End

- means: **return** ———> to the next highest dialogue level

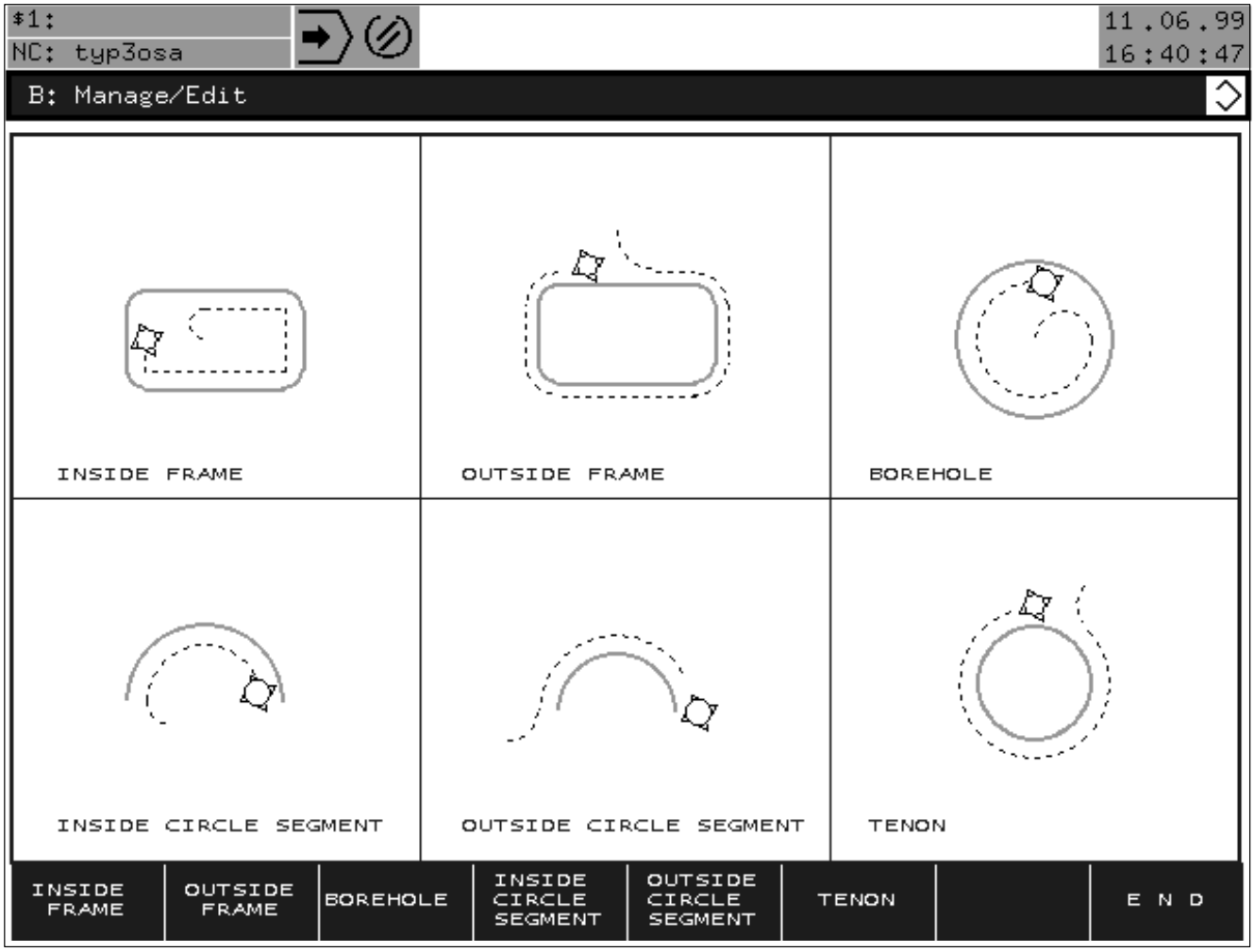
5.1.2 Pocket Milling Cycle Menu

The following figure shows the pocket milling cycle menu. You can select from the cycles shown in the softkey bar.



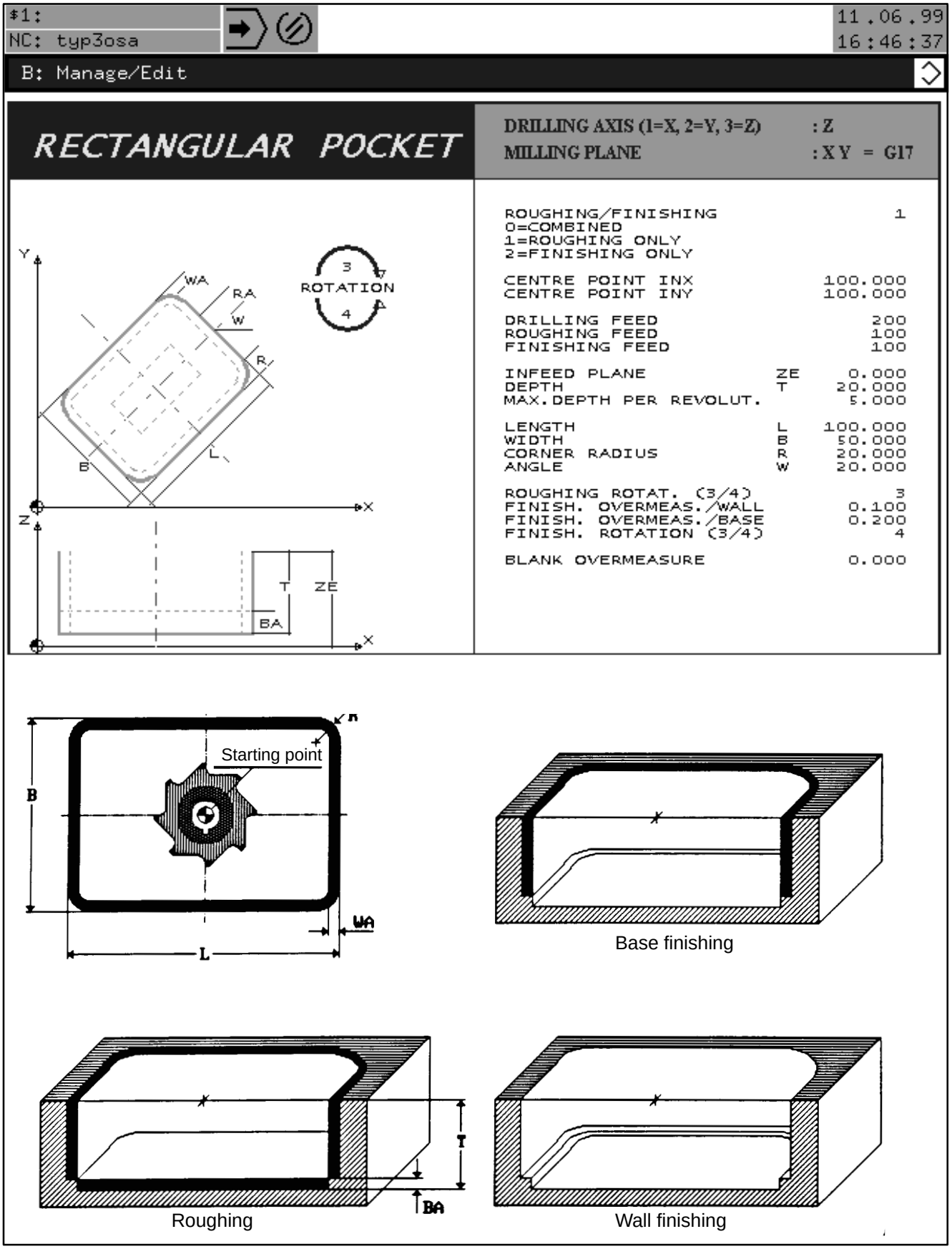
5.1.3 Contour Milling Cycle Menu

The following figure shows the contour milling cycle menu. You can select from the cycles shown in the softkey bar.



5.2 Pocket Milling Cycles

5.2.1 Rectangular Pocket



Programming Example

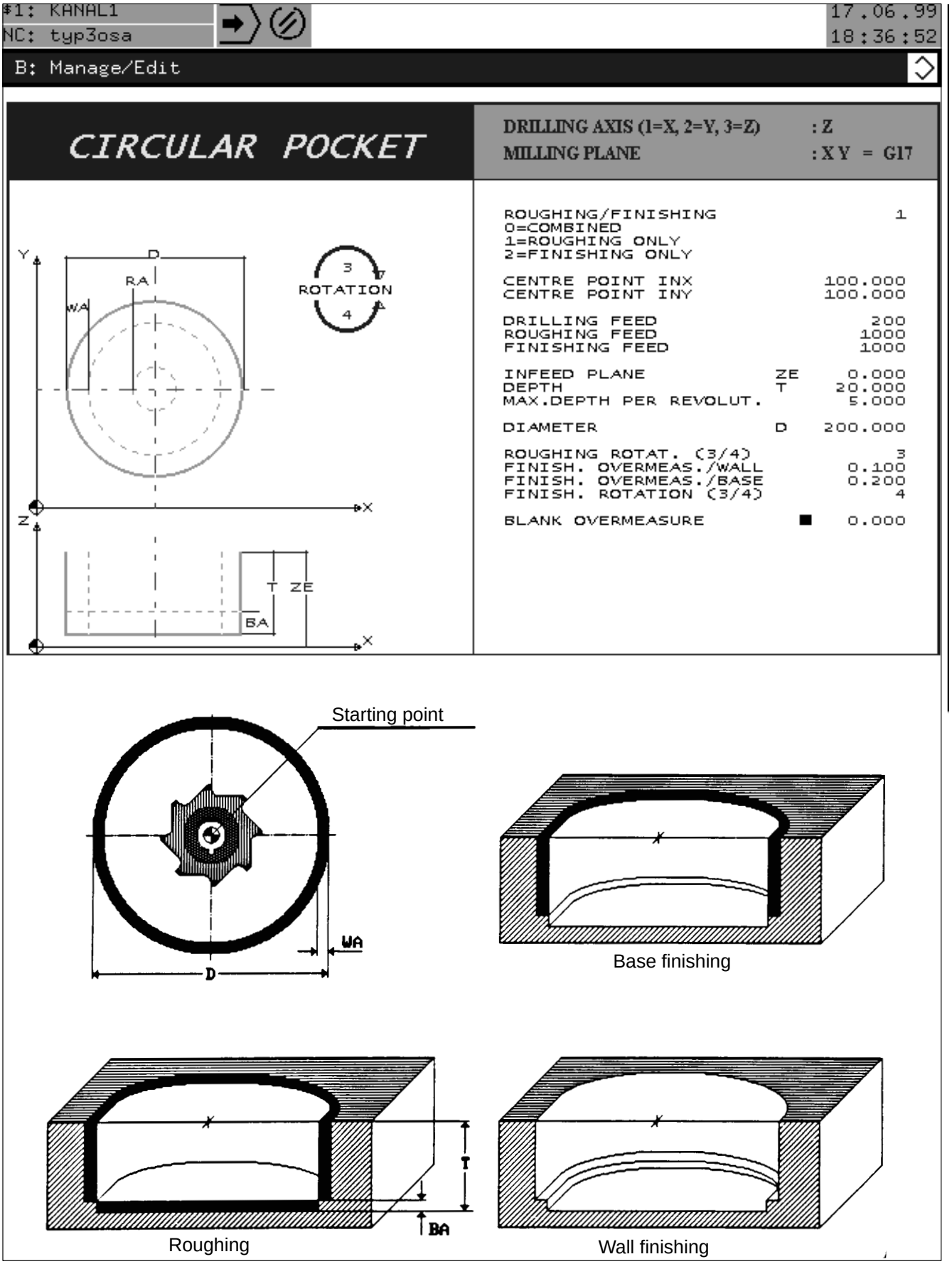
Milling a rectangular pocket with wall finishing.

```
.
.
N101   T1234      Tool number
N102   M6         Tool change
N103   M3 S500    Direction of rotation/speed
N104   GO X0 Y0   Position axes to zero
N105   D1         Call tool number from geometry table
                  (D compensation)
N106   P102       Call rectangular pocket cycle
       [0,        Roughing/finishing combined
       100,       Centre point in X (mm, absolute)
       100,       Centre point in Y (mm, absolute)
       200,       Drilling feed (mm/min)
       1000,      Roughing feed (mm/min)
       1000,      Finishing feed (mm/min)
       0,         Infeed plane (mm, absolute)
       20,        Depth (mm, incremental)
       5,         Maximum depth per revolution (mm, incremental)
       100,       Length (mm, incremental)
       50,        Width (mm, incremental)
       20,        Corner radius (mm, incremental)
       20,        Angle (degrees, absolute)
       3,         Roughing rotation
       0.1,       Finishing overmeasure/wall (mm, incremental)
       0,         Finishing overmeasure/base (mm, incremental)
       4,         Finishing rotation
       0]         Blank overmeasure (mm, incremental)
N107   M5
etc.
```

Messages / Plausibility

ROUGHING/FINISHING FALSE
DRILLING FEED <=0
ROUGHING FEED <=0
FINISHING FEED <=0
DEPTH <=0
MAX. INFEED PER REVOLUTION <=0
MAX. INFEED PER REVOLUTION > DEPTH
LENGTH <=0
WIDTH <=0
CORNER RADIUS <=0
FALSE DIRECTION OF ROTATION FINISHING/ROUGHING
FINISHING OVERMEASURE/WALL <0
FINISHING OVERMEASURE/BASE <0
BLANK OVERMEASURE <0
ACTIVE CUTTER RADIUS <=0
CUTTER RAD. >= ((HALF POCKET WIDTH – WALL OVERMEASURE) * 0.9)
CUTTER RADIUS > CORNER RADIUS – WALL OVERMEASURE
LENGTH < WIDTH
BLANK OVERMEASURE TOO LARGE

5.2.2 Circular Pocket



Programming Example

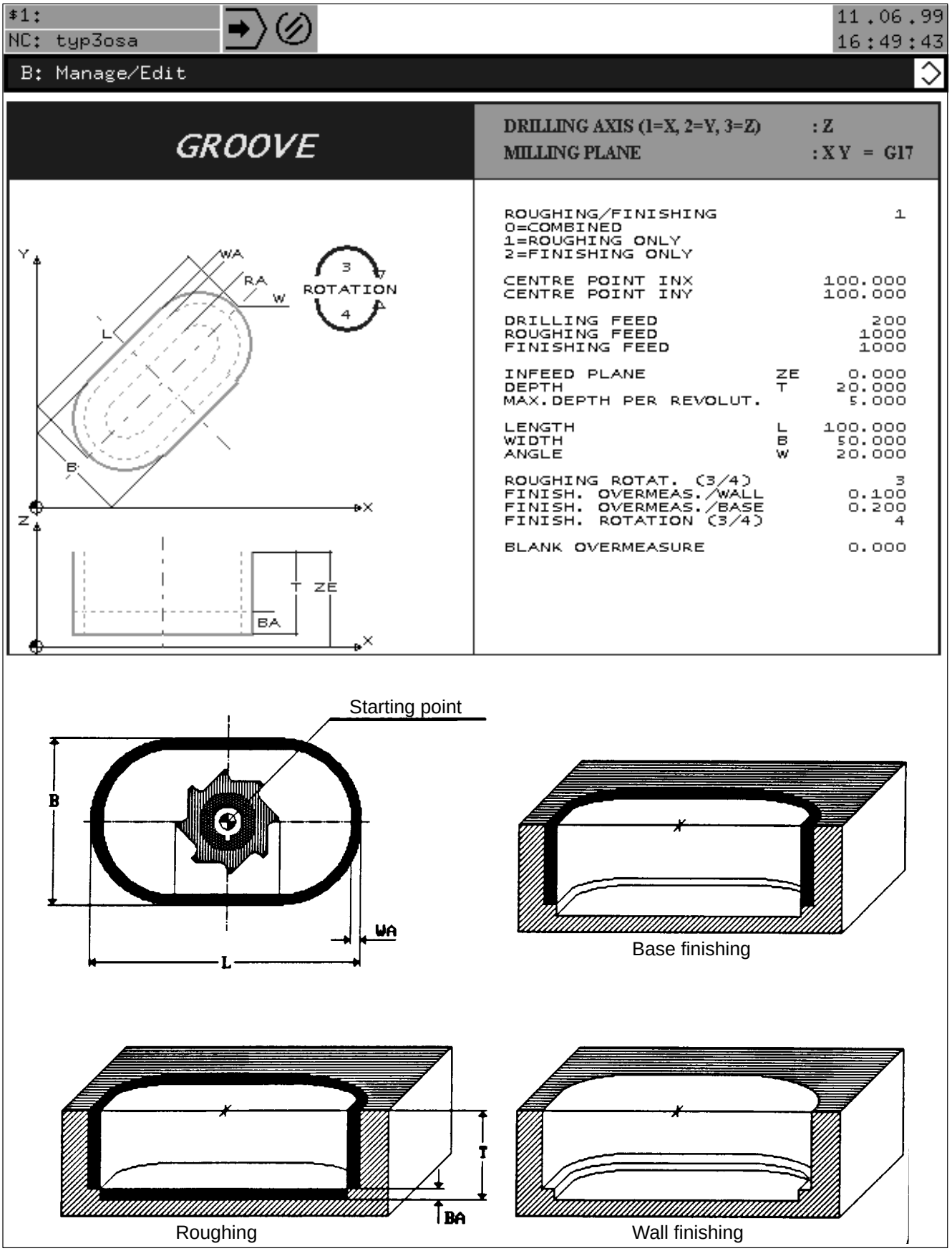
Milling a circular pocket.

```
.
.
N101   T1234        Tool number
N102   M6           Tool change
N103   M3 S500      Direction of rotation/speed
N104   GO X0 Y0     Position axes to zero
N105   D1           Call tool number from geometry table
                    (D compensation)
N106   P9990103     Call circular pocket cycle
       [1,          Roughing
       100,         Centre point in X (mm, absolute)
       100,         Centre point in Y (mm, absolute)
       200,         Drilling feed (mm/min)
       1000,        Roughing feed (mm/min)
       0,           Finishing feed (mm/min)
       0,           Infeed plane (mm, absolute)
       20,          Depth (mm, incremental)
       5,           Maximum depth per revolution (mm, incremental)
       200,         Diameter (mm, incremental)
       3,           Roughing rotation
       0.1,         Finishing overmeasure/wall (mm, incremental)
       0,           Finishing overmeasure/base (mm, incremental)
       4,           Finishing rotation
       0]           Blank overmeasure (mm, incremental)
N107   M5
etc.
```

Messages / Plausibility

ROUGHING/FINISHING FALSE
DRILLING FEED <=0
ROUGHING FEED <=0
FINISHING FEED <=0
DEPTH <=0
MAX. INFEED PER REVOLUTION <=0
MAX. INFEED PER REVOLUTION > DEPTH
DIAMETER <=0
FALSE DIRECTION OF ROTATION FINISHING/ROUGHING
FINISHING OVERMEASURE/WALL <0
FINISHING OVERMEASURE/BASE <0
BLANK OVERMEASURE <0
ACTIVE CUTTER RADIUS <=0
CUTTER RAD. >= ((HALF DIAMETER – WALL OVERMEASURE) * 0.9)
CUTTER RADIUS > CORNER RADIUS – WALL OVERMEASURE
BLANK OVERMEASURE TOO LARGE

5.2.3 Groove


$1:
NC: typ3osa
11.06.99
16:49:43
B: Manage/Edit
GROOVE
DRILLING AXIS (1=X, 2=Y, 3=Z) : Z
MILLING PLANE : X Y = G17
ROUGHING/FINISHING 1
0=COMBINED
1=ROUGHING ONLY
2=FINISHING ONLY
CENTRE POINT INX 100.000
CENTRE POINT INY 100.000
DRILLING FEED 200
ROUGHING FEED 1000
FINISHING FEED 1000
INFEED PLANE ZE 0.000
DEPTH T 20.000
MAX.DEPTH PER REVOLUT. 5.000
LENGTH L 100.000
WIDTH B 50.000
ANGLE W 20.000
ROUGHING ROTAT. (3/4) 3
FINISH. OVERMEAS./WALL 0.100
FINISH. OVERMEAS./BASE 0.200
FINISH. ROTATION (3/4) 4
BLANK OVERMEASURE 0.000
WA
RA
W
3
ROTATION
4
L
B
Y
X
Z
T
ZE
BA
Starting point
B
WA
L
Base finishing
T
BA
Roughing
Wall finishing

Programming Example

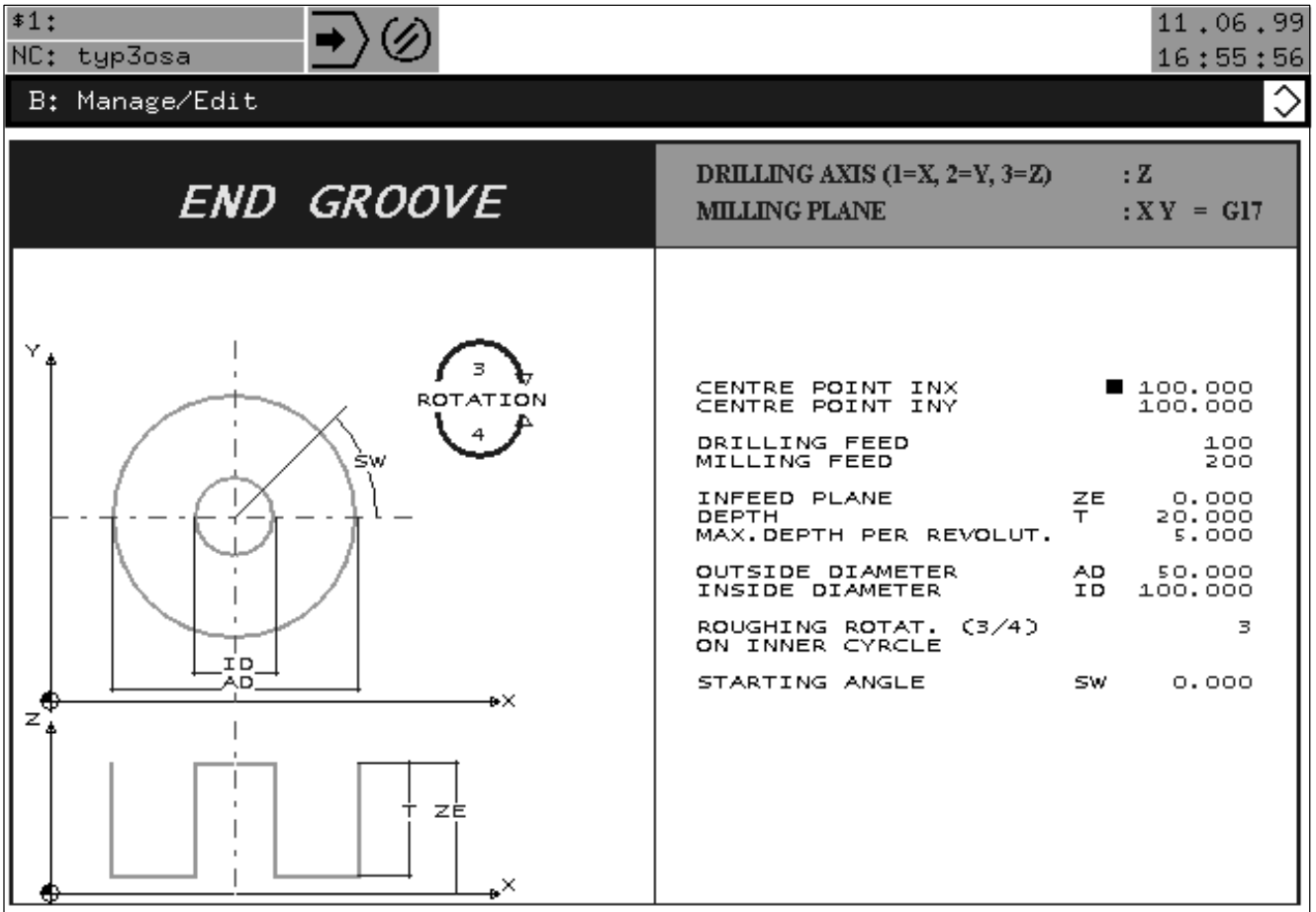
Milling a groove.

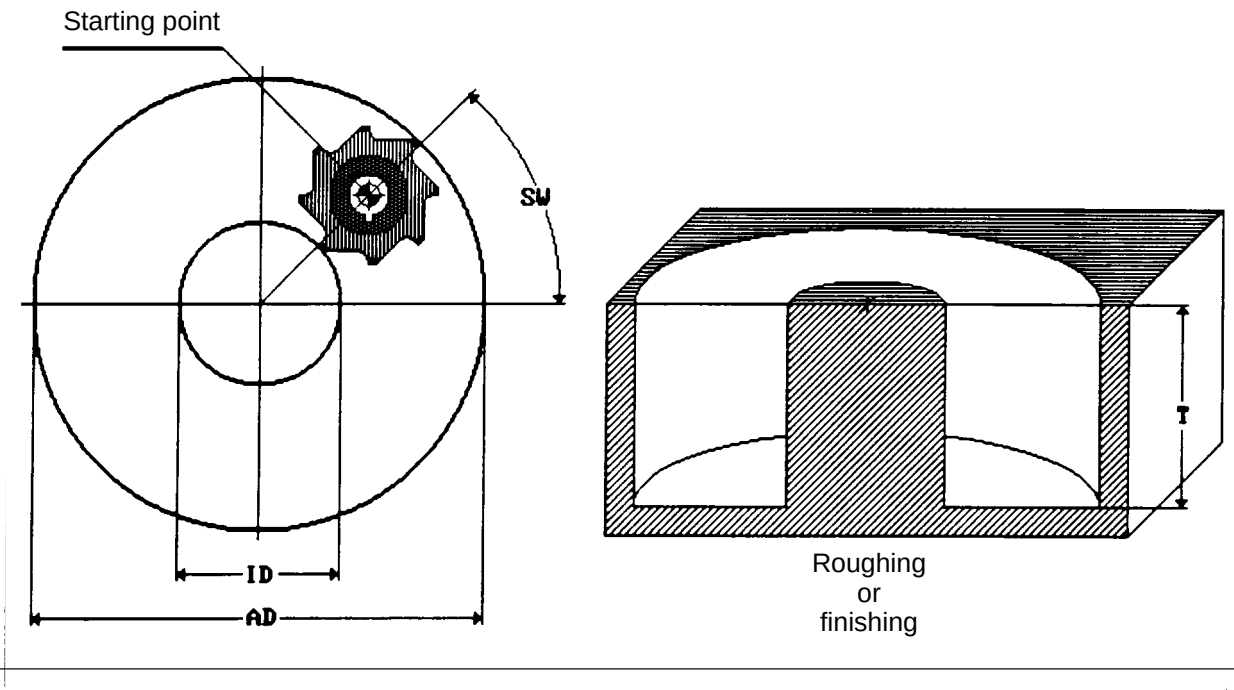
```
.
.
N101   T1234        Tool number
N102   M6           Tool change
N103   M3 S500      Direction of rotation/speed
N104   GO X0 Y0     Position axes to zero
N105   D1           Call tool number from geometry table
                    (D compensation)
N106   P9990104     Call groove cycle
       [2,          Finishing
       100,         Centre point in X (mm, absolute)
       100,         Centre point in Y (mm, absolute)
       200,         Drilling feed (mm/min)
       1000,        Roughing feed (mm/min)
       500,         Finishing feed (mm/min)
       0,           Infeed plane (mm, absolute)
       20,          Depth (mm, incremental)
       5,           Maximum depth per revolution (mm, incremental)
       100,         Length (mm, incremental)
       50,          Width (mm, incremental)
       20,          Angle (degrees, absolute)
       3,           Roughing rotation
       0.1,         Finishing overmeasure/wall (mm, incremental)
       0.1,         Finishing overmeasure/base (mm, incremental)
       4,           Finishing rotation
       0]           Blank overmeasure (mm, incremental)
N107   M5
etc.
```

Messages / Plausibility

ROUGHING/FINISHING FALSE
DRILLING FEED <=0
ROUGHING FEED <=0
FINISHING FEED <=0
DEPTH <=0
MAX. INFEED PER REVOLUTION <=0
MAX. INFEED PER REVOLUTION > DEPTH
LENGTH <=0
WIDTH <=0
FALSE DIRECTION OF ROTATION FINISHING/ROUGHING
FINISHING OVERMEASURE/WALL <0
FINISHING OVERMEASURE/BASE <0
BLANK OVERMEASURE <0
ACTIVE CUTTER RADIUS <=0
CUTTER RAD. >= ((HALF POCKET WIDTH – WALL OVERMEASURE) * 0.9)
BLANK OVERMEASURE TOO LARGE

5.2.4 End Groove





Programming Example

Milling an end groove.

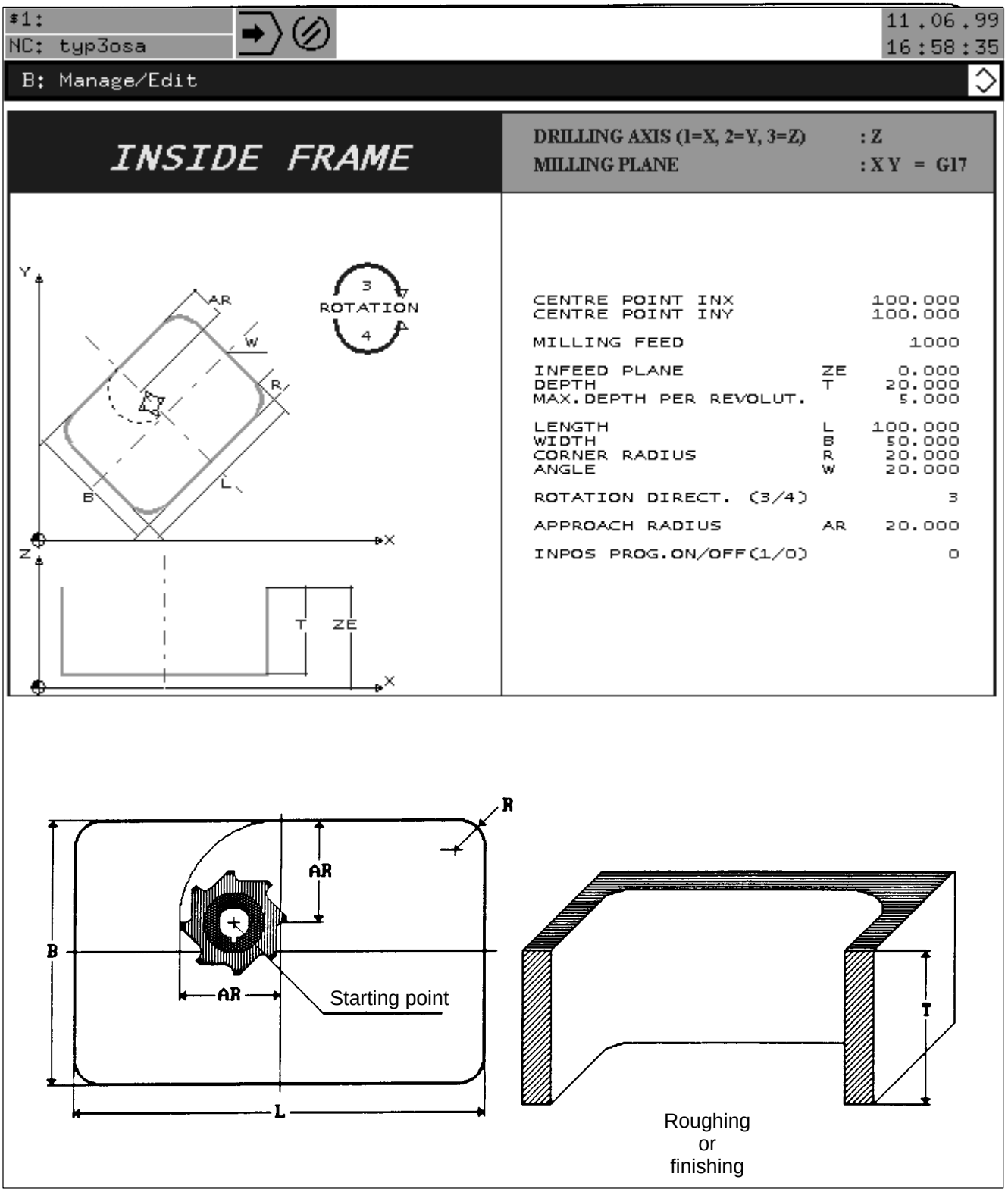
```
.
.
N101   T1234          Tool number
N102   M6             Tool change
N103   M3 S500        Direction of rotation/speed
N104   GO X0 Y0       Position axes to zero
N105   D1             Call tool number from geometry table
                      (D compensation)
N106   P9990106       Call end groove cycle
       [100,          Centre point in X (mm, absolute)
       100,           Centre point in Y (mm, absolute)
       300,           Drilling feed (mm/min)
       1000,          Milling feed (mm/min)
       0,             Infeed plane (mm, absolute)
       10,            Depth (mm, incremental)
       10,            Maximum depth per revolution (mm, incremental)
       150,           Outer diameter (mm, incremental)
       100,           Inner diameter (mm, incremental)
       3,             Direction of rotation
       0]             Starting angle (degrees, absolute)
N107   M5
etc.
```

Messages / Plausibility

DRILLING FEED <=0
MILLING FEED <=0
DEPTH <=0
MAX. INFEED PER REVOLUTION <=0
MAX. INFEED PER REVOLUTION > DEPTH
CUTTER > EFFECTIVE GROOVE WIDTH
INNER DIAMETER >= OUTER DIAMETER
FALSE DIRECTION OF ROTATION
ACTIVE CUTTER RADIUS <=0
INSIDE DIAMETER <=0
OUTSIDE DIAMETER <=0

5.3 Contour Milling Cycles

5.3.1 Inside Frame



Programming Example

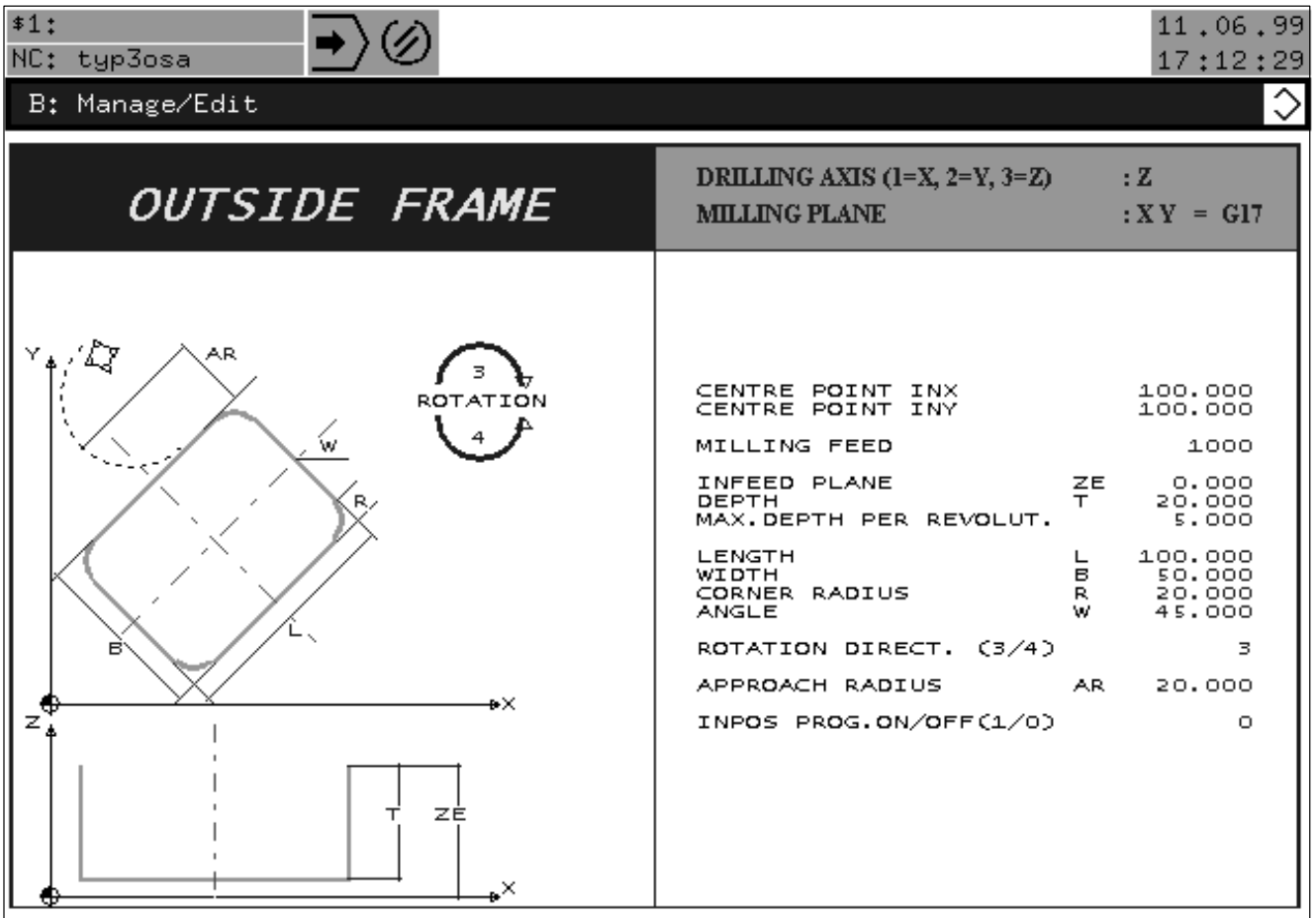
Milling an inside frame.

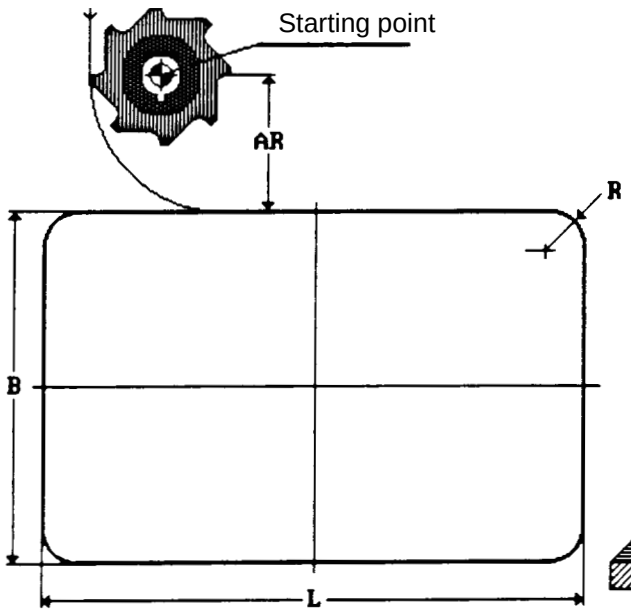
```
.
.
N101    T1234       Tool number
N102    M6          Tool change
N103    M3 S500     Direction of rotation/speed
N104    GO X0 Y0    Position axes to zero
N105    D1          Call tool number from geometry table
                    (D compensation)
N106    P9990110    Call inside frame cycle
        100,        Centre point in X (mm, absolute)
        100,        Centre point in Y (mm, absolute)
        500,        Milling feed (mm/min)
        0,          Infeed plane (mm, absolute)
        20,         Depth (mm, incremental)
        5,          Maximum depth per revolution (mm, incremental)
        200,        Length (mm, incremental)
        90,         Width (mm, incremental
        20,         Corner radius (mm, incremental)
        20,         Angle (degrees, absolute)
        3,          Direction of rotation
        25,         Approach radius (mm, incremental)
        1]          In-position programming on
N107    M5
etc.
```

Messages / Plausibility

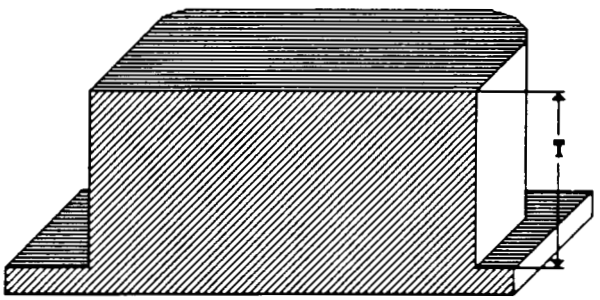
MILLING FEED <=0
DEPTH <=0
MAX. INFEED PER REVOLUTION <=0
MAX. INFEED PER REVOLUTION > DEPTH
LENGTH <=0
WIDTH <=0
CORNER RADIUS <=0
FALSE DIRECTION OF ROTATION
ACTIVE CUTTER RADIUS <=0
CUTTER RAD. >= (HALF POCKET WIDTH * 0.9)
CUTTER RADIUS > CORNER RADIUS
LENGTH < WIDTH
APPROACH RADIUS <= CUTTER RADIUS
APPROACH RADIUS >= HALF WIDTH
IN–POSITION PROGRAMMING FALSE

5.3.2 Outside Frame





The start is made from any position without specifying a starting angle.
The starting point is calculated by the program.
In all positions, a tangential approach to the outer contour **with the approach radius** is ensured.



Roughing
or
finishing

Programming Example

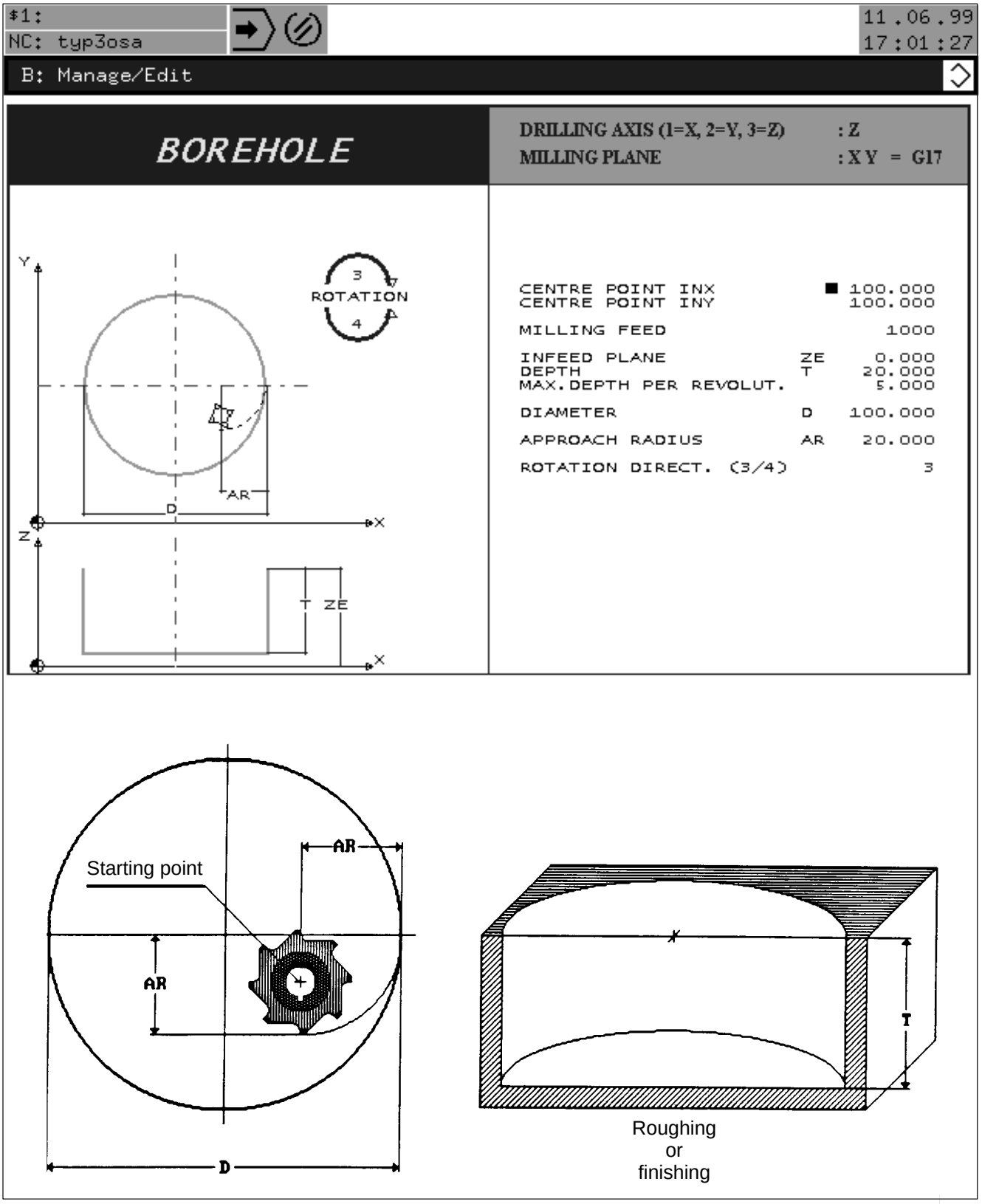
Milling an outside frame.

```
.
.
N101    T1234       Tool number
N102    M6          Tool change
N103    M3 S500     Direction of rotation/speed
N104    GO X0 Y0    Position axes to zero
N105    D1          Call tool number from geometry table
                    (D compensation)
N106    P9990111    Call outside frame cycle
        100,        Centre point in X (mm, absolute)
        100,        Centre point in Y (mm, absolute)
        500,        Milling feed (mm/min)
        0,          Infeed plane (mm, absolute)
        20,         Depth (mm, incremental)
        5,          Maximum depth per revolution (mm, incremental)
        200,        Length (mm, incremental)
        90,         Width (mm, incremental
        20,         Corner radius (mm, incremental)
        20,         Angle (degrees, absolute)
        3,          Direction of rotation
        25,         Approach radius (mm, incremental)
        1]          In-position programming on
N107    M5
etc.
```

Messages / Plausibility

MILLING FEED <=0
DEPTH <=0
MAX. INFEED PER REVOLUTION <=0
MAX. INFEED PER REVOLUTION > DEPTH
LENGTH <=0
WIDTH <=0
CORNER RADIUS <=0
FALSE DIRECTION OF ROTATION
ACTIVE CUTTER RADIUS <=0
LENGTH < WIDTH
APPROACH RADIUS <= CUTTER RADIUS
IN–POSITIONING PROGRAMMING FALSE

5.3.3 Borehole



Programming Example

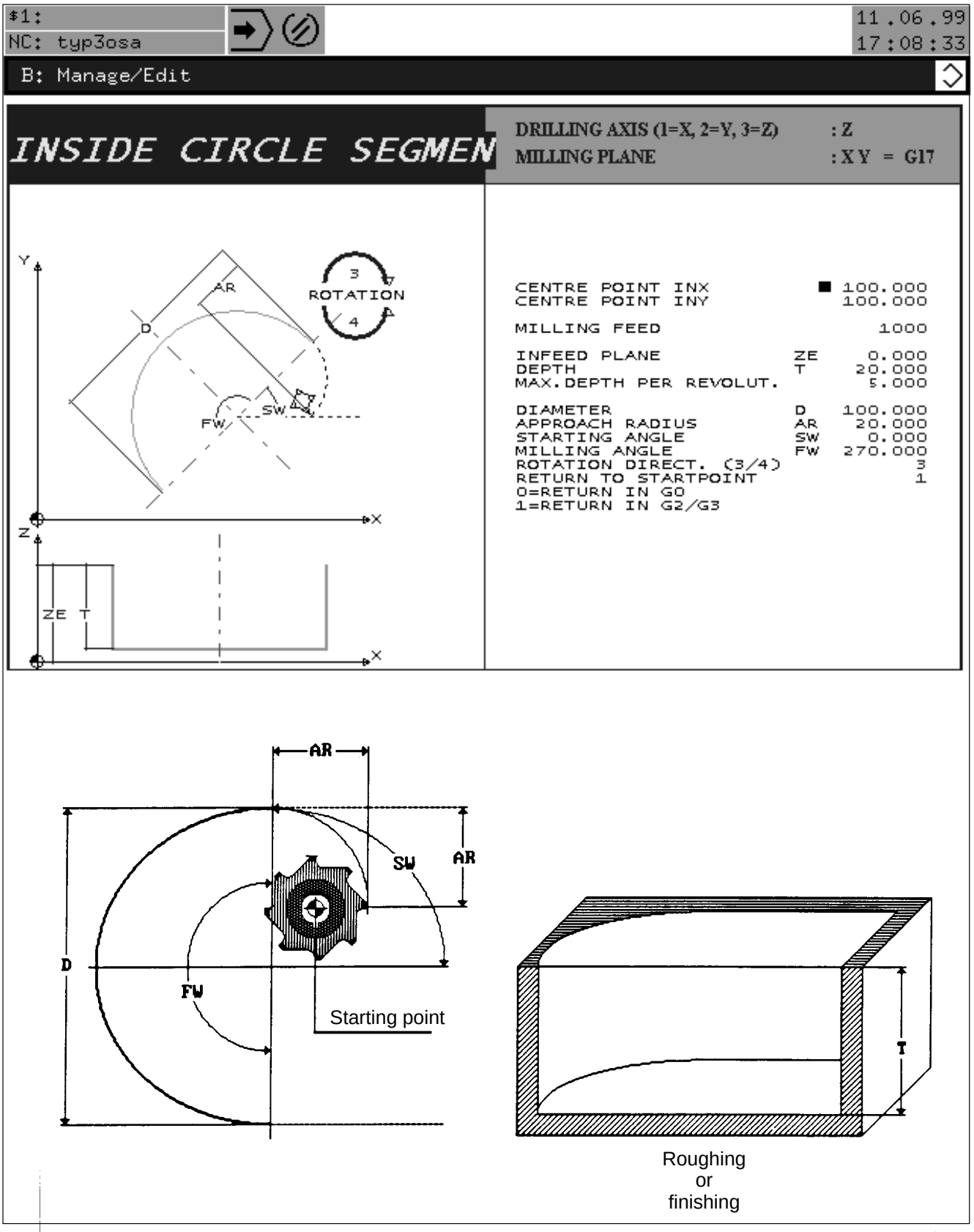
Milling a borehole.

```
.
.
N101    T1234          Tool number
N102    M6             Tool change
N103    M3 S500        Direction of rotation/speed
N104    GO X0 Y0       Position axes to zero
N105    D1             Call tool number from geometry table
                       (D compensation)
N106    P9990112       Call borehole cycle
        100,           Centre point in X (mm, absolute)
        100,           Centre point in Y (mm, absolute)
        1000,          Milling feed (mm/min)
        0,             Infeed plane (mm, absolute)
        10,            Depth (mm, incremental)
        10,            Maximum depth per revolution (mm, incremental)
        100,           Diameter (mm, incremental)
        3,             Direction of rotation
        20]            Approach radius (mm, incremental)
N107    M5
etc.
```

Messages / Plausibility

MILLING FEED <=0
DEPTH <=0
MAX. INFEED PER REVOLUTION <=0
MAX. INFEED PER REVOLUTION > DEPTH
DIAMETER <=0
FALSE DIRECTION OF ROTATION
ACTIVE CUTTER RADIUS <=0
APPROACH RADIUS <= CUTTER RADIUS
APPROACH RADIUS >= HALF DIAMETER

5.3.4 Inside Circle Segment



Programming Example

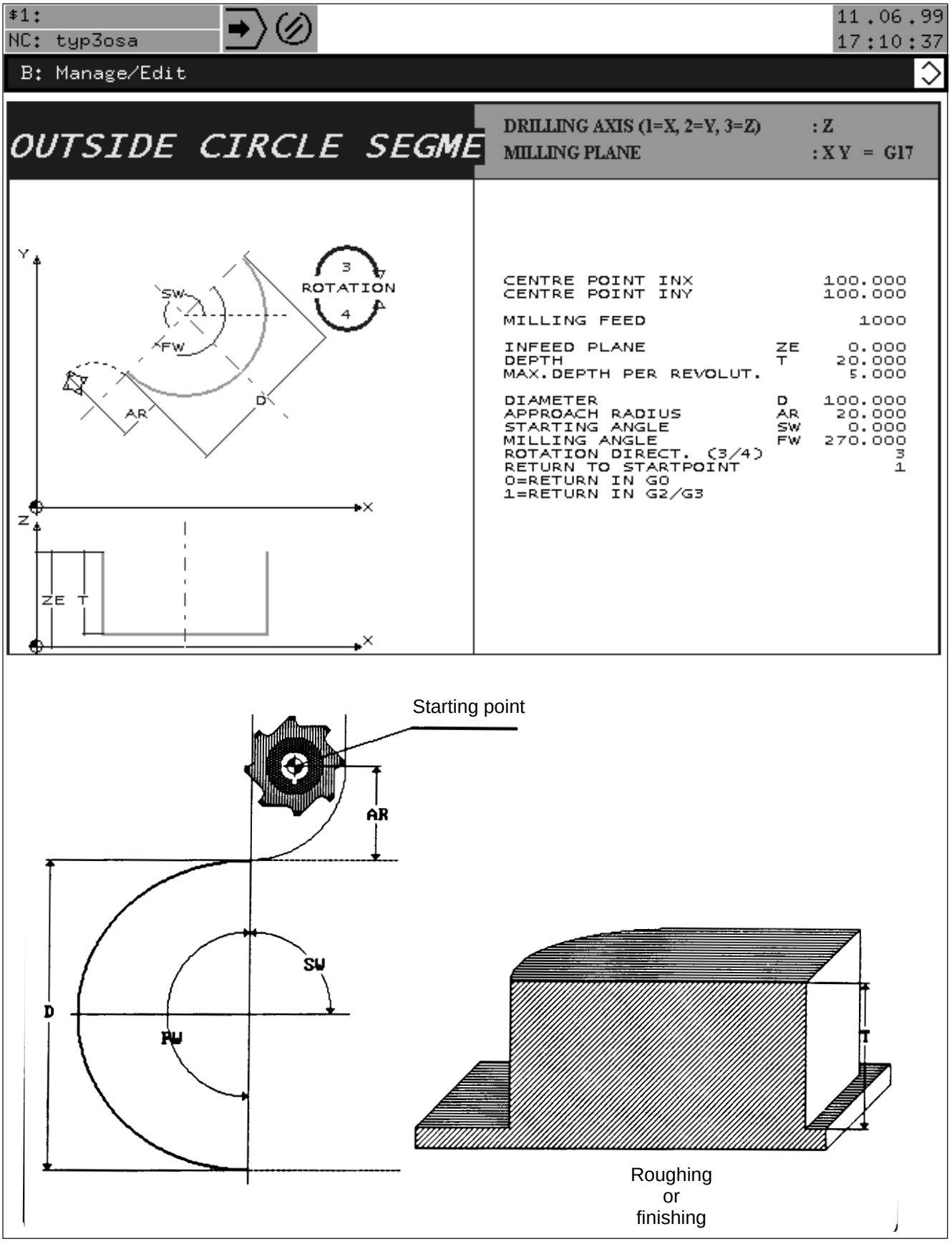
Milling an inside circle segment.

```
.
.
N101   T1234          Tool number
N102   M6             Tool change
N103   M3 S500        Direction of rotation/speed
N104   GO X0 Y0       Position axes to zero
N105   D1             Call tool number from geometry table
                      (D compensation)
N106   P9990113       Call inside circle segment cycle
       100,           Centre point in X (mm, absolute)
       100,           Centre point in Y (mm, absolute)
       1000,          Milling feed (mm/min)
       0,             Infeed plane (mm, absolute)
       20,            Depth (mm, incremental)
       5,             Maximum depth per revolution (mm, incremental)
       100,           Diameter (mm, incremental)
       3,             Direction of rotation
       20,            Approach radius (mm, incremental)
       90,            Starting angle (degrees, absolute)
       180,           Milling angle (degrees, absolute)
       1]             Return to starting point in G2/G3
N107   M5
etc.
```

Messages / Plausibility

MILLING FEED <=0
DEPTH <=0
MAX. INFEED PER REVOLUTION <=0
MAX. INFEED PER REVOLUTION > DEPTH
CUTTER RAD. >= (HALF DIAMETER * 0.9)
DIAMETER <=0
FALSE DIRECTION OF ROTATION
ACTIVE CUTTER RADIUS <=0
APPROACH RADIUS <= CUTTER RADIUS
APPROACH RADIUS >= HALF DIAMETER
RETURN TO STARTING POINT FALSE
CUTTER ANGLE <=0

5.3.5 Outside Circle Segment



Programming Example

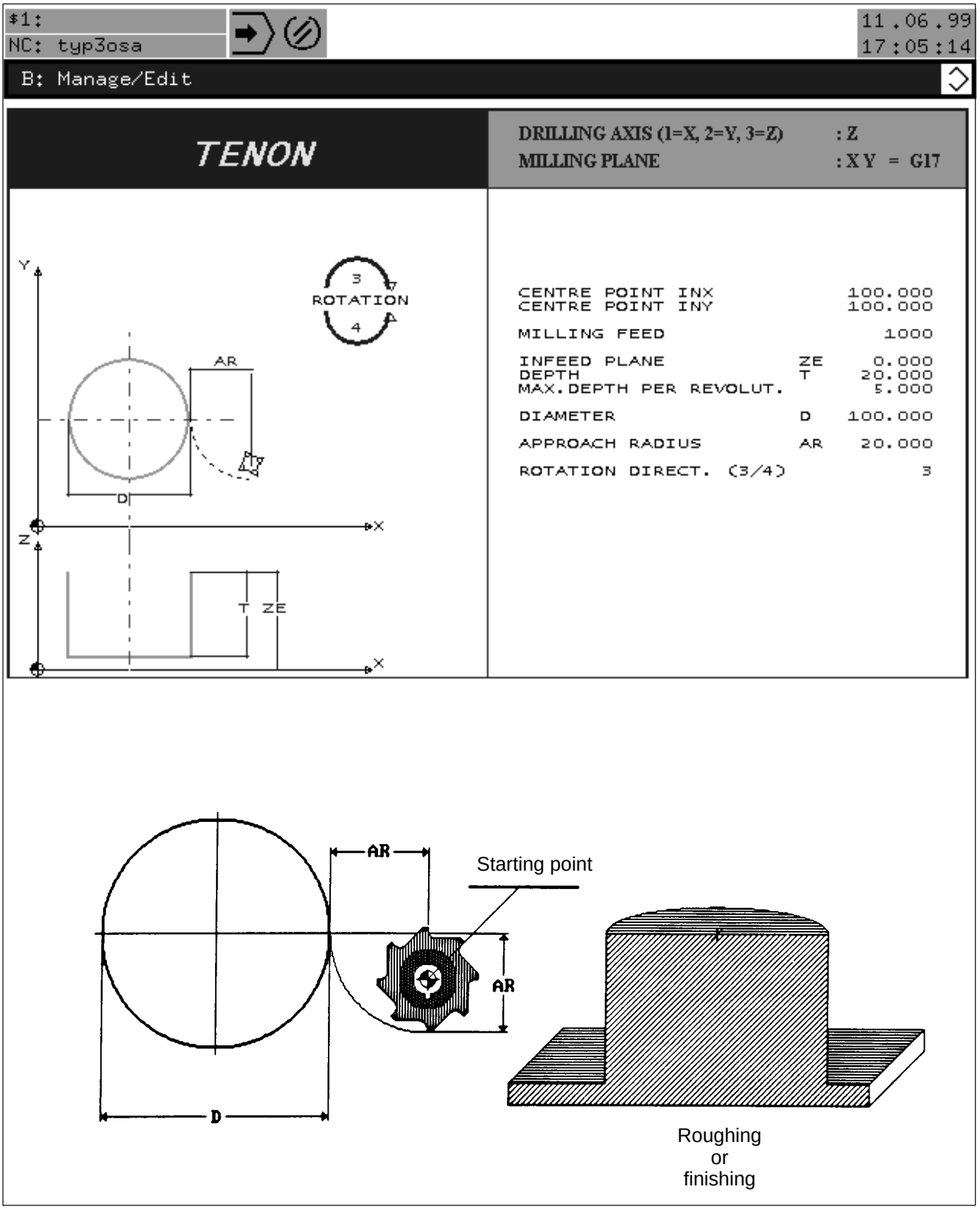
Milling an outside circle segment.

```
.
.
N101   T1234        Tool number
N102   M6           Tool change
N103   M3 S500      Direction of rotation/speed
N104   GO X0 Y0     Position axes to zero
N105   D1           Call tool number from geometry table
                    (D compensation)
N106   P9990115     Call outside circle segment cycle
       100,         Centre point in X (mm, absolute)
       100,         Centre point in Y (mm, absolute)
       1000,        Milling feed (mm/min)
       0,           Infeed plane (mm, absolute)
       20,          Depth (mm, incremental)
       5,           Maximum depth per revolution (mm, incremental)
       200,         Diameter (mm, incremental)
       3,           Direction of rotation
       20,          Approach radius (mm, incremental)
       90,          Starting angle (degrees, absolute)
       180,         Milling angle (degrees, absolute)
       1]           Return to starting point in G2/G3
N107   M5
etc.
```

Messages / Plausibility

MILLING FEED <=0
DEPTH <=0
MAX. INFEED PER REVOLUTION <=0
MAX. INFEED PER REVOLUTION > DEPTH
CUTTER RAD. >= (HALF DIAMETER * 0.9)
DIAMETER <=0
FALSE DIRECTION OF ROTATION
ACTIVE CUTTER RADIUS <=0
APPROACH RADIUS <= CUTTER RADIUS
RETURN TO STARTING POINT FALSE
CUTTER ANGLE <=0

5.3.6 Tenon


$1:
NC: typ3osa
11.06.99
17:05:14
B: Manage/Edit
TENON
DRILLING AXIS (1=X, 2=Y, 3=Z) : Z
MILLING PLANE : X Y = G17
3
ROTATION
4
Y
AR
D
X
Z
T ZE
X
CENTRE POINT INX 100.000
CENTRE POINT INY 100.000
MILLING FEED 1000
INFEED PLANE ZE 0.000
DEPTH T 20.000
MAX. DEPTH PER REVOLUT. 5.000
DIAMETER D 100.000
APPROACH RADIUS AR 20.000
ROTATION DIRECT. (3/4) 3

AR
Starting point
AR
D
Roughing
or
finishing

Programming Example

Milling a tenon.

```
.
.
N101    T1234          Tool number
N102    M6             Tool change
N103    M3 S500        Direction of rotation/speed
N104    GO X0 Y0       Position axes to zero
N105    D1             Call tool number from geometry table
                       (D compensation)
N106    P9990114       Call tenon cycle
        100,           Centre point in X (mm, absolute)
        100,           Centre point in Y (mm, absolute)
        1000,          Milling feed (mm/min)
        0,             Infeed plane (mm, absolute)
        10,            Depth (mm, incremental)
        10,            Maximum depth per revolution (mm, incremental)
        200,           Diameter (mm, incremental)
        3,             Direction of rotation
        20]            Approach radius (mm, incremental)
N107    M5
etc.
```

Messages / Plausibility

MILLING FEED <=0
DEPTH <=0
MAX. INFEED PER REVOLUTION <=0
MAX. INFEED PER REVOLUTION > DEPTH
DIAMETER <=0
FALSE DIRECTION OF ROTATION
ACTIVE CUTTER RADIUS <=0
APPROACH RADIUS <= CUTTER RADIUS

Notes:

6 Technology parameters

The following parameters can be defined in block 1 of cycle P9990100.

These parameters determine the technology of the available milling cycles and should only be modified when required.

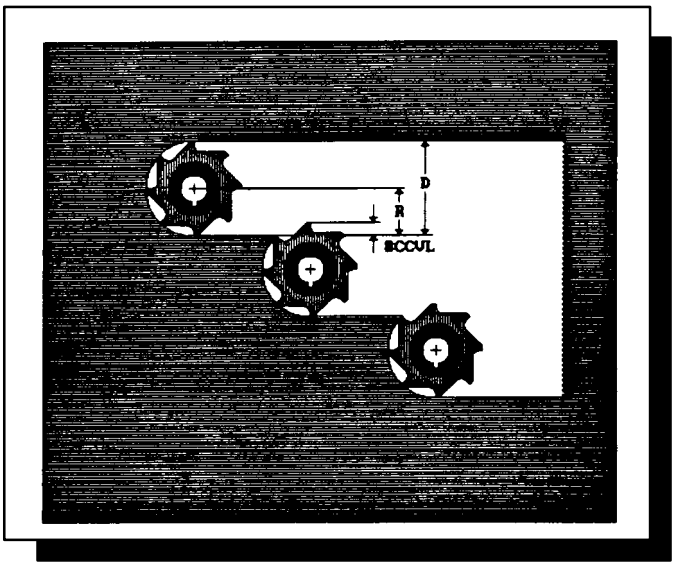
6.1 Overlap Factor (#CCVL)

This parameter defines the size of the cutter overlap. Entering 0.1 means 10% overlap of the cutter radius.

For example, if you choose a milling cutter with diameter = 20mm, specifying #CCVL= 0.1 would produce an overlap of 1mm.

[(D/2] * #CCVL = 10mm * 0.1 = 1mm]

BASIC VALUE = 0.1

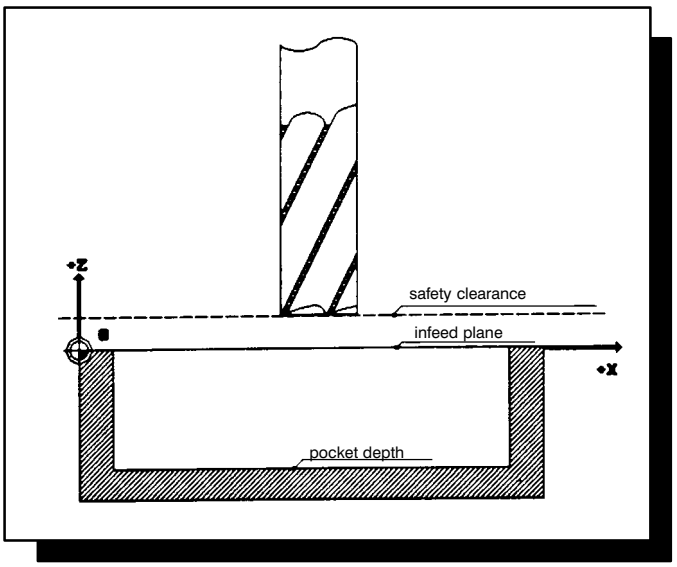


6.2 Safety Clearance (#CCVS)

This parameter is used in two ways in the milling cycles. An input of 1 is equivalent to 1 mm (incremental).

6.2.1 Reverse Rapid Traverse

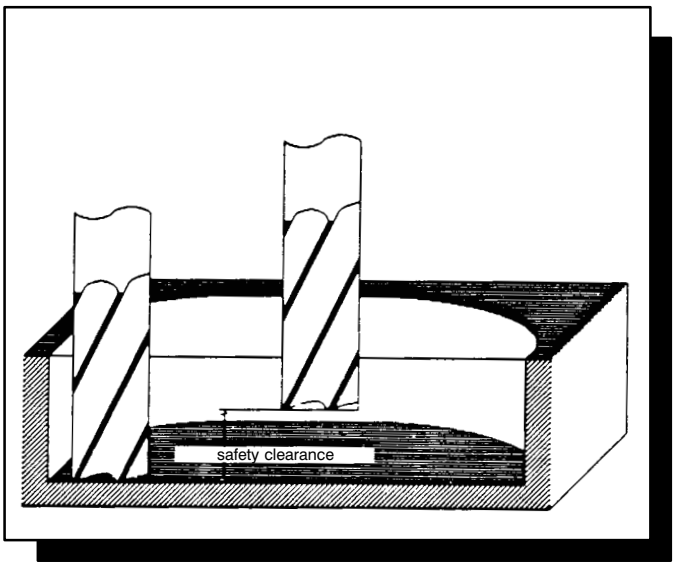
In accordance with the value entered for #CCVS, the rapid traverse travel of the infeed axis is reversed before reaching the infeed plane.



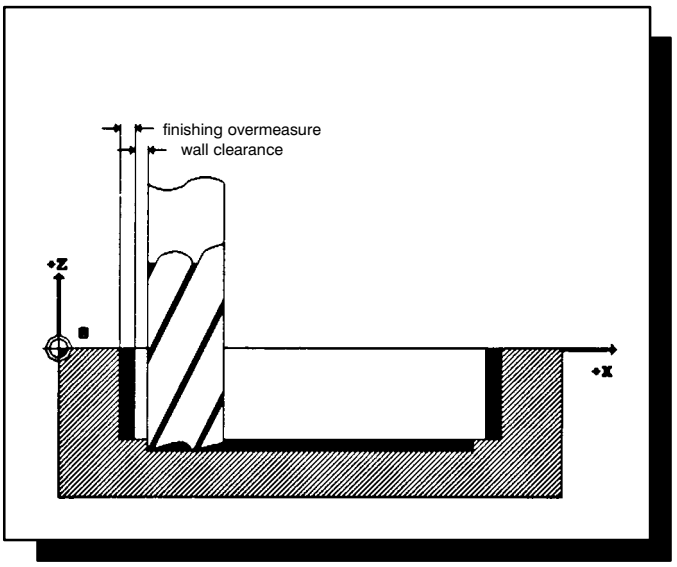
6.2.2 Partial Retraction

If there are several depth infeeds, after machining one depth the infeed axis is partially retracted in accordance with the value entered for #CCVS.

BASIC VALUE = 1 mm



6.3 Wall Clearance during Base Finishing (#CCVW)



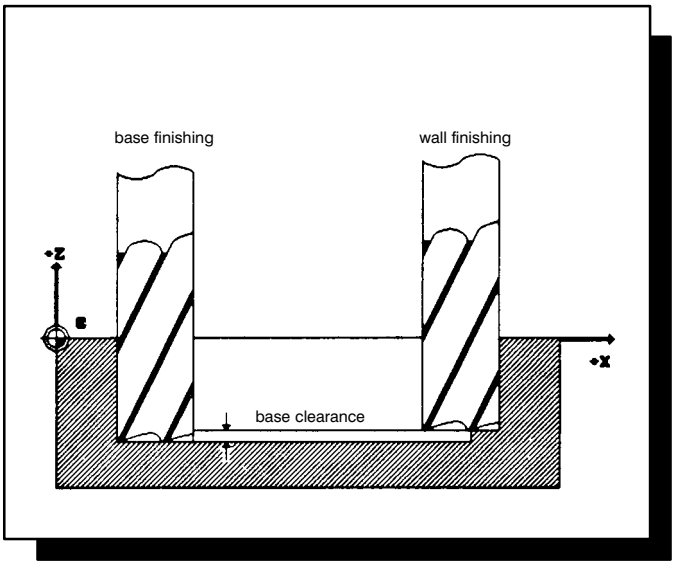
During base finishing, the cutter must not touch the roughed wall for technological reasons.

An input of 0.01 is equivalent to 0.01 mm (incremental).

For example, if you specify #CCVW = 0.01, the cutter will remain 0.01 mm clear of the wall during base finishing.

BASIC VALUE = 0.01

6.4 Base Clearance during Wall Finishing (#CCVB)



During wall finishing, the cutter must not touch the finished base for technological reasons.

An input of 0.01 is equivalent to 0.01 mm (incremental).

For example, if you specify #CCVB = 0.01, the cutter is retracted 0.01 mm from the base after finishing the base.

BASIC VALUE = 0.01

Notes:

7 Installing the machine cycles

7.1 Installation

The dialogues and milling cycles/hole patterns are stored as separate files on CD-ROM.
The Typ3 osa cycle package ”B_m_zyk.exe” contains zipped files for the following CPL cycles:

- Drilling cycles
- Drilling pictures
- Milling cycles
- Contour milling cycles

1. For installation of the cycles it is necessary to copy the file ’B_m_zyk.exe’ in any directory of the Typ3 osa panel hard disk.
   The unzipping of the files takes place automatically into these directories, which are needed for the execution of the cycles.

   After installation on the Typ3 osa PC panel the directory ’c:\typ3pcp\cncfiles\rbzyklen’ is generated. In this directory all necessary files are copied.
   The used BMP files are copied into the directory ’c:\typ3pcp\bin\cplbmp’

2. Terminate all running ”Typ3 osa”-application files.
   Mount the PC directory ’c:\typ3pcp\cncfiles’ for the control. This can be carried out with the Typ3 osa tool ’t3config.exe’. Start afterwards the Typ3 osa application files again.

3. The file ’cpldlg05.dlg’ (call of the main program) must be copied into the control-root directory or user-FEPROM.
   The file’cpldlg05.dlg’ is located in the mount directory ’c:\typ3pcp\cncfiles\rbzyklen’ of the PC panel (for copiing into the OM (Operating mode) ’Manage’ change via soft keys (SK’s) ’switch Directory’ and ’Subdirectory (+)’ into the subdirectory ’rbzyklen’).

4. The files, depending on the language ’Zyktexte.049’,and’Zyktexte.044’ must be copied into the user-FEPROM (directory ’/usrfep’). The files (e.g.’Zyktexte.049’) is always located in the mounted directory ’c:\typ3pcp\cncfiles\rbzyklen’ of the operating panel.

5. All further files have to be stored in a directory, which is a directory for subprograms (compare parameterblock 308000001), or the MACODA parameterblock 308000001 has to be adapted accordingly. The parameterblock 308000001 fixes the sequence of the surching criterias for the CPL-cycles.

   Example: Parameterblock 308000001: Surching path for subprograms

   Parameter 0 .
   1 /usr/
   2 user
   3 mtb
   4 Bosch
   5 /usrfep
   6 /feprom
   7 /mnt/rb>
   8 zyklen
   9

6. With the program ’Linken.cpl’ it is possible to generate all necessary linktables. The programm must only be started in the first channel. The programmexecution is protocolled in the file ’Linken.log’.

7.2 Installed programs

Installed CPL-Programs in the directory ’c:\typ3pcp\cncfiles\rbzyklen’:

Mainprograms

- Hauptdlg.dlg — Overview over the Bosch standard cycles
- Bohrbild.dlg — Mainprogram and overview over drilling pictures
- Bohren.dlg — Mainprogram and overview over drilling cycles
- Kontur.dlg — Mainprogram and overview over contour-milling cycles
- Taschen.dlg — Mainprogram and overview over milling cycles
- Readme.txt — Info-file for installation of the Bosch standard cycles

Subprograms

- Bohrachs.rb — Subprogram for viewing and definition of the drilling axis
- CPL_Rahmen.rb — Subprogram for drawing of CPL grafic frame
- check_NK — Subprogram for elimination of non used digits behind the comma
- RB_Zyklen.ini — Subprogram for initialization of text and colors
- 999999194 — Subprogram for drawing
- 999999195 — Subprogram for drawing
- 999999196 — Subprogram for drawing
- 999999197 — Subprogram for drawing
- 999999198 — Subprogram for drawing
- 999999199 — Subprogram for drawing

Direct program calls from CPL dialog programs

ZYK_G81	CPL-dialog program for drilling cycle G81
ZYK_G82	CPL-dialog program for drilling cycle G82
ZYK_G83	CPL-dialog program for drilling cycle G83
ZYK_G84	CPL-dialog program for drilling cycle G84
ZYK_G85	CPL-dialog program for drilling cycle G85
ZYK_G86	CPL-dialog program for drilling cycle G86
ZYK_G184	CPL-dialog program for drilling cycle G184
ZYK_102	CPL-dialog program for pocket milling cycle ‘rectangular pocket‘
ZYK_103	CPL-dialog program for pocket milling cycle ‘circular pocket‘
ZYK_104	CPL-dialog program for pocket milling cycle ’groove’
ZYK_106	CPL-dialog program for pocket milling cycle ’end groove‘
ZYK_110	CPL-dialog program for contour pocket ’inside frame’
ZYK_111	CPL-dialog program for contour-milling cycle ’outside frame’
ZYK_112	CPL-dialog program for contour-milling cycle ’borehole’
ZYK_113	CPL-dialog program for contour-milling cycle ’inside circle segment’
ZYK_114	CPL-dialog program for contour-milling cycle ’tenon’
ZYK_115	CPL-dialog program for contour-milling cycle ’outside circle segment’
ZYK_150	CPL-dialog program for hole series
ZYK_151	CPL-dialog program for hole circles
ZYK_152	CPL-dialog program for hole grid

Auxilary programs

Linken.cpl	Create link- and sublink tables
Linkliste.txt	Data file for the program ’Linken.cpl’

7.3 Adaption of MACODA datas

1. Parameterblock 909800002: Amount of channels
   The input value has to be 1 higher than requested amount of channels.
   Example: 4 useable channels –> Input value = 5

☞ **For creating this parameter the user-identification has to be not identical with' Endanwender'.**

2. Parameterblock 308000005: CPL-dialog-channel
   In this channel all programs are executed, which are started with Softkey 'CPL-Dialog'.
   Example: 4 –> CPL dialog programs start in 4. channel

3. Parameterblock 308000003: Data areas for CPL-programs
   - Parameter 0: 6144 (Stack)
   - Parameter 1: 65000 (Link tables)

☞ **This parameter is a channel dependant parameter.**
The adjustment has to be according to CPL-dialog-channel (parameterblock 308000005).
After selection of this parameterblock you are in 'System Control Reset' and you can select by Softkey 'next channel' the CPL-dialog-channel for the changes.

4. Parameterblock 308000006: Selection of cycle name and CPL-dialog-program

List with cycle names:

Parameter	Value
0	G81
1	G82
2	G83
3	G84
4	G85
5	G86
6	G184
7	P9990102
8	P9990103
9	P9990104
10	P9990106
11	P9990110
12	P9990111
13	P9990112
14	P9990113
15	P9990114
16	P9990115
17	P9990150
18	P9990151
19	P9990152

☞ **After the next start up of the control the changes of MACODA values are activated.**

5. Parameterblock 308000007: Selection of cycle name and CPL-dialog-program

List of the CPL-dialog-programs:

Parameter	Value
1	ZYK_G82
2	ZYK_G83
3	ZYK_G84
4	ZYK_G85
5	ZYK_G86
6	ZYK_G184
7	ZYK_102
8	ZYK_103
9	ZYK_104
10	ZYK_106
11	ZYK_110
12	ZYK_111
13	ZYK_112
14	ZYK_113
15	ZYK_114
16	ZYK_115
17	ZYK_150
18	ZYK_151
19	ZYK_152

☞ **After the next start up of the control the changes of the MACODA values are activated.**

7.4 Troubleshooting

Possible reasons for failors:

1. Part program failor 452: 'Too less storage for program'
   Part program failor 465: 'Too less storage for program'
   Remedy: Parameter block 308000003 increase parameter 'Linktabellen' for CPL-dialog-channel.

2. Part program failor 782: 'No runtime memory available'
   Remedy: Parameterblock 308000003 increase parameter 'Stack' for CPL-dialog-channel.

3. Part program failor 739: 'No empty CPL window available'
   possible reason: Application 'CPL-Dialog' not yet started.'CPL-Dialog' allready started, e.g. from a different channel.

4. Part program failor 1502: 'Unzulaessiger Dateiname. –> RB_Zyklen.ini'
   possible reason: The language depending text file is not available in user-FEPROM.
   Remedy: Copy text file, e.g. 'Zyktexte.049' into user-FEPROM.

5. Part program failor137: 'File is protected. –> RB_Zyklen.ini'
   possible reason: The language depending text file has no reading access.
   Remedy: Set access rights for text file.

6. CPL-dialog-program does not start, even the softkey is available in editor
   possible reason: The CPL-dialog-channel value is bigger, than the amount of channels in the system.
   Remedy: Check the parameterblocks 308000005 und 909800002.
   Example: 308000005 = 4, CPL-dialog opertates in channel 4
   909800002 = 5, system contains 4 channels

7. Softkey 'CPL-Dialog' not available in the editor
   possible reason: The file 'cpldlg05.dlg' is neither available in the root directory nor in the user-FEPROM.
   Remedy: Copy file in root directory or user-FEPROM.

8. MACODA-Block 909800002 is missing in the block selection box.
   possible reason: The user-identification is equal 'Enduser'.
   Remedy: Set the user-identification non equal 'Enduser' (set GOM Diagnostics,Install,Macoda,Organise,User).

A Appendix

A.1 Index

Bosch-Automationstechnik

Robert Bosch GmbH
Geschäftsbereich
Automationstechnik
Industriehydraulik
Postfach 30 02 40
D-70442 Stuttgart
Telefax (07 11) 8 11-18 57

Robert Bosch GmbH
Geschäftsbereich
Automationstechnik
Fahrzeughydraulik
Postfach 30 02 40
D-70442 Stuttgart
Telefax (07 11) 8 11-17 98

Robert Bosch GmbH
Geschäftsbereich
Automationstechnik
Pneumatik
Postfach 30 02 40
D-70442 Stuttgart
Telefax (07 11) 8 11-89 17

Robert Bosch GmbH
Geschäftsbereich
Automationstechnik
Montagetechnik
Postfach 30 02 07
D-70442 Stuttgart
Telefax (07 11) 8 11-77 77

Robert Bosch GmbH
Geschäftsbereich
Automationstechnik
Antriebs- und Steuerungstechnik
Postfach 11 62
D-64701 Erbach
Telefax (0 60 62) 78-4 28

Robert Bosch GmbH
Geschäftsbereich
Automationstechnik
Schraub- und Einpreßsysteme
Postfach 11 61
D-71534 Murrhardt
Telefax (0 71 92) 22-1 81

Robert Bosch GmbH
Geschäftsbereich
Automationstechnik
Entgrattechnik
Postfach 30 02 07
D-70442 Stuttgart
Telefax (07 11) 8 11-34 75

Robert Bosch GmbH
Geschäftsbereich
Automationstechnik
Schulung AT/VSZ
Berliner Straße 25
D-64711 Erbach
Telefax (0 60 62) 78-8 33

Technische Änderungen vorbehalten

Ihr Ansprechpartner

BOSCH

Robert Bosch GmbH
Geschäftsbereich
Automationstechnik
Antriebs- und Steuerungstechnik
Postfach 11 62
D-64701 Erbach
Telefax (0 60 62) 78-4 28

1070 073 795-101 (99.07) GB · HB NC· AT/PLS · Printed in Germany

Begleitakte für:
Typ3 osa

Universal Milling Cycles

1070 073 795-101 (99.07) GB

Erstausgabe	⊗	Produktbetreuer	Prößdorf
Überarbeitung	○	erstellt mit	Interleaf
Übersetzung	⊗	angelegt am	21.07.99
Autor	Grohmann	Seiten	80

Aufgabe		**Mitarbeiter**	**Datum**
Bestellnummer anlegen oder Index erhöhen			
In Budgetplan aufnehmen			
Konzept vom Dienstleister erhalten			
Korrekturexemplare vom Dienstleister erhalten			
Korrekturexemplar an AT/VTS	Rückgabetermin:		
Korrekturexemplar an AT/ENC	Rückgabetermin:		
Korrekturexemplar an AT/VSZ	Rückgabetermin:		
Korrekturexemplar an AT/VKS	Rückgabetermin:		
Korrekturexemplar an ErB/QSG6	Rückgabetermin:		
Korrekturexemplar an	Rückgabetermin:		
Abnahmeprotokoll von AT/VTS zurück			
Abnahmeprotokoll von AT/ENC zurück			
Abnahmeprotokoll von AT/VSZ zurück			
Abnahmeprotokoll von AT/VKS zurück			
Abnahmeprotokoll von ErB/QSG6 zurück			
Abnahmeprotokoll von zurück			
Korrekturexemplar geprüft			
Korrekturen im Original eingearbeitet / mit Dienstleister besprochen			
Original vom Dienstleister erhalten und Datei abgelegt			
An Übersetzer	Rückgabetermin:		
Übersetzung zurück und Datei ablegen			
Original fertig			
Bestellnummer im SAP anlegen, Deutsch und Englisch			
Freigabe bei TEF10 beantragen			
TEF10 muß Index im Kurztext erhöhen			
Datei an TEF10 übergeben			
Archivierung abgeschlossen			
Original mit Druckfreigabe an VSZ45			
Im Monatsbericht eintragen			
In Dokuliste-Neu mit Korrekturbalken eintragen			
Abschließend im BudgetPlan eintragen			
In Zielentfaltungsstatistik eintragen			
Datei im Archiv-Verzeichnis ablegen			
IDU im Freigabe-Verzeichnis ablegen			
IDU für PDF an ProLeaf			
Belegexemplare von VSZ45 erhalten			
Verteilt, e–mail			
Korrektur-/Büroexemplar ablegen			
Vorgängerausgabe im Archivschrank ablegen			
PDF für Intranet bereitstellen			

Bemerkungen: