

MTC200/MT-CNC

Graphical NC Editor

18VRS

Application Manual

SYSTEM200

Title	Graphical NC Editor
Documentation Type	Application Manual
Document Code	DOK-MTC200-GNE*BES*V18-AW02-EN-P
Internal file reference	<ul style="list-style-type: none"> • Folder 1 / Register 7 • Drawing number 120-1700-B323-02/EN
Purpose of this document	This documentation serves the machine tool user. It describes the facts required for the use of the Graphical NC Editor.

Configuration control

Documentation identification of previous releases	Release date	Comment
109-0768-4190-00	04/97	New issue
109-0768-4190-01	10/97	Supplement - milling
109-1041-4112-01	03/98	New issue - Version 18
120-1700-B323-02	05/99	Supplement: "1.2 Important information about function key layout"

Copyright © INDRAMAT GmbH, 1999

Copying this document and giving it to others and the use or communication of the contents thereof without express authority are forbidden. Offenders are liable for the payment of damages. All rights are reserved in the event of the grant of a patent or the registration of a utility model or design (DIN 34-1).

Validity All rights are reserved with respect to the content of this documentation and the availability of the product.**Published by** INDRAMAT GmbH • Bgm.-Dr.-Nebel-Str. 2 • D-97816 Lohr a. Main
Phone 09352/40-0 • Tx 689421 • Fax 09352/40-4885

Dept. ESM (JA)

Note This document is printed on paper bleached without chlorine.

Contents

1 Graphical NC Editor (GNE)	1-1
1.1 Preface.....	1-1
1.2 Important Information About Function Key Layout.....	1-2
2 Rules for Using the GNE	2-1
2.1 General Information	2-1
2.2 Programming GNE Instructions	2-3
2.3 Modifying GNE Instructions.....	2-5
2.4 Deleting GNE Instructions.....	2-6
3 Instruction Description	3-1
3.1 WINDOW	3-1
3.2 Geometry Definition	3-2
Contour definition	3-2
Contour correction	3-10
Form elements	3-14
Drilling patterns.....	3-14
Copy instructions	3-15
3.3 Machining Cycles.....	3-17
General information	3-17
Turning Cycles: Cutting, Contouring, Residual Cutting, Grooving, Thread Cutting, Centric Boring	3-19
Milling cycles.....	3-34
Drilling cycles.....	3-39
4 Data Adaptation	4-1
5 NC EDITOR - Special Function F3	5-1
6 Error Messages	6-1
7 Index	7-1

1 Graphical NC Editor (GNE)

1.1 Preface

Function The GNE represents an effective and high-precision aid to support NC part programming. It enables the user to define geometric elements (such as contour elements) and their machining in an easy way.

The GNE generates NC code in the syntax that has been defined for the MTC200 (see NC Programming Instructions 17VRS, DOK-MTC200-NC**PRO*V17-ANW1-DE-P).

Note: Due to the extremely large number of combination possibilities between geometry and machining variation, the user must check the GNE results and run in the program carefully.

Ordering information The GNE is an optional component of the graphical user interface for the MTC200 and can be ordered under the type name SWS-MTC200-GNE-18VRS-MS.

1.2 Important Information About Function Key Layout

The variety of operator panels (BTV01, BTV20, BTV30 and other PCs) requires flexible function key layout on the screen. The controller manufacturer delivers a default function key layout. At the time of delivery, the function key assignment is F1 to F8 and a separated function key/machine key exists. This corresponds to the function key layout of e. g. a BTV01.

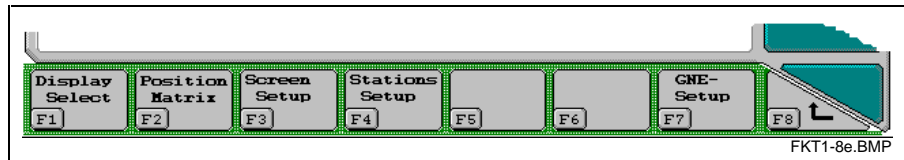


Figure 1-1: Function key layout F1 to F8

The BTV20 function keys F2 to F9 have ist place below the display. To assign the function keys directly to the screen display, it is possible to modify the screen settings. The screen settings can be modified in the 'Screen Setup' menu of the main menu 'Setup'. It is also possible to determine the representation of the softkey on the right. With BTV20 and BTV30 the separated softkey on the right side can be switched to a 'complete' key.

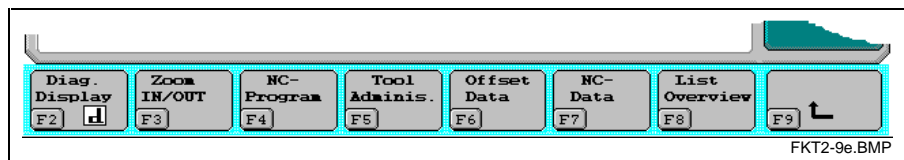


Figure 1-2: Function key layout F2 to F9 with 'complete' right softkey.



CAUTION

Screen representations and notes about the function keys

All screen representations, descriptions and notes about the function keys in this documentation refer to the default settings of the software at the time of delivery.

⇒ Please consider this information if you use the software with another PC or if you changed the screen settings concerning function key layout.

2 Rules for Using the GNE

2.1 General Information

- Application** The GNE provides the possibility of efficiently programming process-typical machining tasks such as roughing turning contours or milling along contours in a conversationally guided and graphically supported way.
- Invocation** Within the NC editor, the GNE is invoked via the soft key <F6> „Graphical NC editor“. The GNE is activated („entry“) before the line that contains the cursor. (see Fig. 2-1).

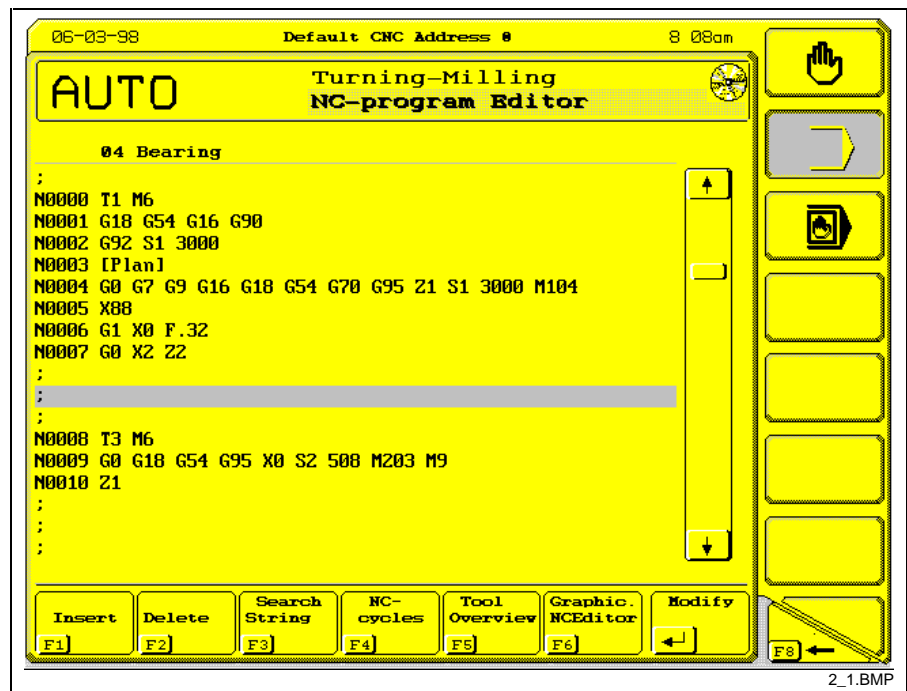


Fig. 2-1: NC editor - invoking the GNE via the F6 soft key

- Dialog** The graphically interactive dialog is supported by a large number of menus and input screens. The dialog guidance has been designed such that the graphical representation in the workpiece window is not covered by the input menus. The programmer has an overview of the most important data of his program without calling up the data input menus.
- Result** The results of the dialog guidance are copied immediately before the „entry“ location.

GNE Instructions

The GNE instructions are subdivided into three main groups:

- Window instructions
- Geometry definition instructions, and
- Machining cycle instructions

Window instructions

The *Window* instructions determine the reference zero point, the scale of representation on the screen, the machining process and the axis assignment within the GNE instructions.

Geometry definitions

Geometry definitions determine the machining location (for example contour definition, contour element).

Machining cycles

Machining cycles are technology subroutines that perform certain machining processes, for example, contour cutting or contour milling.

Exiting GNE

When the GNE is exited and the system returns to the NC editor, the results are inserted exactly at the entry point (see Fig. 2-2) .

Note: It is possible to leave the GNE at any time using the key combination <CTRL>+<E>. All currently made GNE entries will be lost then.

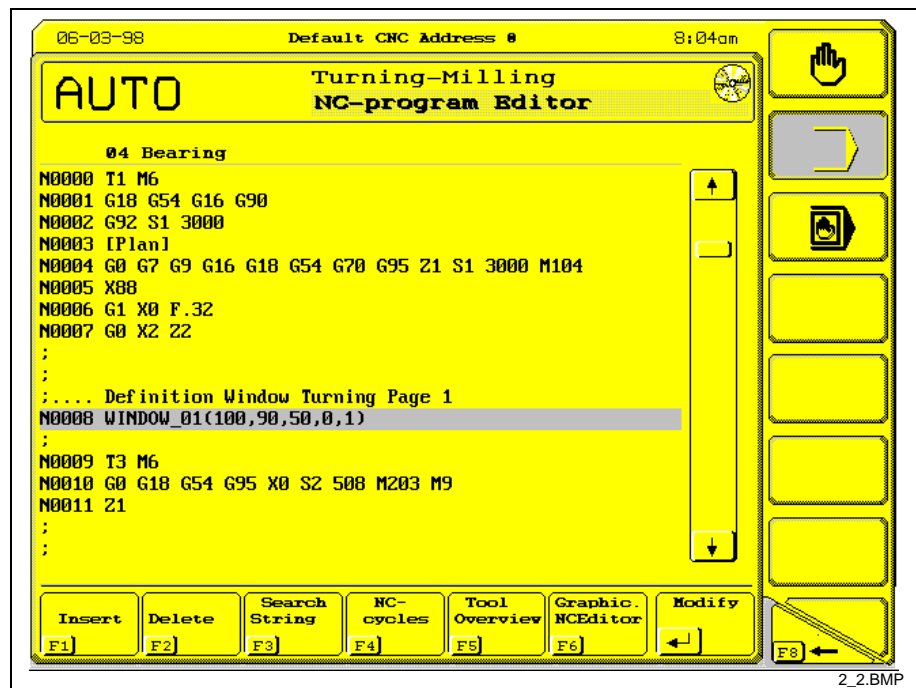


Fig. 2-2: GNE instruction

2.2 Programming GNE Instructions

First step	Position the cursor to the NC program line <i>before</i> which an insertion shall be made. This line must not contain any GNE instruction (see 2.3 Modifying GNE). If it is necessary to insert immediately before a GNE instruction, a comment block (;) must be inserted first, using the NC editor.
Second step	Press the <F6> (GNE) soft key.
First invocation	A WINDOW must be defined prior to calling the GNE main menu if no GNE instruction exists before the point of entry.
Subsequent invocation	An immediate branch is made to the GNE main menu, if a 'WINDOW' has already been defined within the NC program.
Menu	The 'WINDOW, contour and cycles' instructions are called up from the main menu (see Fig. 2-4) or from a menu next in order.
Zoom	The displayed graphic can be scaled down using soft key <F3>, a detail can be selected using soft key <F4> and the original representation can be restored using soft key <F5>.
Units	Only the unit defined in the controller's machine parameter set is valid. It is displayed in the top left corner of the screen.
Exceptions	Note: The parameters of GNE instructions may not have NC variables assigned in the NC program.

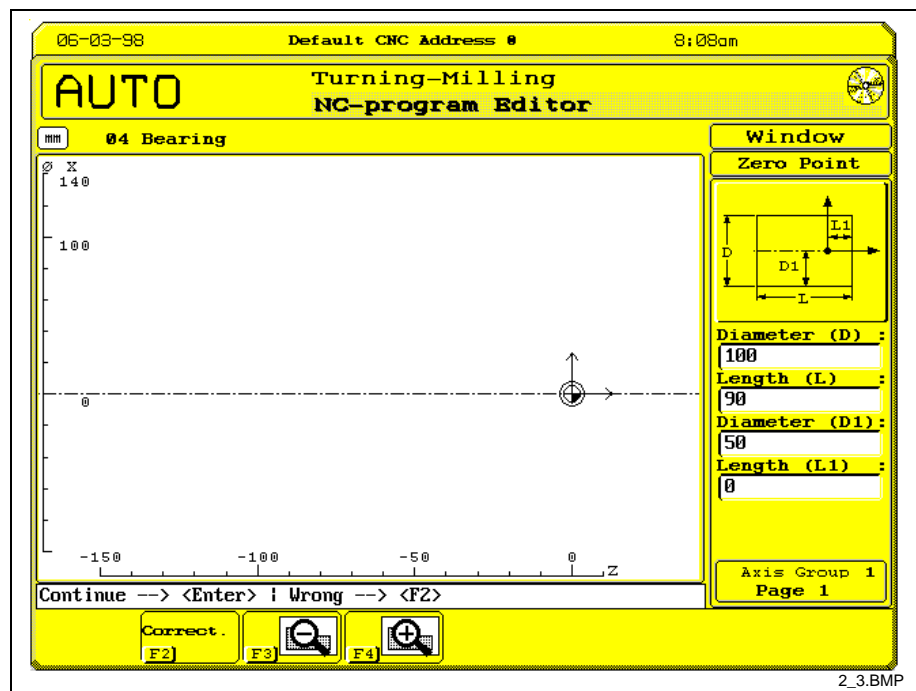


Fig. 2-3: GNE - first instruction - Window instruction

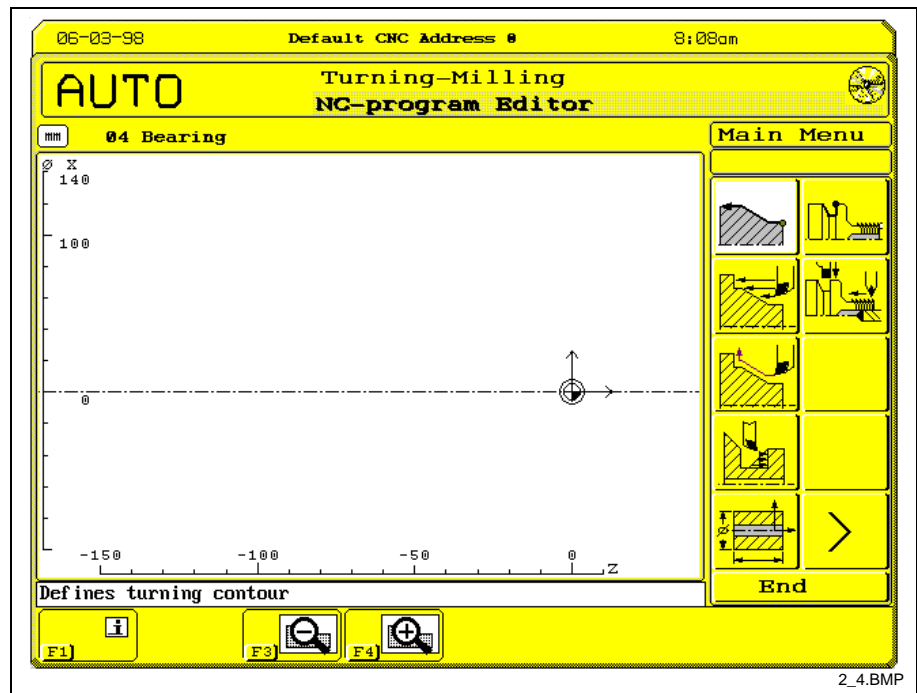


Fig. 2-4: GNE main menu

2.3 Modifying GNE Instructions

- First step** Position the cursor on the NC block that contains the *GNE instruction* which is to be modified.
- Second step** Press soft key <F6> 'Graphic. NC Editor'. An automatic branch is made to the correction of the relevant instruction.
- Single-line instructions** In case of a „WINDOW_“ (see Fig. 2-5) or „CYCLE_“, „FORM_“, or „COPY_“ instruction, the instruction's active parameter are accepted in the conversational screens.
- Multiple-line instructions** In case of geometry definitions that consist of multiple GNE lines, the cursor can be positioned to any instruction within the sequence of instructions.

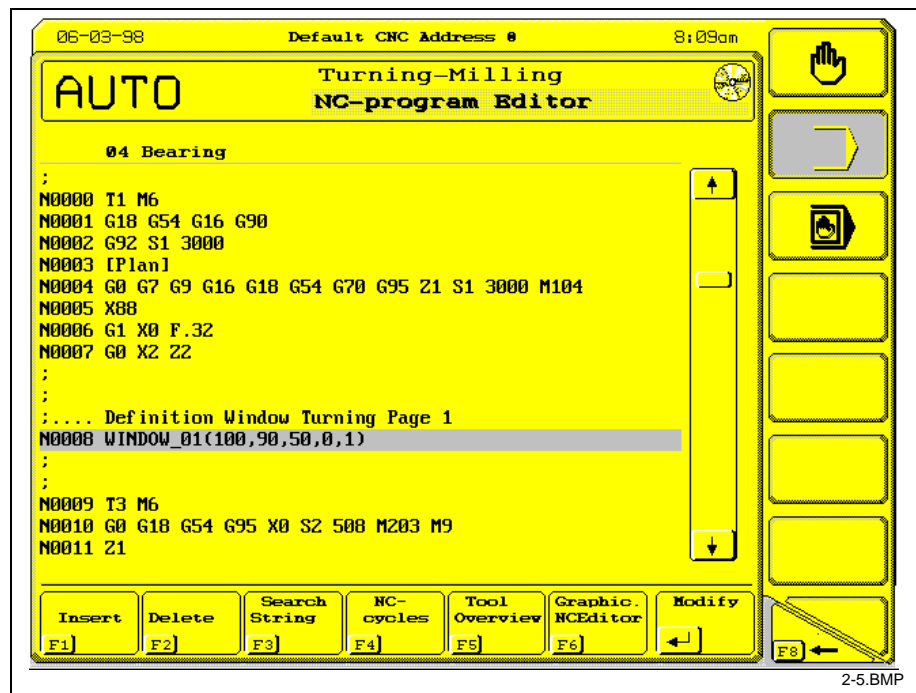


Fig. 2-5: Modifying a Window instruction

2.4 Deleting GNE Instructions

Invocation Use the soft key <F2> to delete a GNE instruction. Please note, that the procedure always follows the inverse sequence of dialog programming.

CYCLE



CAUTION

CYCLE

⇒ This instruction is deleted without any warning.

COPY, FORM

This instruction is deleted after a warning message has been issued.

Note: All instructions that refer to the geometry elements and machining cycles must first be deleted.

CONT

The entire contour definition is deleted (after a warning) if the cursor is positioned on an instruction that belongs to the contour definition.

Note: All instructions that refer to the geometry elements and machining cycles must first be deleted.

Message	Cause/remedial action
Deleting not permitted - reference in block N...	The parts program contains instructions that refer to this instruction. ⇒ Delete those instructions first.

3 Instruction Description

3.1 WINDOW

Function	The 'WINDOW' instructions determine the reference to the zero point, the scale of representation on the screen, the machining process in principle and the axis assignment within the GNE instructions. If multiple axis combinations are possible, they can be selected using soft key <F7>.
Application	It is differentiated between turning (WINDOW_01) and milling/boring (WINDOW_02) technology. The geometry or machining definitions that are to be described later, are in context to the programmed WINDOW. This means that after a WINDOW_01 instruction only geometry and machining instructions are allowed that were developed for turning applications and the related axes. This context is validated during conversational programming.
Invocation	The call is made via the corresponding symbols in the main menu. A branch into a WINDOW instruction is made automatically if no WINDOW instruction was programmed in the preceding blocks before the point of entry.
Programming	Turning: WINDOW_01 (parameter list) Milling: WINDOW_02 (parameter list)
Parameter	The parameters 1...4 define the screen section and the zero point position within the selected screen. Parameter 5 defines the axis assignment number (see 4. Data Adaptation).
Validity	The WINDOW instructions (see Fig. 2-3) are valid until canceled.
Exceptions	Due to a new WINDOW instruction in conversational programming, only the geometric elements are available that have been defined in the WINDOW with the same axis allocation. All other geometry elements are optically hidden (see 4. Data Adaptation). The compatibility of axis group numbers must be observed when parts programs are replaced.
Clamping/chucking positions	If various equal axis groups are defined within a data adaptation (see 4. Data Adaptation), this can be used for rechucking or rotary table operations, because each axis group is handled separately.

3.2 Geometry Definition

Contour definition

Function	The contour definition consists of arcs, lines and transitional elements that describe the contour's geometry. The contour description by itself does not initiate any machine movement.
Invocation	Contour definition call is performed by selecting the relevant symbol in the GNE dialogue.
Start point	Contour definition starts with the localization of a start point (see Fig. 3-1).
Transitional elements	In the following, the contour course is given by transitional element selection (line, right arc, left arc) (see Fig. 3-2).
End	Contour definition is terminated by pressing the menu key END.
Transitions	The kind of transition of a contour element to the subsequent element is determined by selecting tangential - non tangential or by inserting transitional elements (arc, chamfer, relief groove) (see Fig. 3-3 through Fig. 3-6).
<hr/>	
	Note: The participating contour elements must intersect.
<hr/>	
Coordinates	The contour's position and size is determined by coordinate value input. The coordinate values correspond to the contour elements end point. The data input remains until a calculation of the element's geometry can take place or data input is terminated by pressing the '?' key. When '?' is pressed, it must be ensured that the unknown end point coordinates can be calculated internally after inputting the data of the subsequent elements. Unnecessary geometry definition or incorrect input is prevented through the conversational guidance.

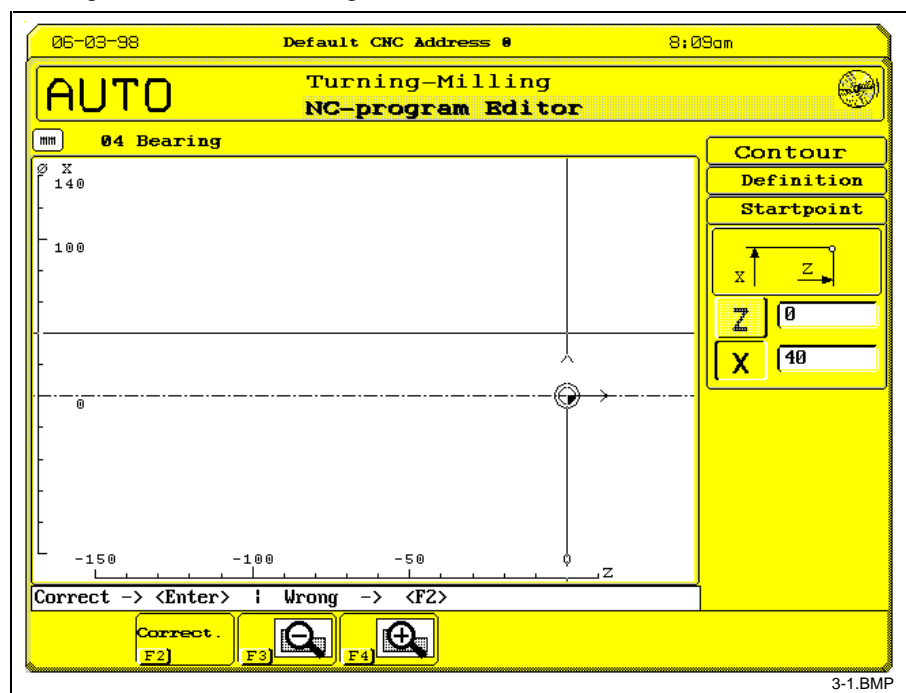


Fig. 3-1: Contour definition - start point

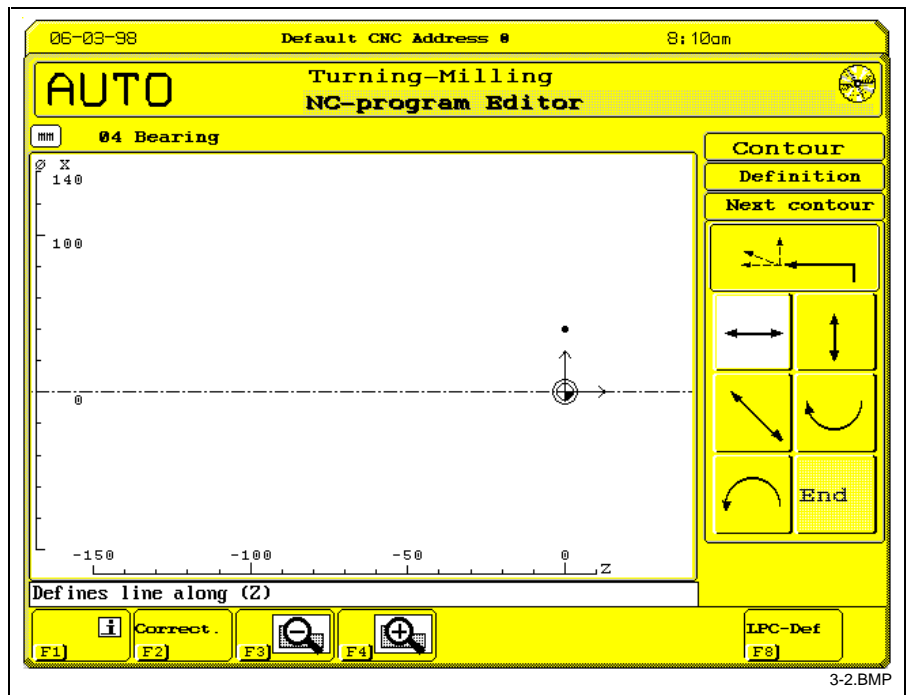


Fig. 3-2: Contour definition - subsequent element selection

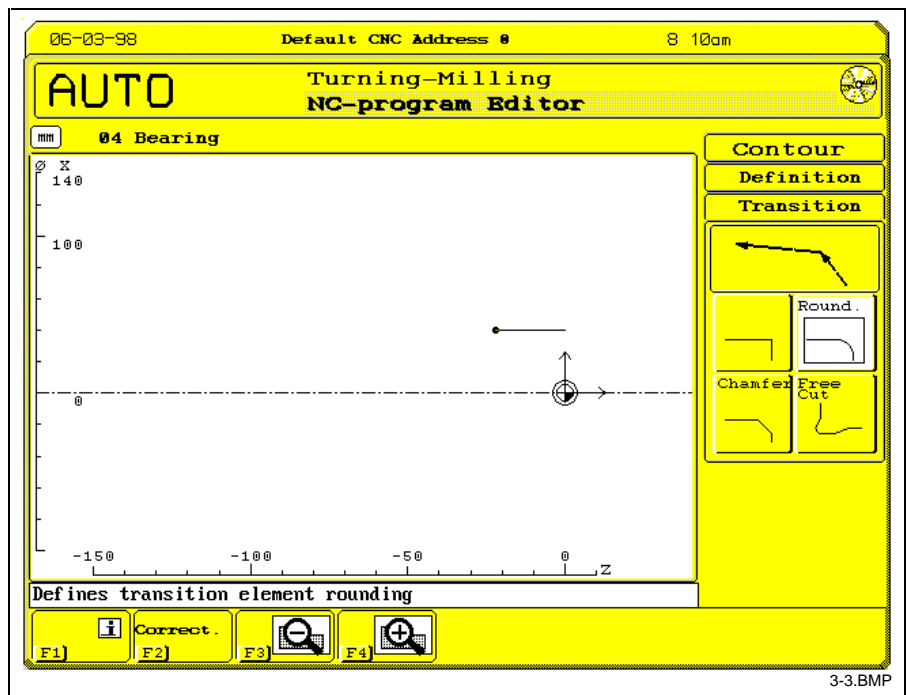


Fig. 3-3: Contour definition - transition element selection

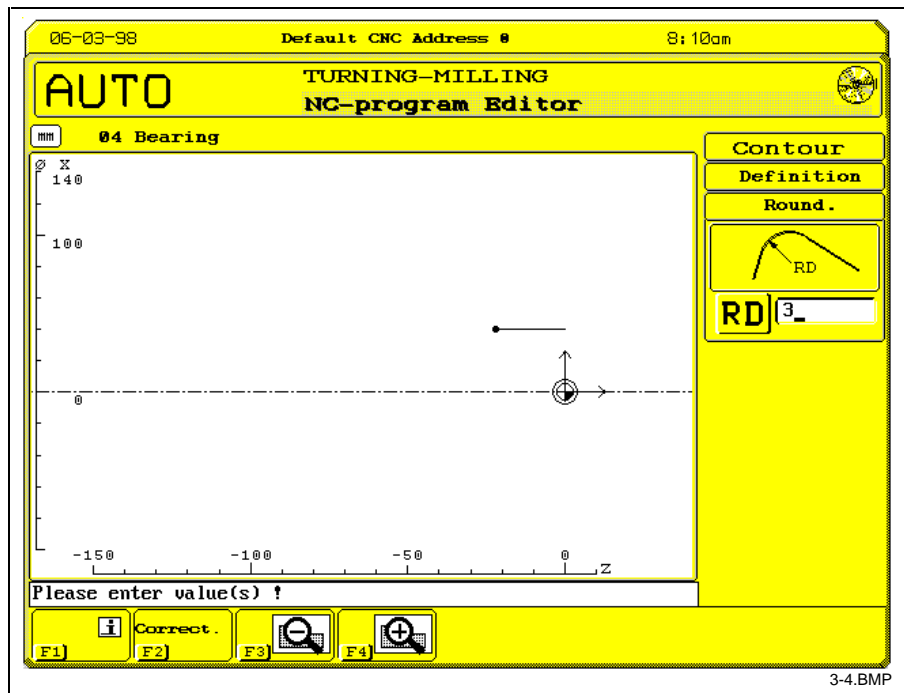


Fig. 3-4: Contour definition - Transition element - radius

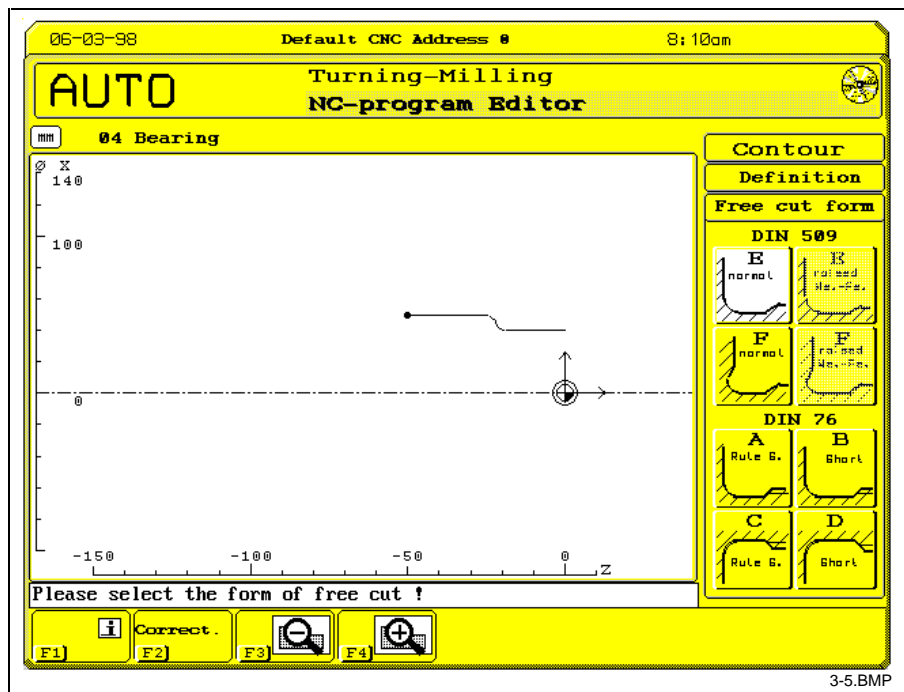


Fig. 3-5: Contour definition - Transition element - DIN relief groove

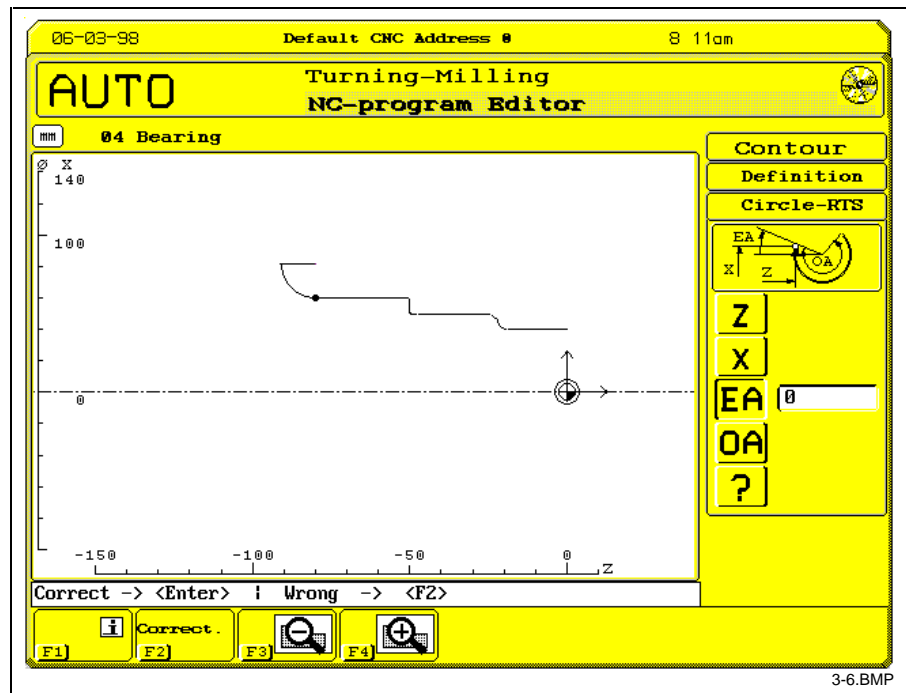


Fig. 3-6: Contour definition - arc

Auxiliary constructions

Within contour definition there is a possibility of using auxiliary functions for point, line and arc calculations by pressing soft key <F8> (see Fig. 3-7 through Fig. 3-11). The calculation results can then be accepted in the contour definition via cursor selection (soft key <F6>).

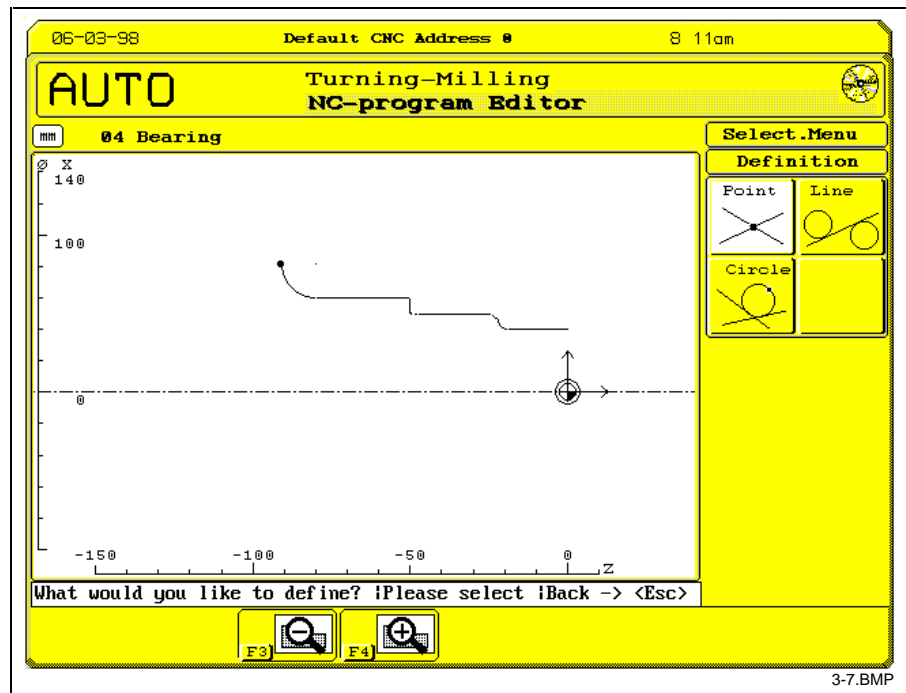


Fig. 3-7: Contour definition - point - line - arc selection

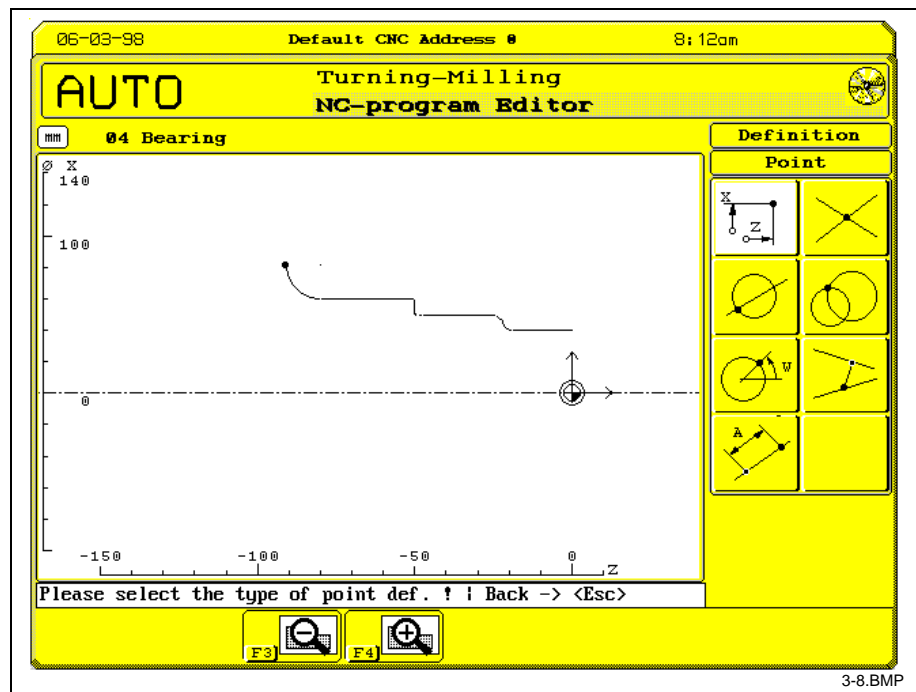
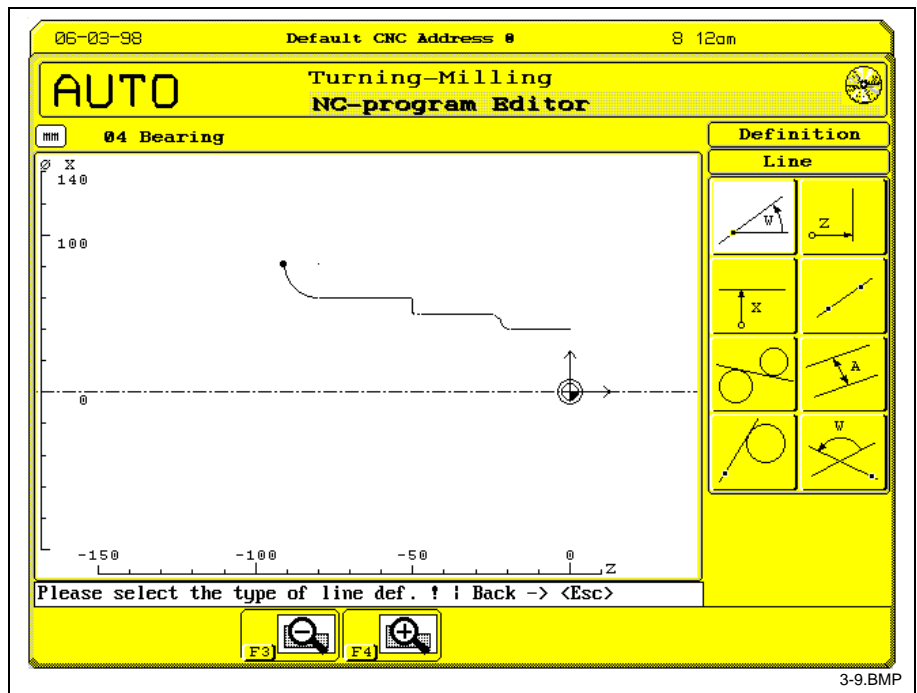
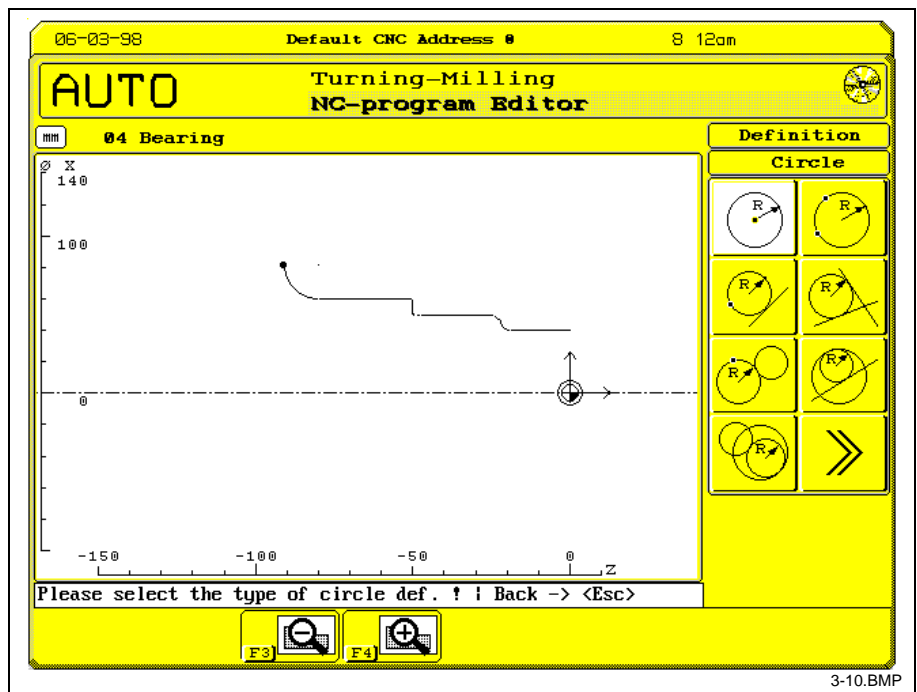


Fig. 3-8: Point definition



3-9.BMP

Fig. 3-9: Line definition



3-10.BMP

Fig. 3-10: Circle definition I

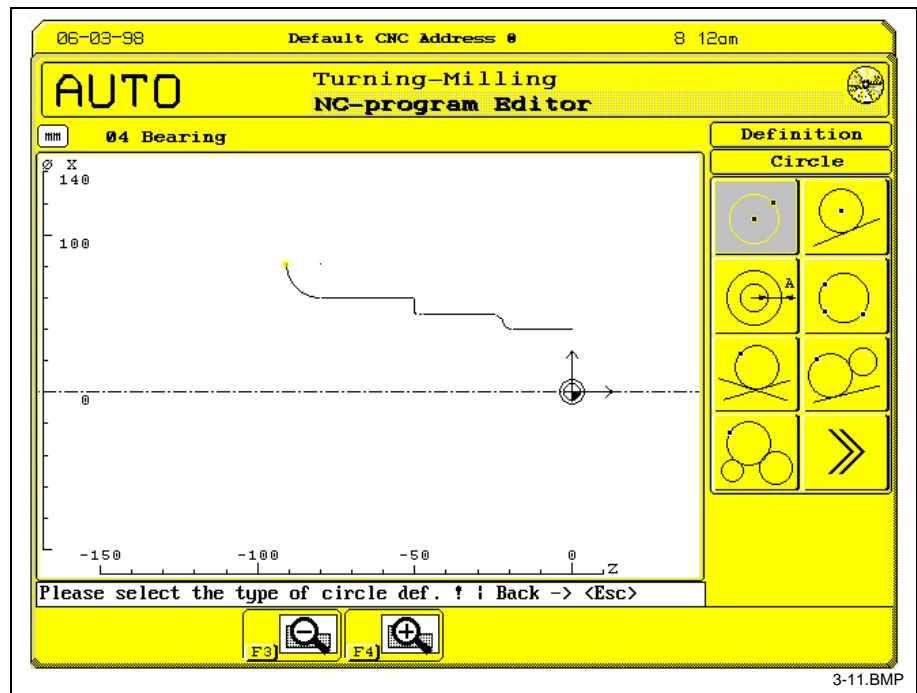


Fig. 3-11: Circle definition II

Fits

During the contour definition of a turning part, the GNE permits tolerance allowances of fits (DIN 7154T1) to be displayed after nominal dimensions, code and quality have been entered. This permits the allowances required for production to be entered as coordinate values into the contour definition (see Fig. 3-12 through Fig. 3-14) .

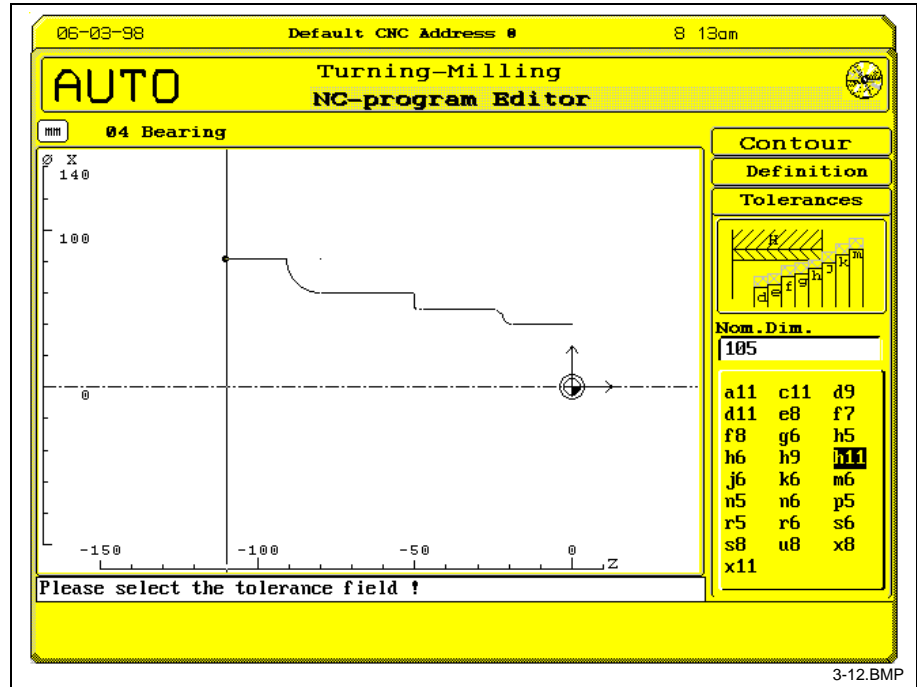


Fig. 3-12: Fit selection - nominal diameter h11

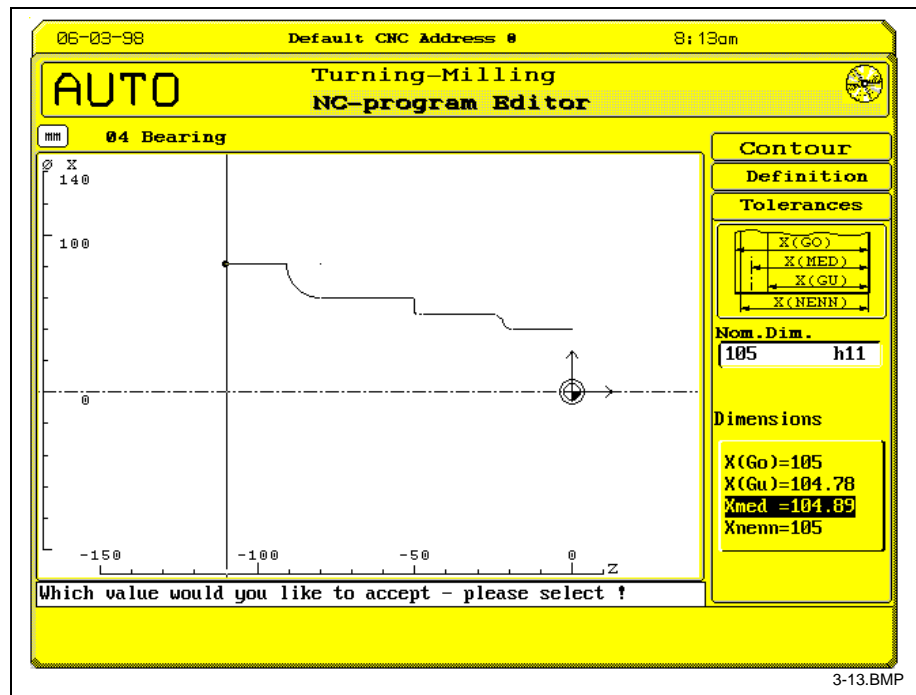


Fig. 3-13: Fit selection- medium diameters

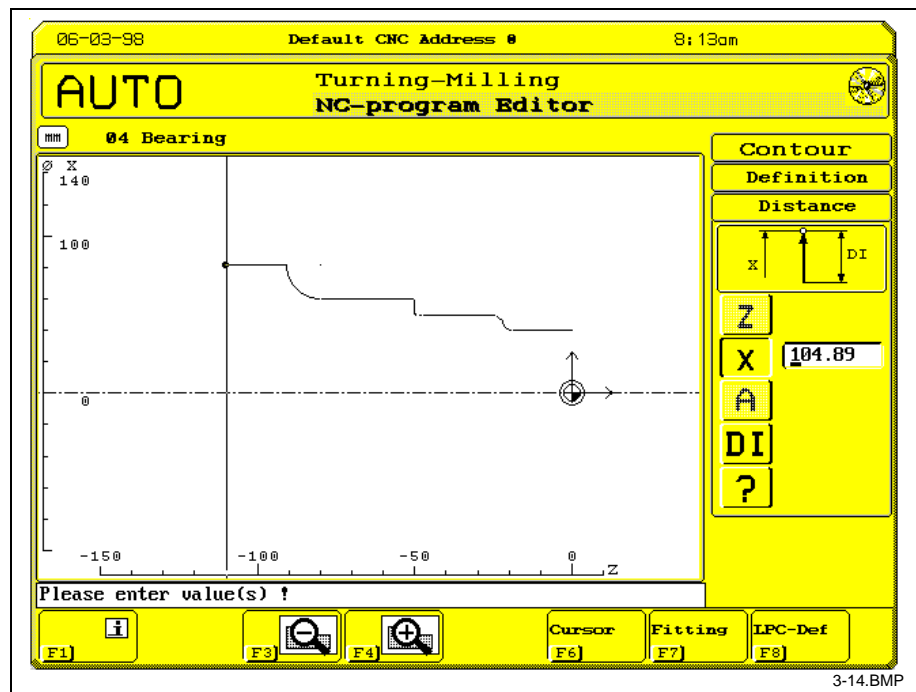


Fig. 3-14: Fits - return to contour definition

Overview of the Principles of the GNE Contour Definition Syntax

Main word	Explanation
CONT (a, b, c)	Contour header a= No. Of the contour b= (0= outside 1= inside for turning contours) 0= milling contour c= Axis assignment (see 3.1 WINDOW - parameter - or 4. Data Adaptation)
START	Contour start point
LINE...	Line
CCW...	Circle - counter-clockwise
CW...	Circle - clockwise
RD...	Corner rounding
CF...	Chamfer
RELIEF...	Relief groove
END_CONT	Contour end

Note: The GNE geometry's entire number of elements defined in an NC program (arc, line, corner rounding, chamfer=1 element, relief groove=3 elements) is limited to 200 elements.

Contour correction

Invocation Contour correction dialogue entry during contour definition is performed via soft key <F2>. In case of a contour already defined in the NC program, the cursor is positioned to any NC block of the contour definition and soft key <F6> is pressed (see Fig. 3-15).

Note: Contour modification should always be carried out using the GNE dialogue.

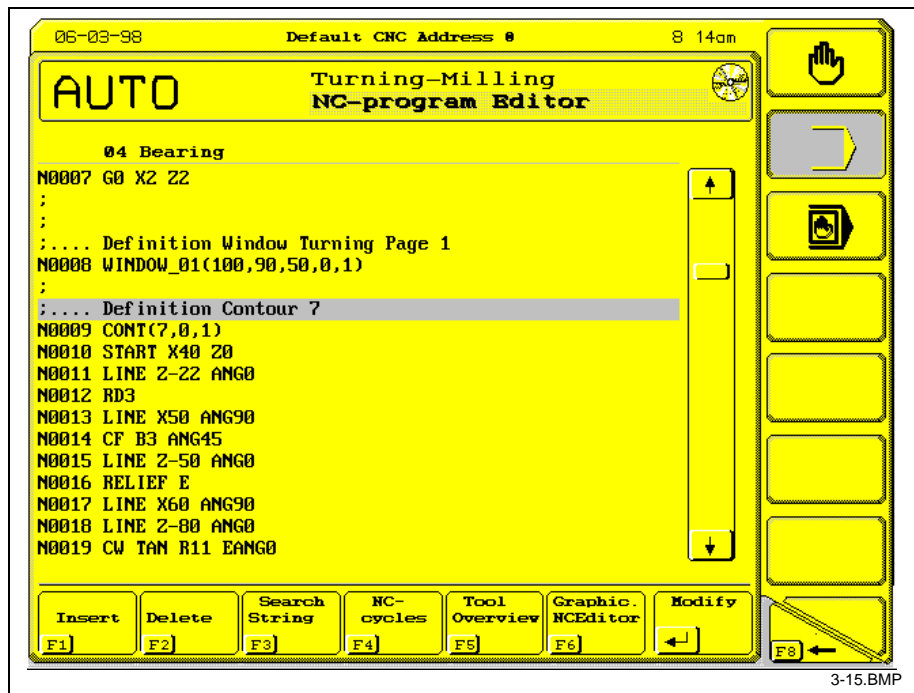


Fig. 3-15: Invocation - correction of a GNE contour

The contour correction dialogue contains the following options (see Fig. 3-16):

- Deleting contour and transition elements
- Inserting transition elements
- Modifying contour and transition elements
- Extending contours (at the end of the contour)

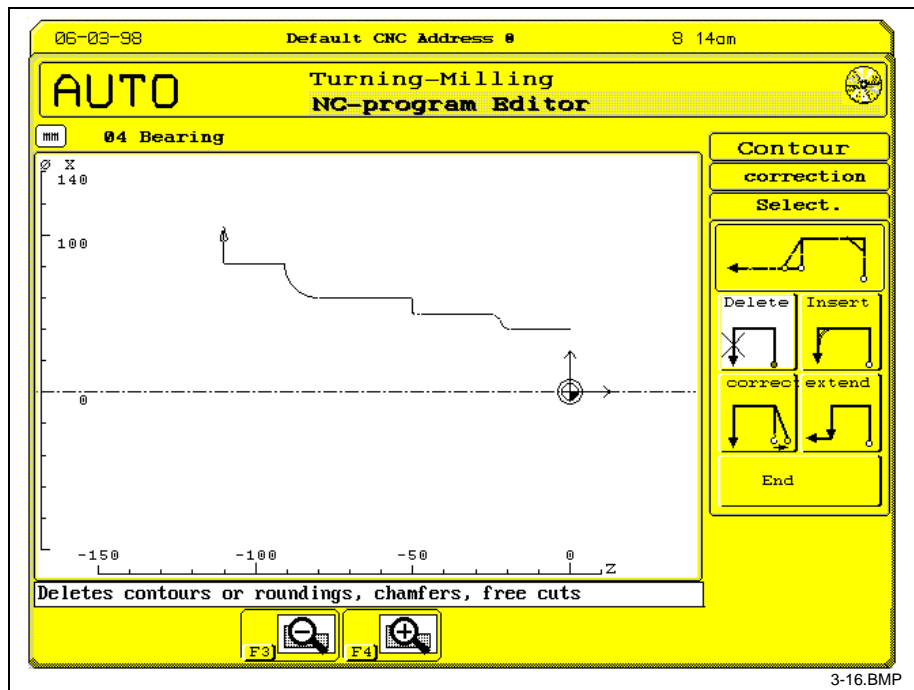


Fig. 3-16: Contour correction selection

Delete

Contour A contour is only deleted from a certain point to the contour end. The area to be deleted is selected via soft key <F6> or <F7> (see Fig. 3-17).

Corner roundings, chamfers, relief grooves

The transition element to be deleted is selected via soft keys <F6> or <F7>.

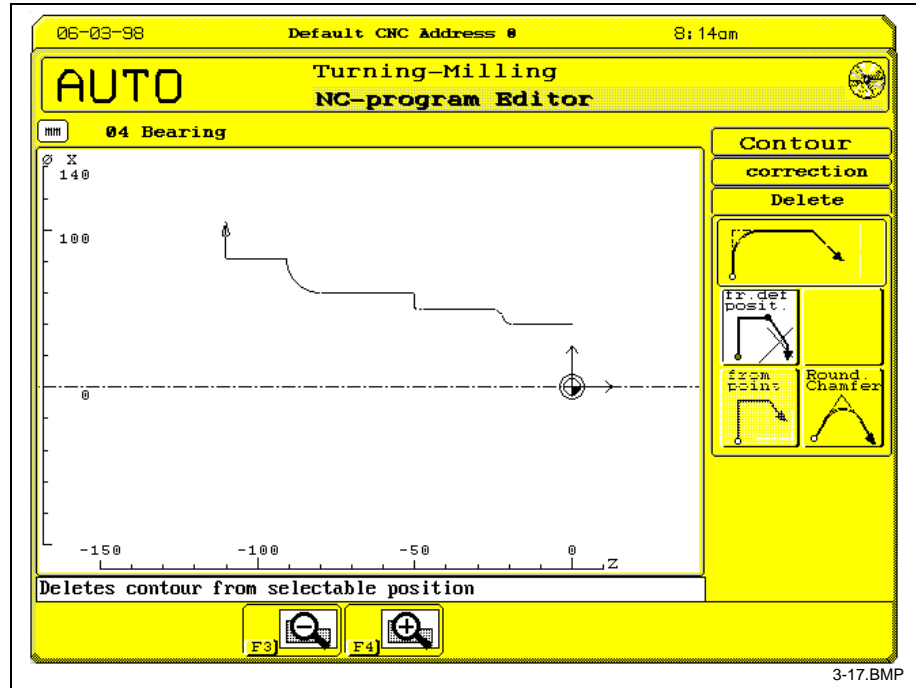


Fig. 3-17: Deleting contour elements

Insert

Using this function, transition elements (chamfers, corner roundings, relief grooves) can be inserted (see Fig. 3-18). The insert position is selected via soft keys <F6> or <F7>.

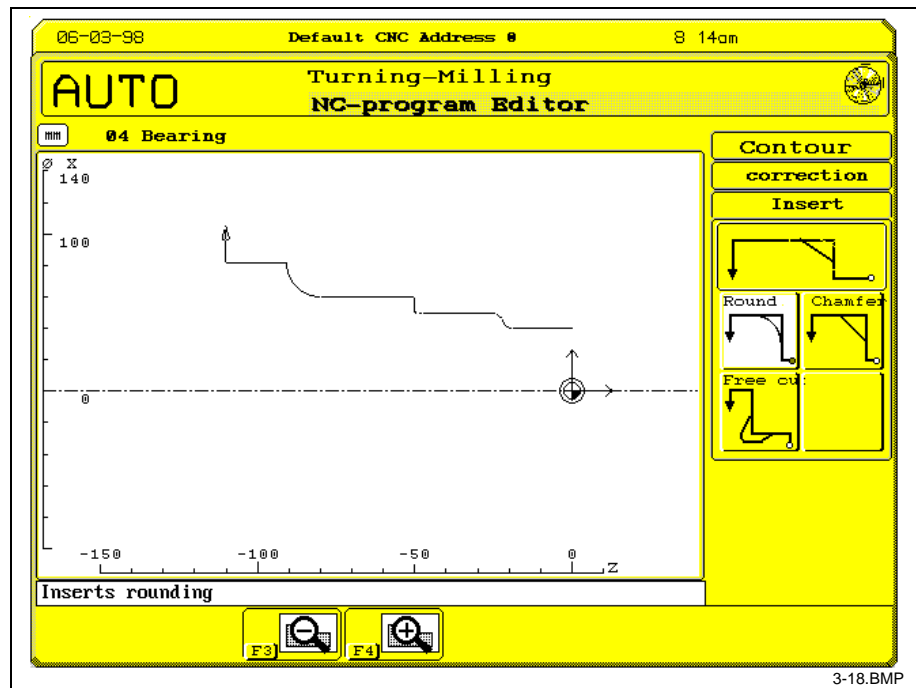


Fig. 3-18: Inserting contour elements

Correct

- Contour elements** The element to be corrected is selected via soft key <F6> or <F7>. The input data that has already been defined for the contour element is displayed next. The operation is terminated by pressing soft key <F7>.
- Contour start point** Correction of start point coordinate values. The function is terminated by pressing soft key <F7>.
- Transition elements** The transition element to be corrected (corner rounding, chamfer, relief groove) is selected via soft key <F6> or <F7> (see Fig. 3-19) . The operation is terminated by pressing soft key <F7>.

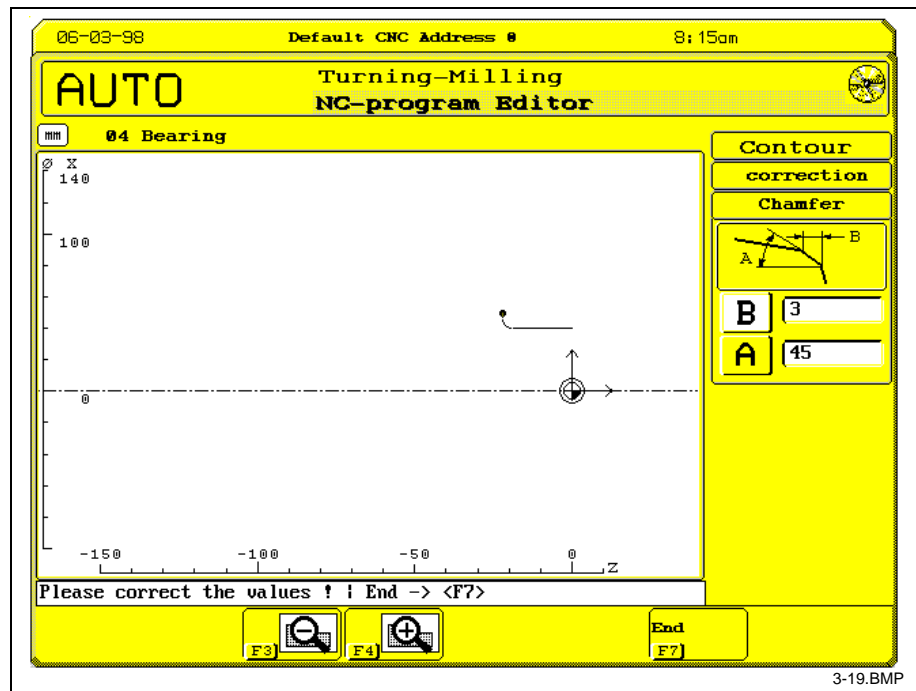


Fig. 3-19: Transition element correction

End Contour correction is terminated via the menu key „Terminate“.

Note: If the NC program already contains machining cycles, which refer back to the modified contour, the following warning is output: '**Check machining cycle of the corrected contour**'. It is recommended to check the relevant machining cycles which refer to the corrected contour (see 2.3 Modifying GNE).

Message	Cause / remedial action
Corner rounding too large	Corner rounding does not fit between the elements defined. ⇒ select a smaller radius
Check machining cycles of the modified contour	A cycle was defined within the NC program that relates to the modified contour. ⇒check whether cycle parameter are still correct although contour was modified.

Form elements

Function Form element definitions serve the complex description of a form element's geometrical form. The form definition itself does not initiate any machine movement. Only the subsequent machining cycle call performs actual cutting. A form element is handled like a defined contour.

Invocation Definition call is performed by selecting the relevant symbol from the GNE dialogue.

Programming

Turning form elements



Radial cut	FORM_20(Parameter list)
Axial cut	FORM_21(Parameter list)
Radial contour cut	FORM_22(Parameter list)
Axial contour cut	FORM_23(Parameter list)
thread	FORM_25(Parameter list)
Centered boring	FORM_30(Parameter list)

Form elements for milling



Slotted hole, straight	FORM_50(Parameter list)
Slotted hole, round	FORM_51(Parameter list)
Arc	FORM_52(Parameter list)
Polygon	FORM_53(Parameter list)
Text, straight	FORM_54(Parameter list)
Text, round	FORM_55(Parameter list)
Rectangle	FORM_56(Parameter list)
Rectangle, centered	FORM_57(Parameter list)
Polygon train	CONF(Parameter list)

Parameter list The first parameter determines the consecutive geometry (contour) number. With different milling form elements, the second parameter determines the geometry number of a drilling position. The further sequence mainly corresponds to the dialogue's menu sequence. The last parameter defines the axis group assignment.

Note: The parameters may only be registered or modified in the dialogue. The GNE geometry's entire number of elements defined in an NC program (arc, line, corner rounding, chamfer=1 element, relief groove=3 elements) is limited to 200 elements.

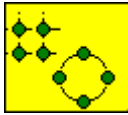
Drilling patterns

Function

This function is used for describing drilling patterns in a plane. The drilling pattern function does not initiate any movement. Machining proper is only implemented by the subsequent call of a GNE drilling cycle that related to the drilling pattern.

Invocation The definition is invoked by selecting the corresponding icon in the graphical NC editor.

Drilling patterns



Programming

Single points	PATERN_01 (Parameter list)
Radius angle	PATERN_02 (Parameter list)
Circle segment	PATERN_03 (Parameter list)
Matrix	PATERN_04 (Parameter list)
Line	PATERN_05 (Parameter list)

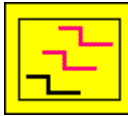
Parameter list The first parameter defines the consecutive number of the drilling pattern. The further sequence chiefly corresponds to the menu sequence of the dialog. The last parameter defines the axis group allocation.

Note: The parameters may only be registered or modified in the dialog.

Copy instructions

Function A copy instruction is used for copying geometry instructions (contours, form elements,...). They only define; they do not initiate any motion.

Invocation The definition is invoked by selecting the corresponding icon in the GNE dialog (see Fig.).



Programming

Mirroring	COPY_01(Parameter list)
Shifting	COPY_02(Parameter list)
Rotating	COPY_03(Parameter list)

Parameter list The first parameter determines the consecutive number of the result geometry (contour) number. With different milling form elements, the second parameter determines the geometry number of a drilling point. The further sequence mainly corresponds to the dialogue's menu sequence. The last parameter defines the axis group assignment.

Grouping The <F7> soft key defines whether or not the original is included in the new geometry. This is shown by the visual display.

Base selection The original geometry and the copied geometry are combined in a group „result geometry“.

Note: Only milling contours and milling form elements are copied. The parameters may only be registered or modified in the dialogue. The GNE geometry's entire number of elements defined in an NC program (arc, line, corner rounding, chamfer=1 element, relief groove=3 elements) is limited to 200 elements.

Example 1

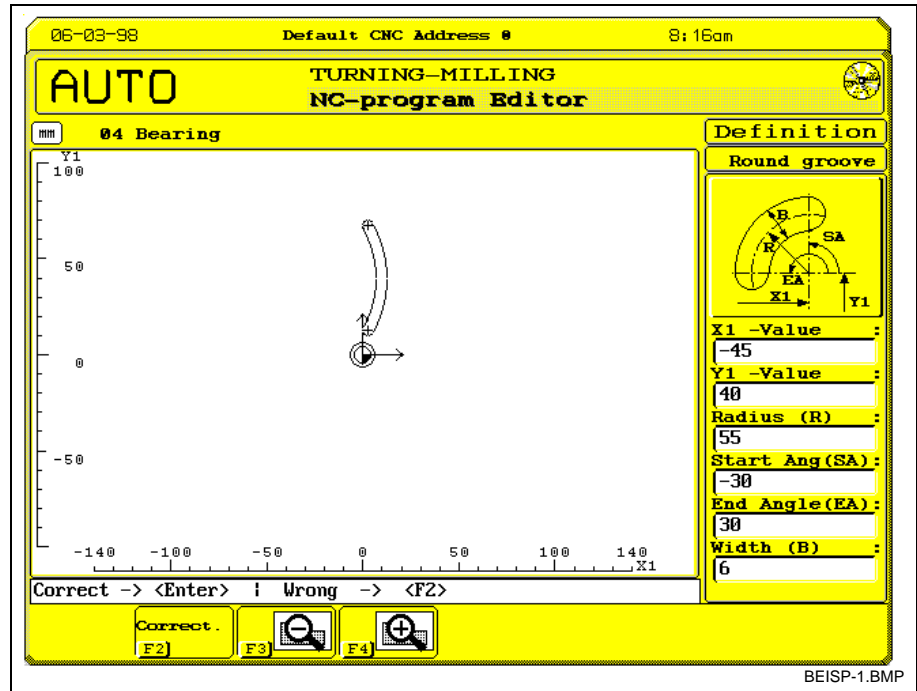


Fig. 3-20: Definition round groove

Example 2

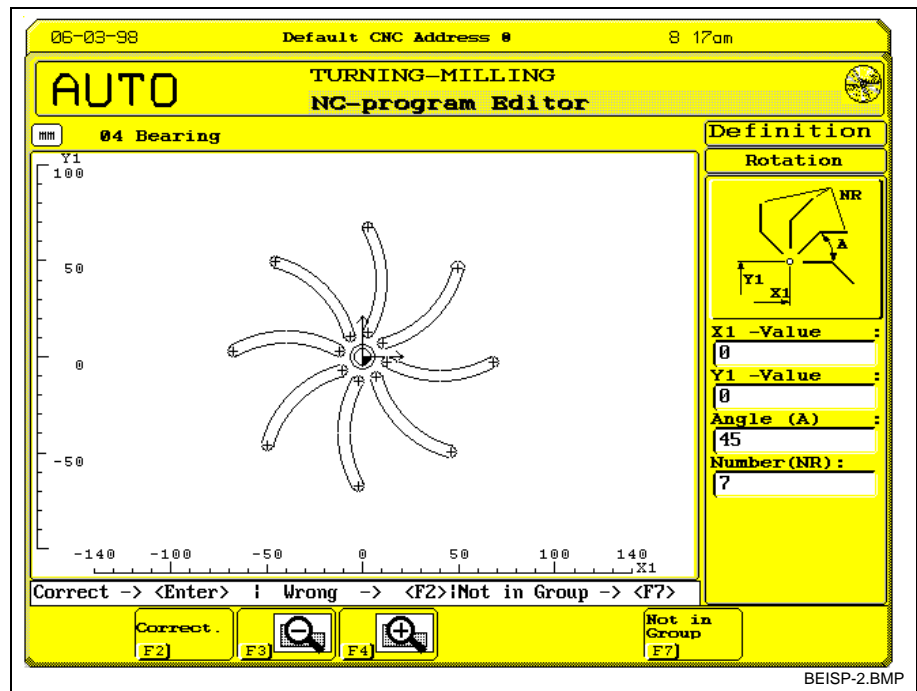


Fig. 3-21: Copying by rotation 7x45 degrees

3.3 Machining Cycles

General information

Function	Machining cycles are technology-related subroutines that perform a specific machining process, for example contour cutting, contour milling, boring, etc. The concrete machining sequence results from the cycle's parameter values.
Invocation	The cycle call is performed by selecting the relevant symbol from the GNE dialogue. Starting from the previously defined contour, the user performs programming using the dialogue.
Prerequisites	Prior to a cycle call, the tool and the relevant spindle speed, cutting feed, turning direction and if necessary the spindle gear must be programmed in the NC program. If a spindle that is different to the first one is used, defining the reference spindle may become necessary (see MT-CNC, Folder 1, Register 2 NC Programming Instructions, Drawing no. 109-0668-4183-06, Item. 4.5.2 - SPF command, page 4-49)
Start position	A safe start position within the plane must be programmed, from where the tool can start the cycle without collision. The axes always move simultaneously to the cycle start point.
<hr/>	
	Note: To achieve a high accuracy in thread cutting, the interpolation conditions 'G6' and 'G8' should be set before the cycle is invoked.
<hr/>	
Return conditions	Usually, the conditions that were valid before the cycle invocation exist at the end of the cycle. These are, in particular, G codes of the groups 5, 6 and 7, and 4 and 13 for transformation, edge no. upon contact.
Output variants	The results are provided in two variants when the GNE is exited and, consequently, the machining sequence is accepted in the current NC program (see Fig. 3-22): <ul style="list-style-type: none"> • Output as a cycle call with parameter list, or • output as a sequence of NC blocks

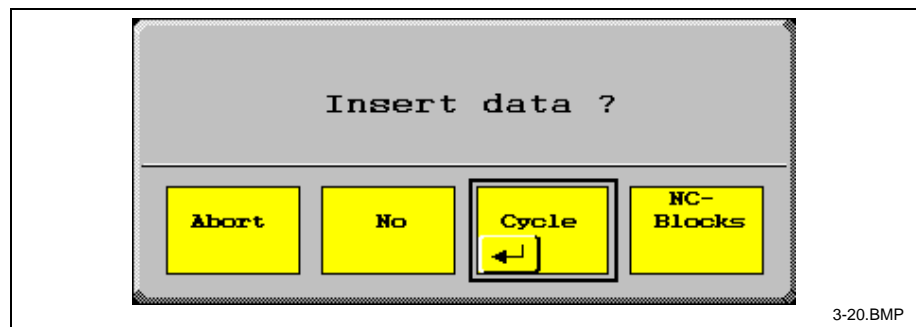


Fig. 3-22: Output as cycle selected

Cycle call	In an output as a cycle call with a parameter list, the actual NC blocks are generated later via the compiler and inserted into the internal NC program at this position.
-------------------	---

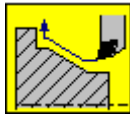
NC blocks	The output blocks are generated immediately and inserted into the NC program and not in the cycle call. If necessary, modifications are possible. All instructions (for example WINDOW...,CONTUR...) defined since the last GNE call, are not accepted into the NC program. To accept these for correction or further processing, the GNE must first be terminated with „CYCLE“ (this saves the instructions in the NC program), before it is called up again at the „CYCLE...“ position.
Modifications	The cycle should only be modified in the GNE dialogue. Transformation from 'cycle output' into 'NC block output' is possible at any time. This means that NC block output is also possible at the end of the programming process. This has the advantage that all definitions are retained in the parts program.
Variables	Furthermore care must be taken, that no variables are assigned to 'CYCLE_' parameter.

Note: Due to shortage of space it is not possible to display all parameter combinations including their theoretically possible effects. For that reason, only common facts are described in the parameter or cycle description below.
Due to the extremely high number of combination possibilities between geometry and machining variation, the user must check the GNE results and debug the program carefully.

Cycle overview

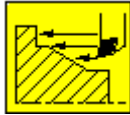
Turning cycles	Milling cycles	Drilling cycles
Roughing	Contour milling	Boring - centering
Contour cut	Pocket milling	Deep hole drilling
Residual cut	Angle milling	Thread cutting
Grooving		Reaming
Thread machining		Boring
Centric boring - drilling - reaming - thread cutting		Backboring
		Step boring

Turning Cycles: Cutting, Contouring, Residual Cutting, Grooving, Thread Cutting, Centric Boring



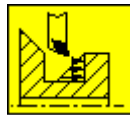
Contour cut

Using a *contouring cycle* (CYCLE_10), a previously defined contour can be machined in a single cut.



Cutting

Using a *cutting cycle* (CYCLE_11), a previously defined contour can be machined in several steps and in different variants.



Residual cutting

Using the *residual cutting cycle* (CYCLE_12), a contour, which could not be machined completely due to the tool's geometry, can be machined.



Form element

Using the *form element cycle* (CYCLE_13), (CYCLE_31,...) and (CYCLE_25), thread cutting operations can be performed at contours - grooving operations at center borings and/or threads.

The call of the various cycles takes place in the GNE main menu (see Fig. 3-23).

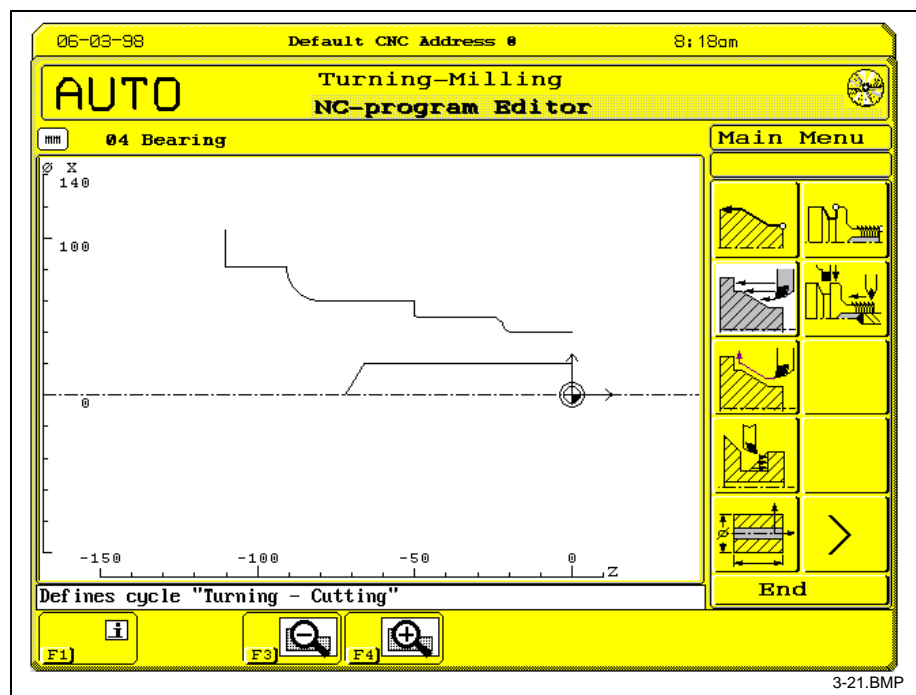


Fig. 3-23: GNE main menu: Cutting selection

Programming

Contour cut	CYCLE_10 (Parameter list)
Cutting	CYCLE_11 (Parameter list)
Residual cutting	CYCLE_12 (Parameter list)
Form element - grooving operation	CYCLE_13 (Parameter list)
Form element - thread	CYCLE_25 (Parameter list)
Form element - center boring	CYCLE_31 (Parameter list)
Form element - center reaming	CYCLE_32 (Parameter list)
Form element - center thread cutting	CYCLE_35 (Parameter list)

Parameter list

The parameter description sequence basically corresponds to the dialogue's menu sequence (see Fig. 3-24). All parameter are separated by commas. The parameter significance results from the location within the parameter list. The parameter may only be input or modified in the dialogue. The first parameter defines the number of the geometry definition that is to be processed.

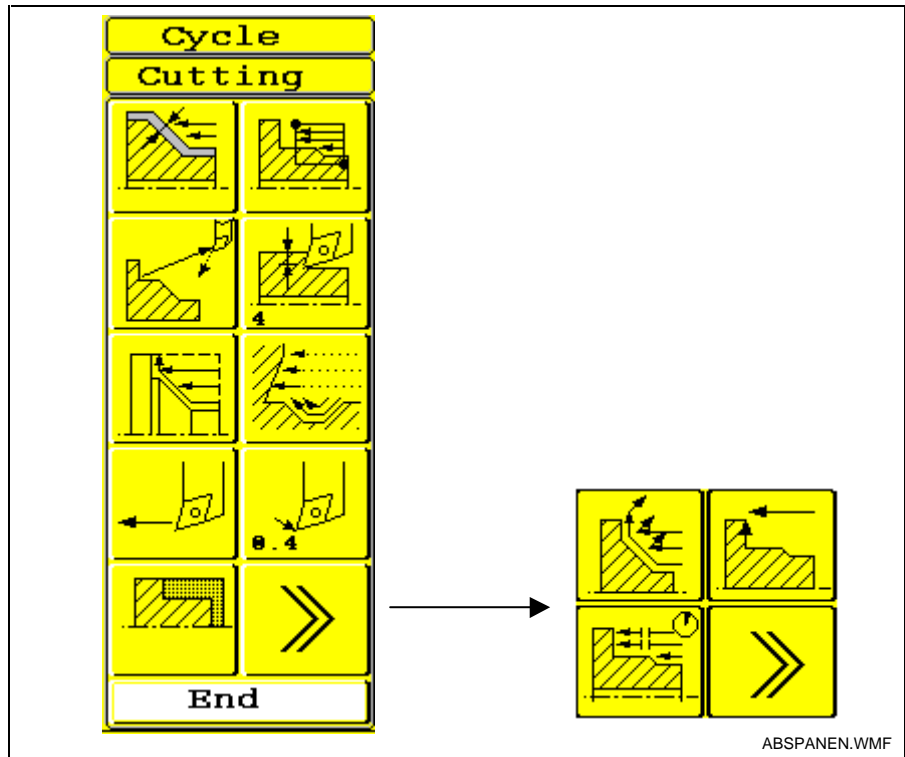
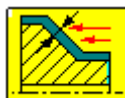


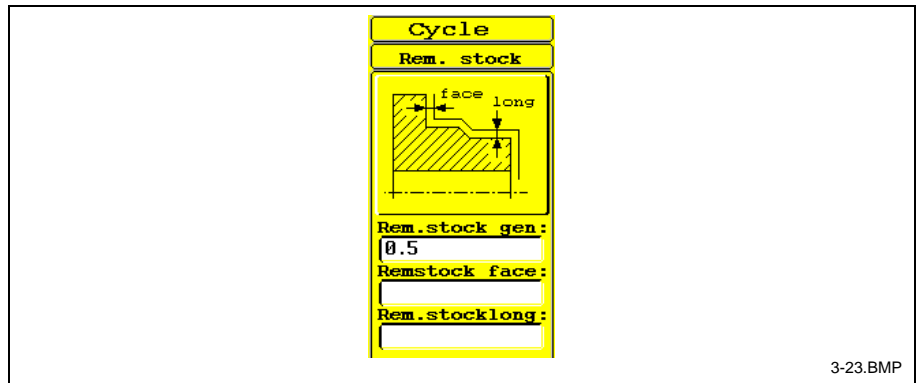
Fig. 3-24: Parameter overview: Example cutting

Remaining stock



Up to three different kinds of remaining stock can be programmed (see Fig. 3-25)

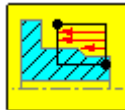
- Remaining stock, general; valid for the entire contour, if 'remaining stock, face' and 'remaining stock, longitudinal' are not programmed.
- remaining stock, face; valid only for all face axes
- remaining stock, longitudinal; valid only for all longitudinal axes



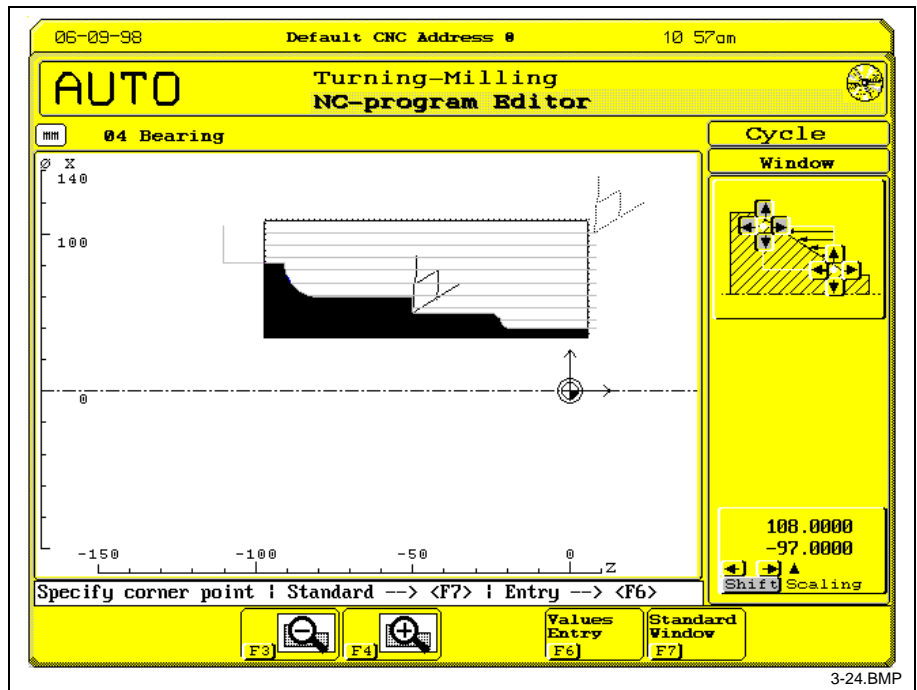
3-23.BMP

Fig. 3-25: Remaining stock parameters

Machining window



Using the machining window, the machining range, with respect to the defined contour, can be limited or extended (see Fig. 3-26). Without window definition (standard window F7), the window corners and contour start point or contour end point are identical. In case of residual cutting, a machining window must be defined if several residual cutting zones exist.



3-24.BMP

Fig. 3-26: Machining window

Tool retract At the end of machining, there are four tool retraction variants available (see Fig. 3-27).

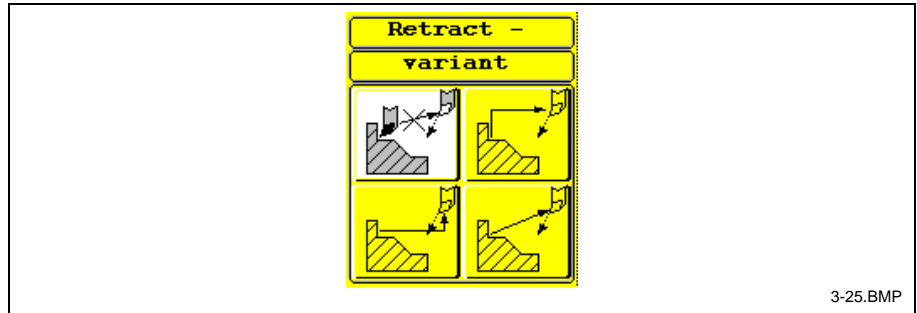
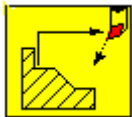


Fig. 3-27: Tool retract variants



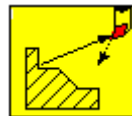
- Tool does *not* move to the start position



- Tool moves to the start position - first face axis, then longitudinal axis



- Tool moves to the start position - first longitudinal axis, then face axis



- Tool moves to the start position - longitudinal and face axis simultaneously

The 'cutting depth' parameter determines the maximum in-feed per rough cut.

Cutting depth

This parameter is invalid for the contouring cycle.

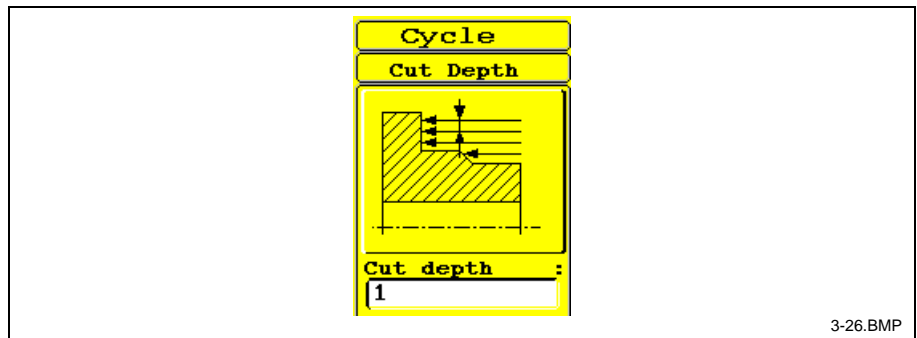
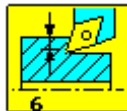
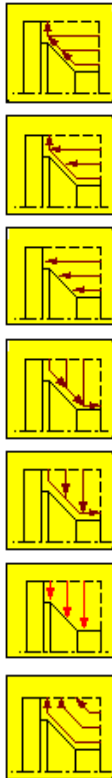


Fig. 3-28: Cutting depth

Cutting variants

There are seven cutting variants available (see Fig. 3-29).



- axes parallel, longitudinal, following the contour in each cut
- axes parallel - longitudinal, contour cutting at the end
- axes parallel - longitudinal
- axes parallel - face, following the contour in each cut
- axes parallel - face, contour cutting at the end
- axes parallel - face
- contour parallel

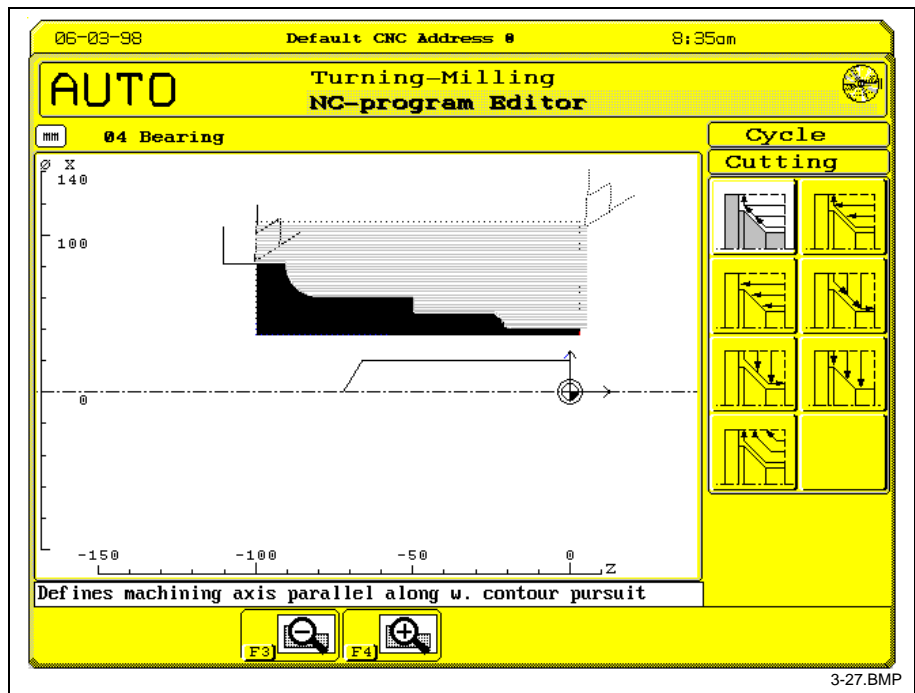
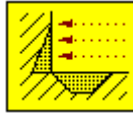


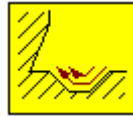
Fig. 3-29: Cutting variants

Undercut variants

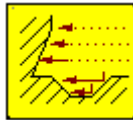
If there are undercut areas within the contour that is to be machined or within the machining window, it can be selected whether



- they are not machined,

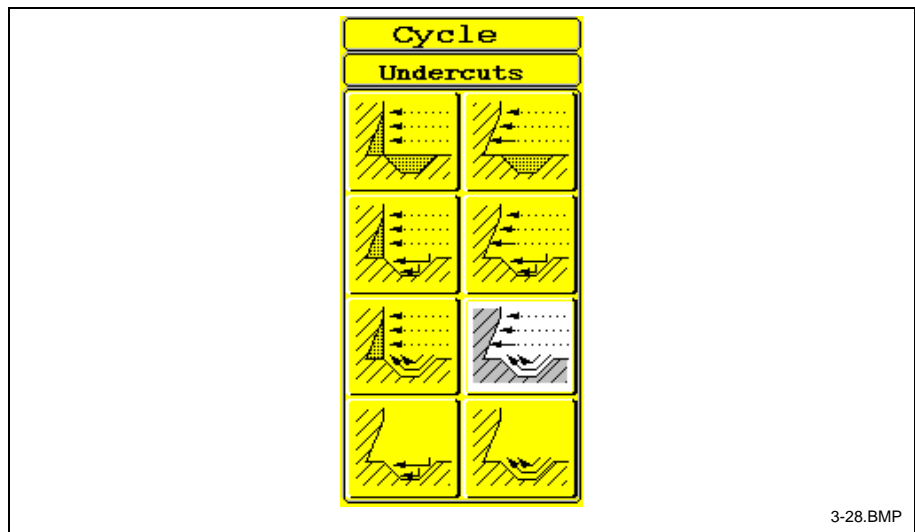


- they are machined, or



- only the undercut areas are machined (see Fig. 3-30).

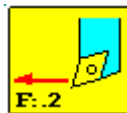
Undercutting can be performed axis parallel - longitudinal or contour parallel.



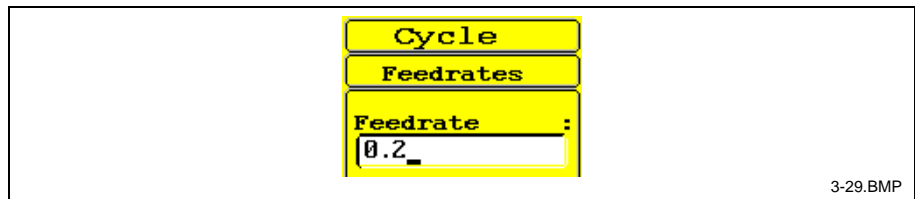
3-28.BMP

Fig. 3-30: Undercut variants

Feed rate



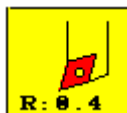
- Machining feed rate definition



3-29.BMP

Fig. 3-31: Machining feed rate

Machining tool



Here, the cycle is informed about the machining tool's tool edge geometry. The tool edge geometry must correspond to the tool actually used (see Fig. 3-33). The tool edge data consists of:

- tool edge radius
- corner angle
- clearance angle

The NC program always tries to machine the contour using the tool specified. A warning is output if contour cannot be machined completely due to the tool data. The contour can then be completed with another tool (see 'residual cutting cycle').

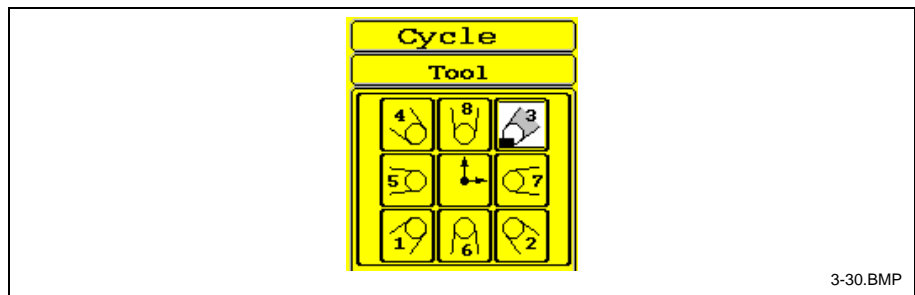


Fig. 3-32: Tool data: Tool edge orientation

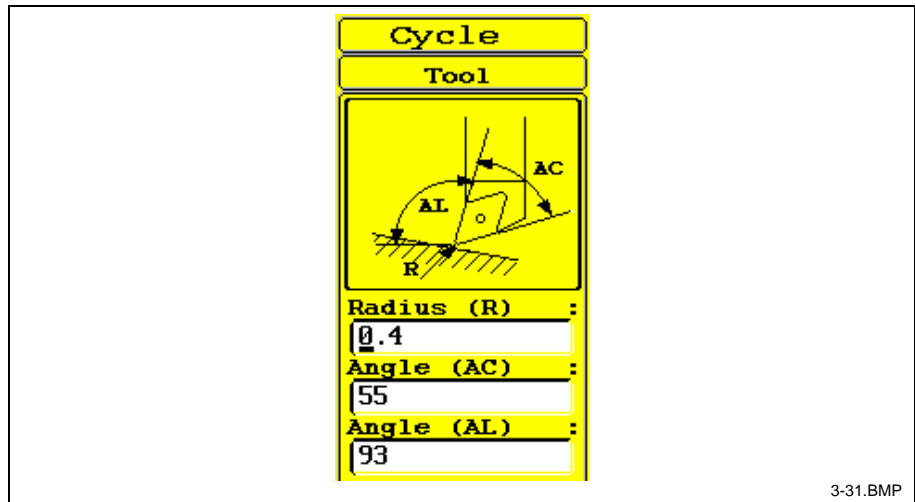


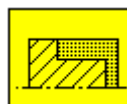
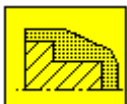
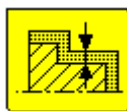
Fig. 3-33: Tool parameters

Note: The GNE results are always determined from the tool geometry data that is specified in the cycle. The machine stops if there is a discrepancy between the actual tool data and the tool data defined in the cycle. An additional message may be issued via the SPS (see 4. Data Adaptation of Technical Parameters).

Roughing tool



Initial form

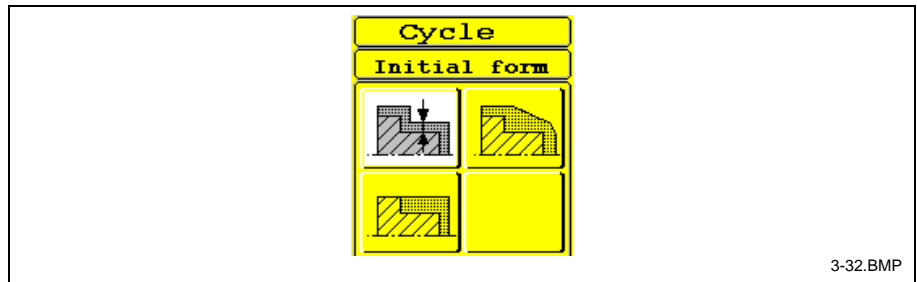


Here, the residual cutting cycle is informed of the roughing tool's tool edge geometry data. It consists of:

- roughing tool corner angle
- roughing tool clearance angle

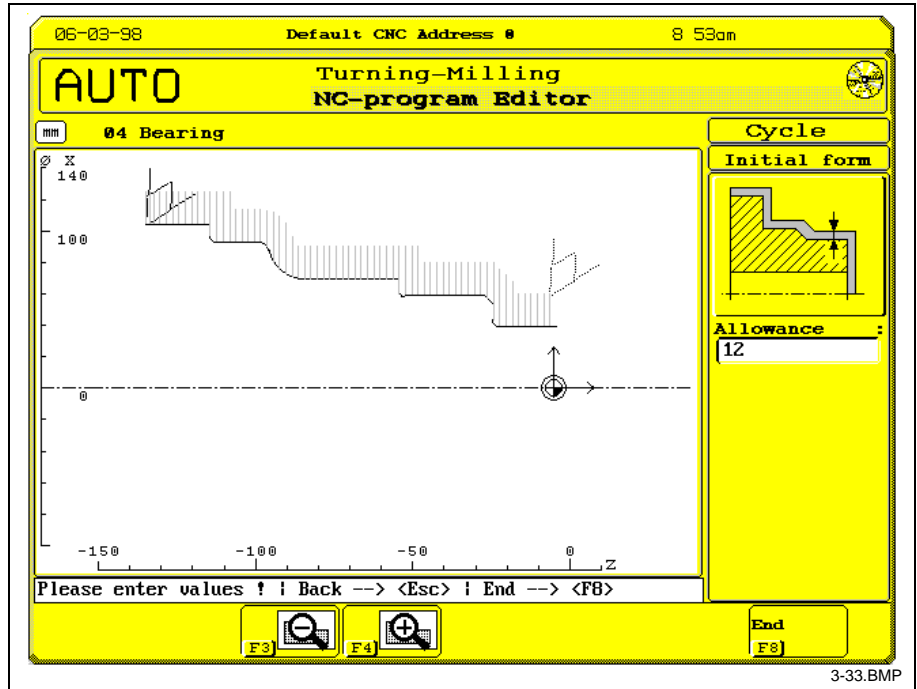
There are three different variants available for initial form definition (see Fig. 3-34) .

- Allowance - contour parallel, for example a blank with regular allowance (see Fig. 3-35)
- Allowance is already described by the contour or presented to be described (see Fig. 3-36 and Fig. 3-37)
- Allowance - cylinder



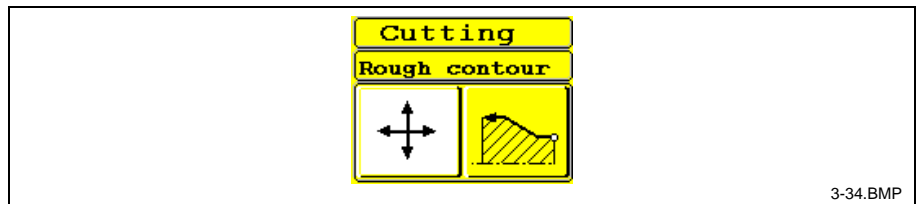
3-32.BMP

Fig. 3-34: Initial form selection



3-33.BMP

Fig. 3-35: Allowance - contour parallel



3-34.BMP

Fig. 3-36: Blank contour selection or new definition

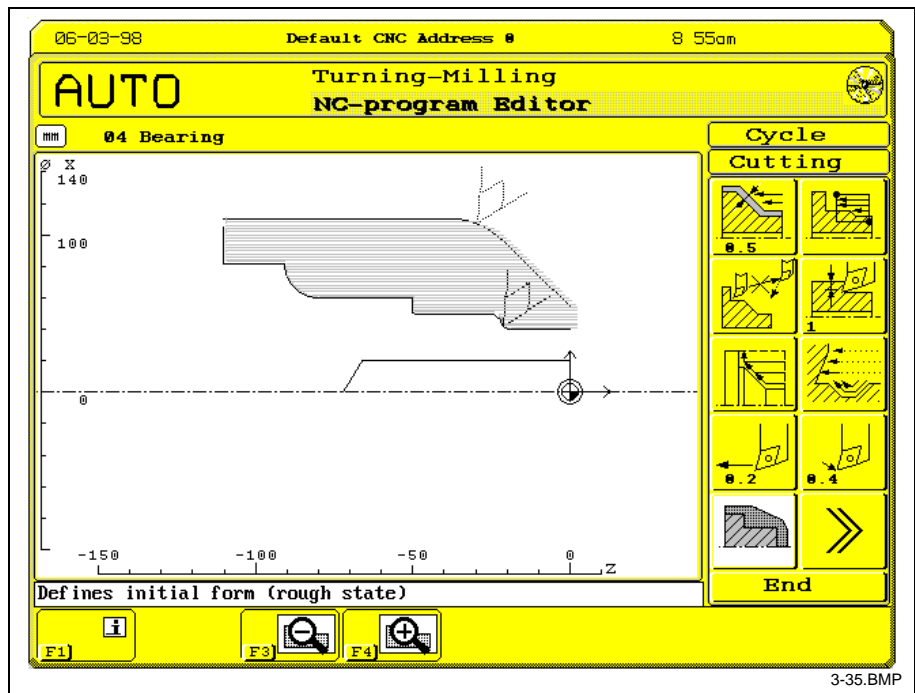


Fig. 3-37: Allowance - described via rough contour

Peculiarities

Note: Internally, the programmed blank contour is projected onto the turning axis at the beginning and at the end. This „section“ includes the „area“ that is to be machined. Thus, a „cylinder“ element must be programmed at the „material side“ when a blank contour is defined at the face side.

Retract variants

There are four different retract variants available (see Fig. 3-38)



- Retracting at the end of cut and cycle end (standard).
- retracting at the end of cut without retracting at the end of the cycle.
- without retracting at the end of cut, but retracting at the end of the cycle.
- without retracting at the end of cut and at the end of the cycle.

The retract amount can be set (see 4 Data Adaptation)

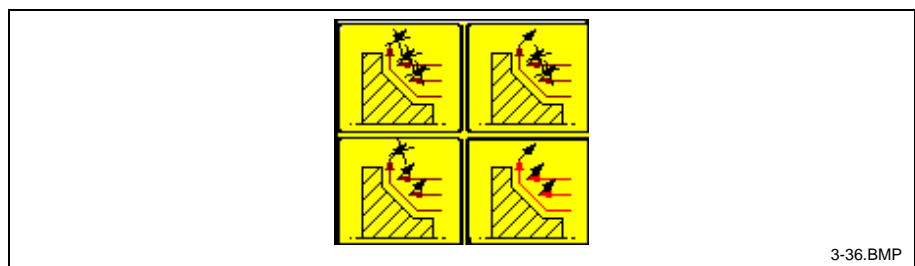
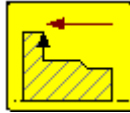


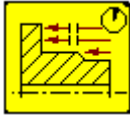
Fig. 3-38: Retract variant

Machining direction



The machining direction in relation to the direction of the contour definition can be changed using this function.

Chip breakage



The parameter 'distance chip breakage' and 'dwell time' determine the distance and period of time for chip breakage purposes. (see Fig. 3-39)

Cycle

Chip breakage

Dist.chip brk:

Dwell time :

3-37.BMP

Single element

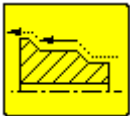


Fig. 3-39: Chip breakage parameter

A separate allowance and feed rate can be assigned to a contour element within a contour cutting cycle.

The allowance defined for the entire contour is ignored in this case.

Approach and run-out



In addition to the machining window, the tool's start and end point can be affected using 'approach' and 'run-out'. The parameter's mode of action is shown in Fig. 3-40 and Fig. 3-41.

Cycle

Approach

Approach (SDT) :

Approach (SDN) :

Run-out (RO) :

3-38.BMP

Fig. 3-40: Tangential approach

Cycle

Run-Out

Approach (SDT) :

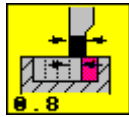
Approach (SDN) :

Run-out (RO) :

3-39.BMP

Fig. 3-41: Tangential run-out

Coverage

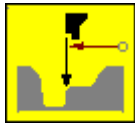


Auxiliary function - plunge cutting

is the ratio of chip width and tool edge width in plunge cutting operations.

Start point

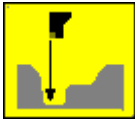
defines the start point coordinate for plunge cutting operations



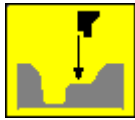
- Start with programmable start coordinate



- Start at the contour start



- Start at the deepest bottom



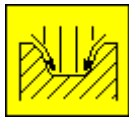
- Start at the first axis-parallel bottom

Roughing variant

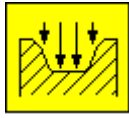
There are four variants available:



- axis-parallel with contour tracking per cut



- axis-parallel with contour cut at the end

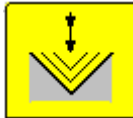


- axis-parallel



- contour cut

Infeed variants



- Thread cutting infeed „axis-parallel“

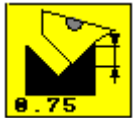


- Thread cutting infeed „1 edge“

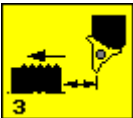


- Thread cutting infeed „2 edges“

Infeed depth



- Defines the first infeed increment and the number of blank steps in thread cutting



- Determines approach and run-out in thread cutting

Tool

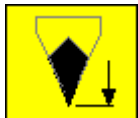


- Thread cutting with overhead tool



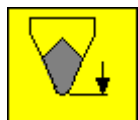
- Thread cutting in normal position

Tool setting point



Here it is shown whether the setting point of the thread cutting tool refers to the theoretical tool tip or to the rounding. This is important for the calculation of the thread depth. The selection is made within data adjustment - „technical parameters 2“.

- Thread cutting tool: Setting point at tool tip



- Thread cutting tool: Setting point at rounding

Depth infeed variant



- The depth infeed follows the relationship:
 $C, C\sqrt{2}, \dots C\sqrt{n}$

Legend: C = 1st infeed depth



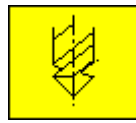
- The depth infeed follows the relationship
 $C, 2C, 2C\sqrt{2}, \dots 2C\sqrt{n}$

Legend: C = 1st infeed depth

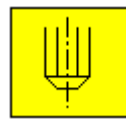
Note: In order to achieve a high thread cutting accuracy, the interpolation conditions 'G6' and 'G8' should be set before the cycle is invoked.

Auxiliary function - center boring

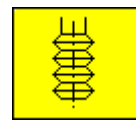
Boring



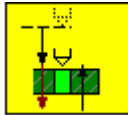
Reaming



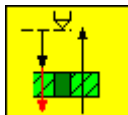
Thread cutting



Tool retracting

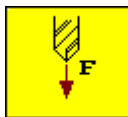


- Tool does not move back to the approach level.



- Tool moves back to the approach level.

Feed rate



The feed rate is entered as mm/inch or mm/revolution.
- selectable via <F7> key.

Drilling

Feedrate

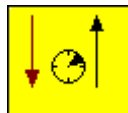
Feed/Revolution

Feedrate :

.33

VORSCHUB.BMP

Dwell



- Defines the dwell (in seconds) at the bore bottom.

Drilling

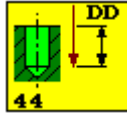
Dwell Time

[s]

Dwell time :

.3

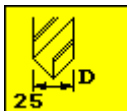
VERWEIL.BMP

Bore depth

- The infeed movements are performed on the basis of the parameters level, bore depth, safety distance and return distance. Level is always an absolute dimension. The other data items are relative to level.

Deep-drilling	
Depth	
Level (LV) :	<input type="text" value="0"/>
DrillDepth DD:	<input type="text" value="44"/>
S.Distance SD:	<input type="text" value="2"/>
R. niveau RL:	<input type="text" value="2"/>

BOHREN8.BMP

Tool

- Specification of the tool diameter and of other relevant tool data.

Sequence - drilling cycles

- At the beginning of the first cut, the current turning tool is positioned simultaneously in both axes at rapid speed from the previously programmed start position and the previously programmed speed.
- All steps are executed according to the cycle parameter.
- Active cuts are performed at the specified feed rate; lift-off and retract movements at rapid traverse speed.

At the end of the cycle, the tool either moves back to the start position or stops at the end of the last cut, depending on the return variant.

Message	Cause / remedial action
Contour not processed completely.	Tool shape does not tolerate a complete machining. ⇒ Change tool parameters or rework contour
Circle radius is too small for the tool	⇒ Change tool parameters or rework contour
Illegal change of the machining allowance	Combination of contour and machining allowances cannot be specified ⇒ Modify machining allowance
Modified tool radius	Discrepancy between tool radius of NC program cycle and tools list ⇒ Change the parameters in the NC program
Contour outside of initial form or window	There is no material for machining ⇒ either incorrect machining window or wrong initial contour
Contour was modified	Confirm cycle in GNE dialog analogous to „Modification“

Cutting cycle example

Using face cutting, a contour shall be roughed with 0.5 mm finishing allowance and 1 mm cutting depth. The machining allowance is limited by a contour. The feed rate is 0.21 m/rev. At the end of the cycle, the tool shall move back to the start point, simultaneously with both axes (see Fig. 3-42)

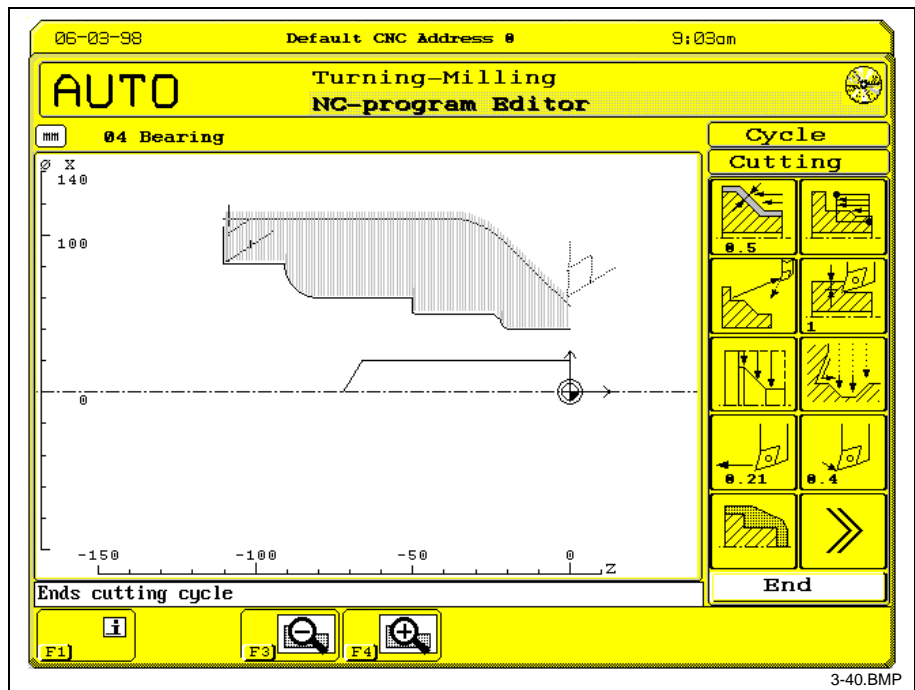


Fig. 3-42: Cutting cycle example

Milling cycles

Function The contouring cycle can be used for machining a previously defined contour.

Invocation Cycle call takes place within the GNE main menu (see Fig. 3-43) .

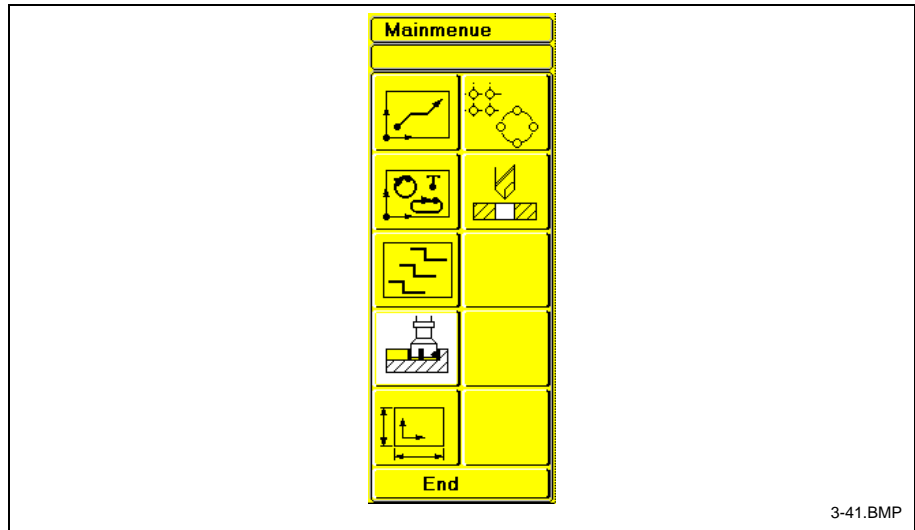


Fig. 3-43: GNE main menu - milling opened

Contour milling



Pocket milling



Angle milling



Programming

Contour milling	CYCLE_40 (Parameter list)
Pocket milling	CYCLE_41 (Parameter list)
Angle milling	CYCLE_42 (Parameter list)

Parameter list

The parameter description sequence basically corresponds to the dialogue's menu sequence (see 3-44). All parameters are separated by commas. The parameter significance results from the location within the parameter list. The parameter should only be input or modified in the dialogue.

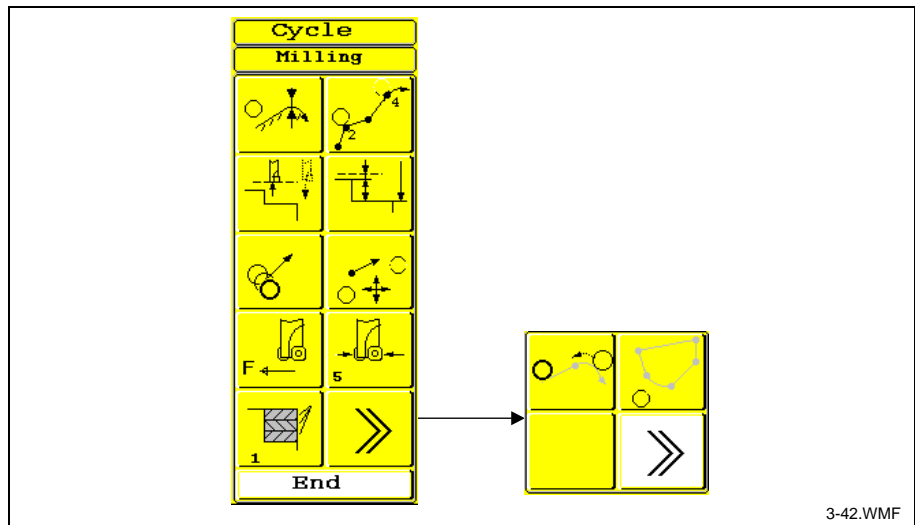


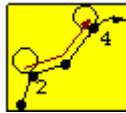
Fig. 3-44: Parameter example contouring

Allowance



Allowance of the entire contour in the respective plane.

Partial contour



Using the 'point-to-point' definition, there is a possibility of machining only parts of the defined contour. The contour points are numbered optically. The partial contour is defined by selecting the first and the last contour point via the cursor. If the entire contour is to be machined again, soft key <F7> must be pressed. Example (see Fig. 3-45).

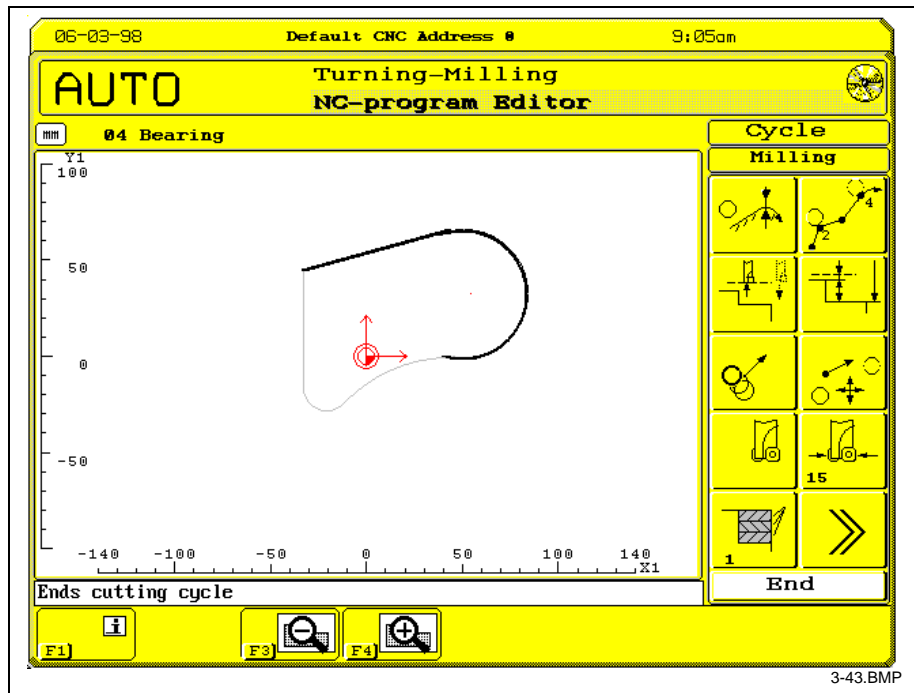


Fig. 3-45: Example: Partial Contour 2 through 4

Tool retract

At the end of machining, there are two tool retract variants available (see Fig. 3-46)

- tool does not move to the start position
- tool only lifts off to the start position in the infeed axis

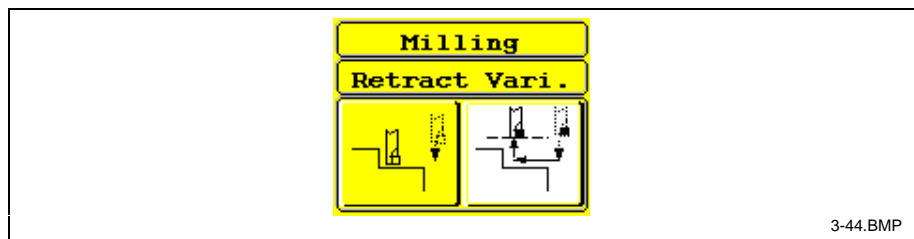
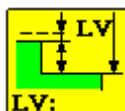
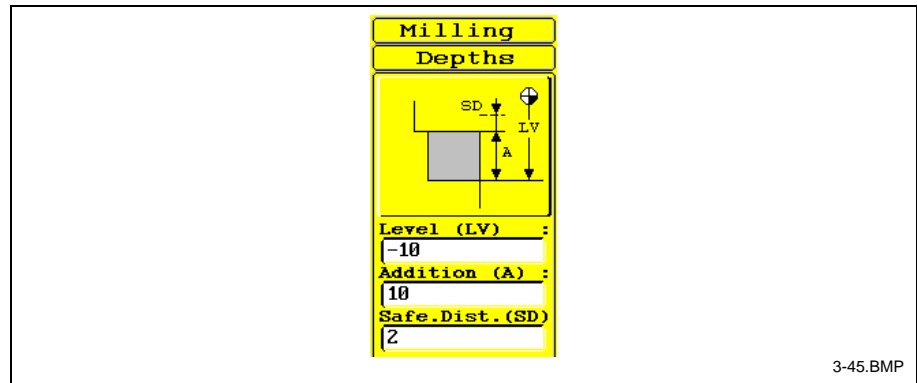


Fig. 3-46: Tool retract variants

Plunge



Plunge movements at machining level correspond to parameter 'level', 'addition' and 'safety distance' (see Fig. 3-47). 'Level' defines the milling bottom in absolute dimensions, and 'addition' the total height to be machined. The 'safety distance' is required for depth in-feed and when machining in several cuts is performed. It indicates the distance between the tool and the material on the plane.



3-45.BMP

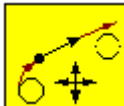
Fig. 3-47: Plunge

Cutter compensation

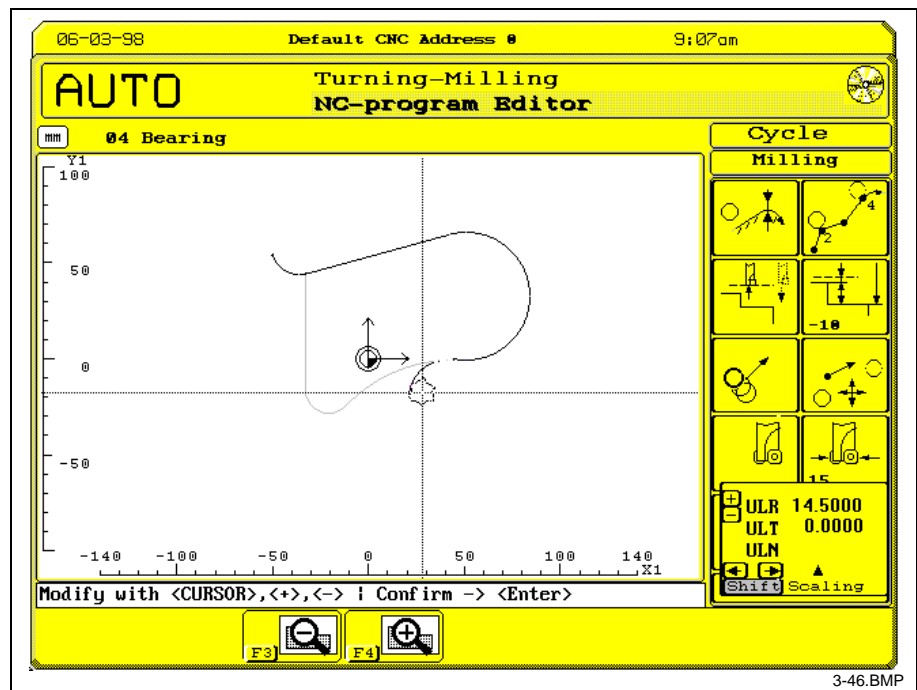


The tool orientation with relation to the defined contour can be defined (analogous to G40/G41/G42).

Approach and run-out



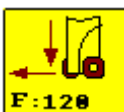
Using this parameter, the 'approach' and 'run-out' in relation to the programmed contour can be modified. The tangential and perpendicular components of the 'approach' and 'run-out' are determined via cursor. The '+' and '-' keys determine the 'approach' and 'run-out' arc. It is possible to combine all variants. The parameter's numerical value is displayed (see Fig. 3-48) .



3-46.BMP

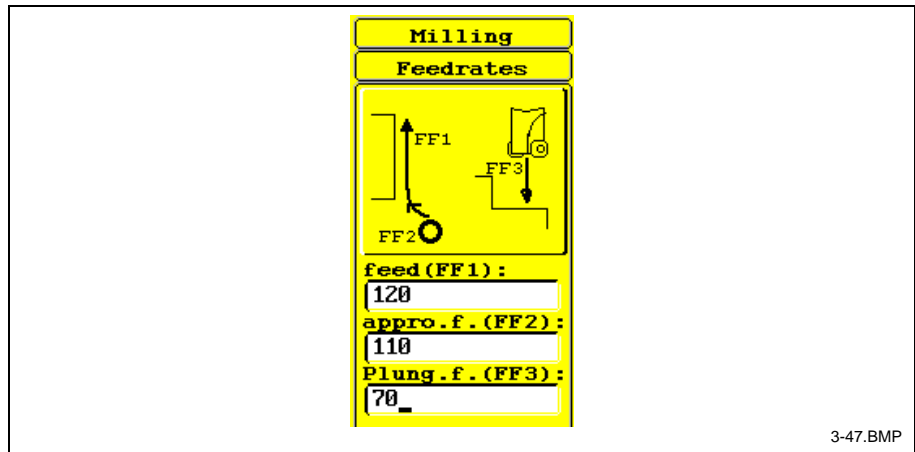
Fig. 3-48: Example: '14.5 mm run-out radius to an open contour

Feed rates



The machining feed rate is subdivided into the following components (see Fig. 3-49)

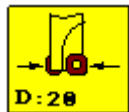
- Feed rate - valid for the entire contour
- Approach feed - valid for additional approach movement
- Plunge feed rate - valid for in-feed movement. Press <F7> to apply rapid traverse for plunge movement.



3-47.BMP

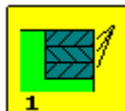
Fig. 3-49: Feed rate components

Tool diameter

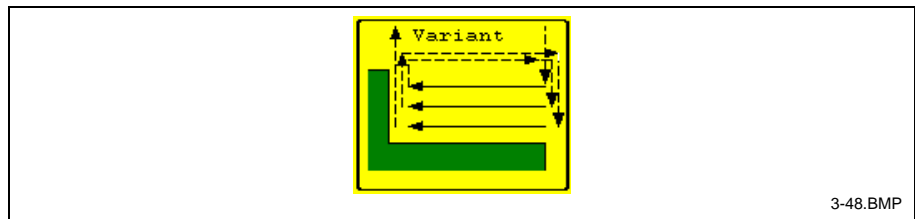


Tool diameter definition

Number of cuts

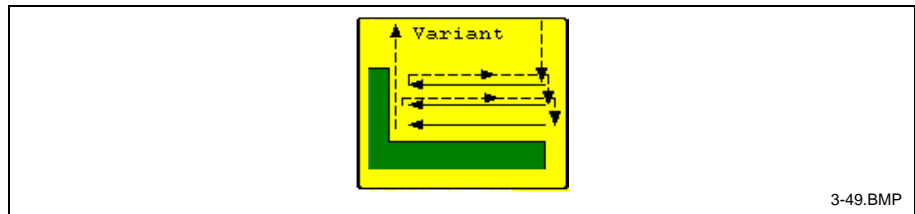


Standard cut segmentation (if no. of cuts >1) is performed depending on the allowance defined (see 'allowance'). The three different in-feed variants are selected via soft key <F7> (see Fig. 3-50 and Fig. 3-51).



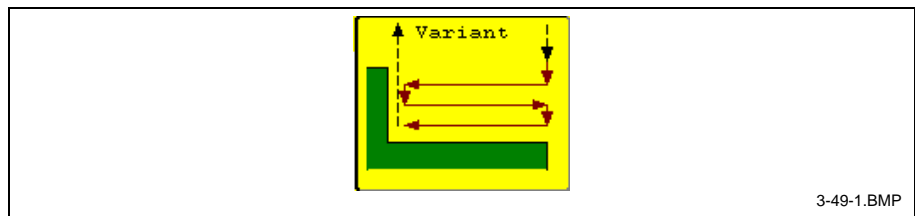
3-48.BMP

Fig. 3-50: Infeed variant „Lift off to 'Allowance' level



3-49.BMP

Fig. 3-51: Infeed variant „Lift off by safety distance'



3-49-1.BMP

Fig. 3-52: Infeed variant without lift-off - new approach

Machining direction



Setting the machining direction and changing the tool path correction at the same time.

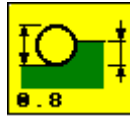
- in the contour definition direction
- opposite to the contour definition direction

Start point



With closed contours, any start point can be chosen.

Contact ratio



Auxiliary function - pocket milling

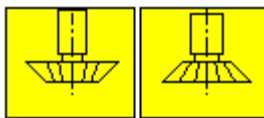
- defines the ratio of milling width and tool diameter.

Start point



- Here, one or more „lowering“ positions or an approach or run-out can be defined.

Tool



Auxiliary function - Angle milling

- Type selection with subsequent parameter input of the major tool data.

Machining depth



The level value is entered as an absolute dimension. The diameter is the tool diameter which is in contact with the workpiece. This diameter must agree with the radius value in the tools list.

Sequence - milling cycles

- Simultaneous positioning of the current tool at rapid traverse from the previously programmed start position and at the previously programmed speed at the beginning of the cut.
- Moving to cut depth in the programmed infeed rate. Moving along the approach path at the approach feed rate. Moving along the contour and the run-out at tool path feed rate.
- Number of cuts >1: Retracting according to in-feed variant. Either absolutely to 'level + addition + safety distance' or relatively to the 'safety distance'; or infeed at the end of the cut at feed rate (hunting).
- At the cycle end, the tool either lifts off to the start position (infeed axis) or is stopped at the end of the last cut (depends on return variant).

Message	Cause / remedial action
Arc radius too small for the tool or contour bottle-neck	Tool diameter too small ⇒ change tool diameter or rework contour

Contouring example

Outer contour roughing in a single cut using a 20-mm shank-type cutter, 0.3 mm allowance. The allowance across the entire contour amounts to 12 mm. The feed rate amounts to 120 mm/min. The approach is performed via approach arc. At the end, the contour is left using a tangential run-out. Then, the tool should retract to the start position (see Fig. 3-53) .

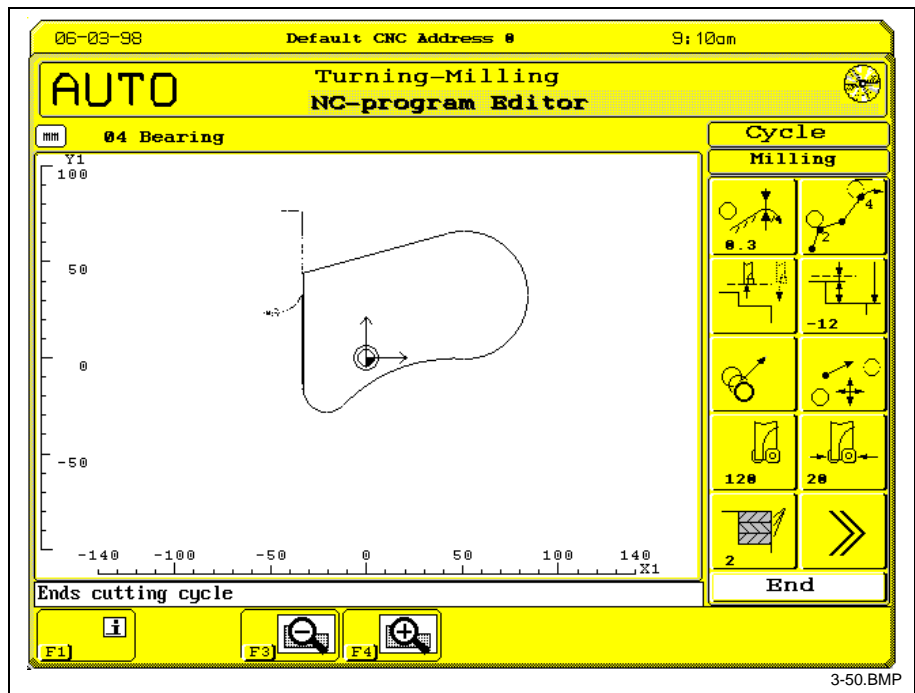
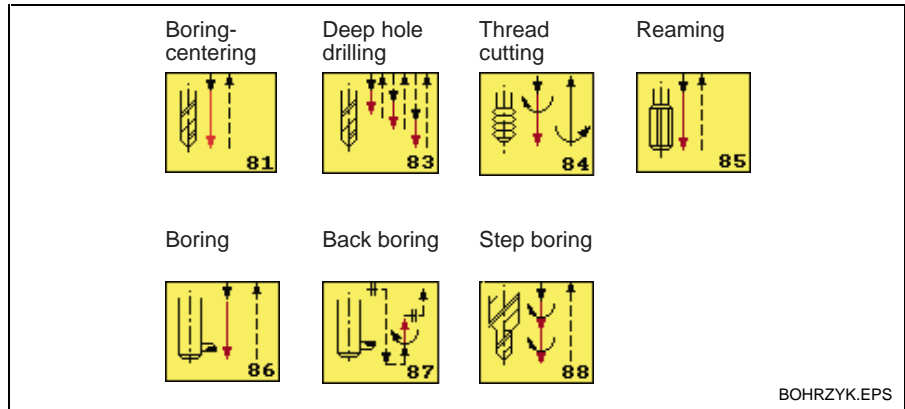
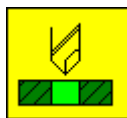


Fig. 3-53: Example: Contour cut milling

Drilling cycles

Function Drilling cycles can be used for machining previously defined hole patterns.

Invocation The functions are invoked in the corresponding GNE menu



BOHRZYK.EPS

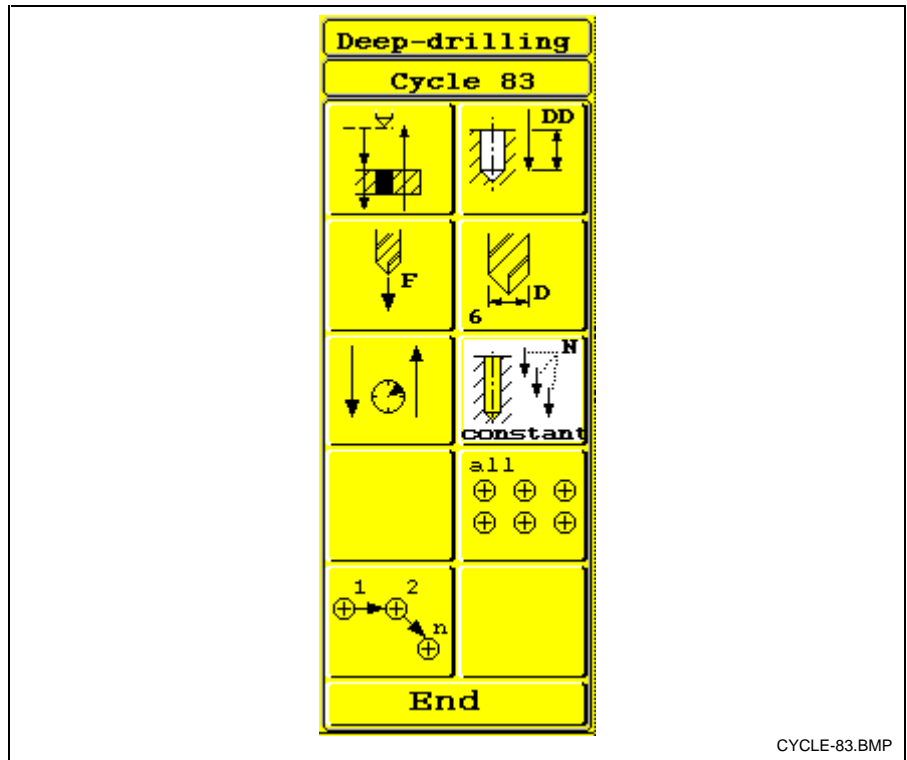
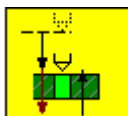


Fig. 3-54: Example: Deep hole drilling cycle

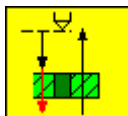
Programming

Center drilling	CYCLE_81 (Parameter list)
Deep hole drilling	CYCLE_83 (Parameter list)
Tapping	CYCLE_84 (Parameter list)
Reaming	CYCLE_85 (Parameter list)
Boring	CYCLE_86 (Parameter list)
Backward boring	CYCLE_87 (Parameter list)
Step boring	CYCLE_88 (Parameter list)

Tool retracting

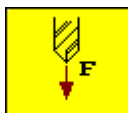


- The tool does not move back to the start level



- The tool moves back to the start level

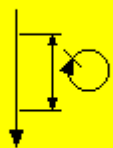
Feed rate



- Feed rate input selectable as mm/inch or mm/revolutions.
- Selectable <F7> key.

Drilling

Feedrate

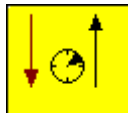


Feed/Revolution

Feedrate :

VORSCHUB.BMP

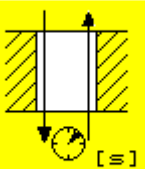
Dwell



- defines the dwell (in seconds) at the borehole bottom.

Drilling

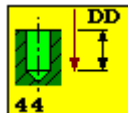
Dwell Time



Dwell time :

VERWEIL.BMP

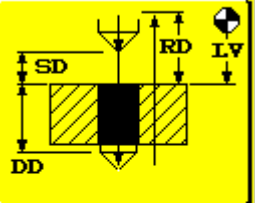
Drilling depth



- The infeed movements are performed on the basis of the parameters level, drill depth, safety distance, and return distance. Level is always an absolute dimension. The other data items are relative to the level value.

Deep-drilling

Depth



Level (LV) :

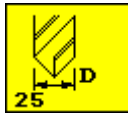
DrillDepth DD:

S.Distance SD:

R. niveau RL:

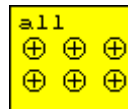
BOHREN8.BMP

Tool

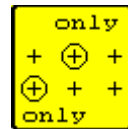


- Specification of the tool diameter and other relevant tool data.

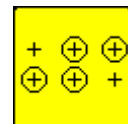
Point modification



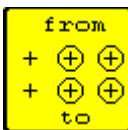
- All drilling points of the selected hole pattern are machined.



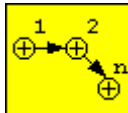
- Only selected points are machined (up to 4 drilling points).



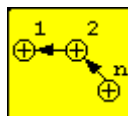
- The selected points are excluded (up to 4 drilling points).
- The selection of the points that are to be machined is delimited by two points. The sequence selected during the definition must be observed.



Machining direction



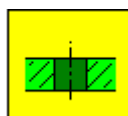
- The hole pattern is machined in the forward direction (in the definition direction).



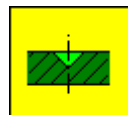
- The hole pattern is machined in the reverse direction (opposite to the definition direction).

Auxiliary function - drilling

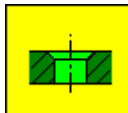
Drill variants



- Drilling



- Countersinking



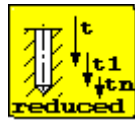
- Chamfering

Auxiliary function - deep hole drilling

Deep hole drilling variants



- Deep hole drilling with constant depth segments and modifiable run-out path via the <F7> function key.



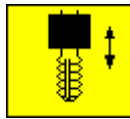
- Deep hole drilling with first depth segment and constant reduced further depth segment. The run-out path can be modified via the <F7> function key.



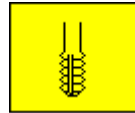
- Deep hole drilling with specification of the first depth segment.

Auxiliary function - Tapping chuck

Tool chucking variants



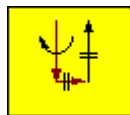
- which compensatory chuck



- rigid tapping

Auxiliary function - boring

Retract variants



- Defines spindle stop during retraction



- Defines retraction with rotating spindle

4 Data Adaptation

Function Data adaptation makes it possible to influence the following components:

- Technical parameters,
- Menu selections,
- Setting of user-related NC blocks
- Axis assignments
- Archiving GNE parameters

Invocation A branch is automatically made to data adaptation, if the graphical NC Editor (GNE) is called up for the first time for the selected station (process). Otherwise, the <GNE selection> function key <F7> within the standard GUI „Setting“ is used for invocation. Additionally, whenever a new GNE instruction shall be created via the NC editor <F6> (except for the first instruction), data adaptation can be invoked via the <F7> function key within the GNE main menu.

Technical parameters

Various input data or selections, such as the maximum cutting depth, run-out values, menu selection etc. may be preselected as technical parameters. Here, commands may be defined that shall be generated automatically if there is a discrepancy between the actual tool parameters and the tool parameters that are programmed in the cycle. These commands must then be interpreted by the machine manufacturers within the SPS program.

Menu setting

Using the menu settings, there is a possibility to arrange the required menus for conversational guidance according to the particular machine configuration from a catalogue of functions. Starting with the main menu, up to four subsequent menus can be called. The menu structure is defined via soft key operation. An integrated screen editor allows you to arrange customized menu items. The dialogue for menu structuring is self-explanatory.

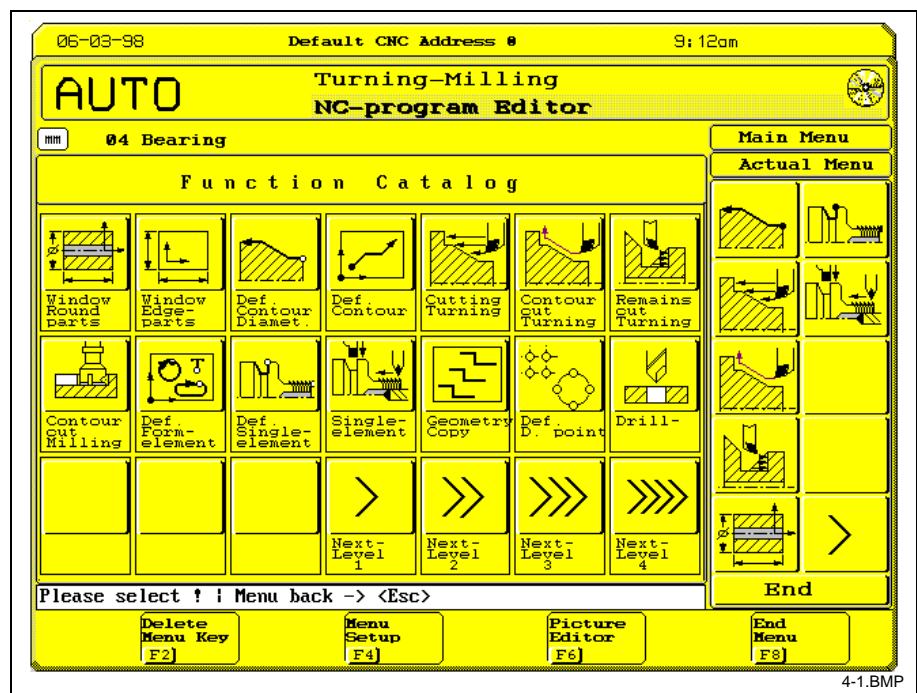


Fig. 4-1: User-specific menu generation

Setting user-related NC blocks

This function permits up to 12 different block sequences to be predefined. Each sequence may consist of up to 12 individual blocks. The total number of characters within a sequence is limited to 295 characters. The text entry can be used for designating the defined blocks. The sequences are invoked via the GUI editor „Special function“, function key <F3> in the „Editor window“.

Axis assignment

The axis assignment menu defines the possible combinations (planes) for the WINDOW instruction, which are required for machining. There are 8 combinations possible for the milling technology and 3 for turning. The axis combination number is entered in the WINDOW instruction as the fifth parameter (turning 1...3, milling 11...18). These depend first of all of the machine possibilities and of the parameter setting within the controller. Each axes combination is numbered consecutively and represented graphically. This is also the case, if equal combinations are defined repeatedly. It can be used for multi plane machining or rechucking operation. .

Default settings:

With axis assignment for milling machines (x-y-z), the f6 key (default) is available for supporting the new entry (<F6> - new milling).

Note: When parts programs are exchanged (import/export), it must always be ensured that axis group numbers and their axis designations are compatible. The last parameter of each window instruction and of each geometry definition is the number of the axis group.

Typical settings

1 st main axis	2 nd main axis	3 rd main axis	Comment
+Z	+X	n/a	Turning G18
+X	+C	+Z	Drilling/milling transformation face
+X	+Y	+Z	Drilling/milling G17
+X	-Z	+Y	Drilling/milling G18
-Z	+Y	+X	Drilling/milling G19
-X	+Y	-Z	Drilling/milling „other side“ G17
+X	+Z	-Y	Drilling/milling „other side“ G18
+Z	+Y	-X	Drilling/milling „other side“ G19
+X	+C	+Z	Drilling without transformation

Saving GNE parameters

This function permits all GNE data items (technical parameters, menu selections, axis assignments) to be saved on an external data carrier and/or loading these items from there.

Note: Always use a blank diskette when you store data on a diskette.

5 NC EDITOR - Special Function F3

Application	Using the <F3> function key, the NC editor permits the following functions to be used within a modification or insertion: <ul style="list-style-type: none"> • User-related NC blocks • Determining fit dimensions • Speed / cutting velocity conversion • Formula interpreter • Point - line - circle calculations • Special NC functions Pressing soft key <F8> returns back to and enters the results in the NC editor.
User-related NC blocks	This function enables predefined block sequences to be accepted. The content is defined either via the <F3> function key or via the data adaptation function „Setting user-related NC blocks“.
Fits	The clearance dimensions of the fitting (shaft/bore, DIN 7154T1) can be calculated after inputting the nominal diameter, tolerance and the dimension variant (upper, lower, intermediate) required for machining.
Spindle speed calculation	The spindle speed is calculated based on diameter and cutting velocity.
Cutting velocity	The cutting velocity is calculated based on diameter and spindle speed.
Formula interpreter	Calculation of formulae e.g. $\text{SIN}(33.7) * 115.35$

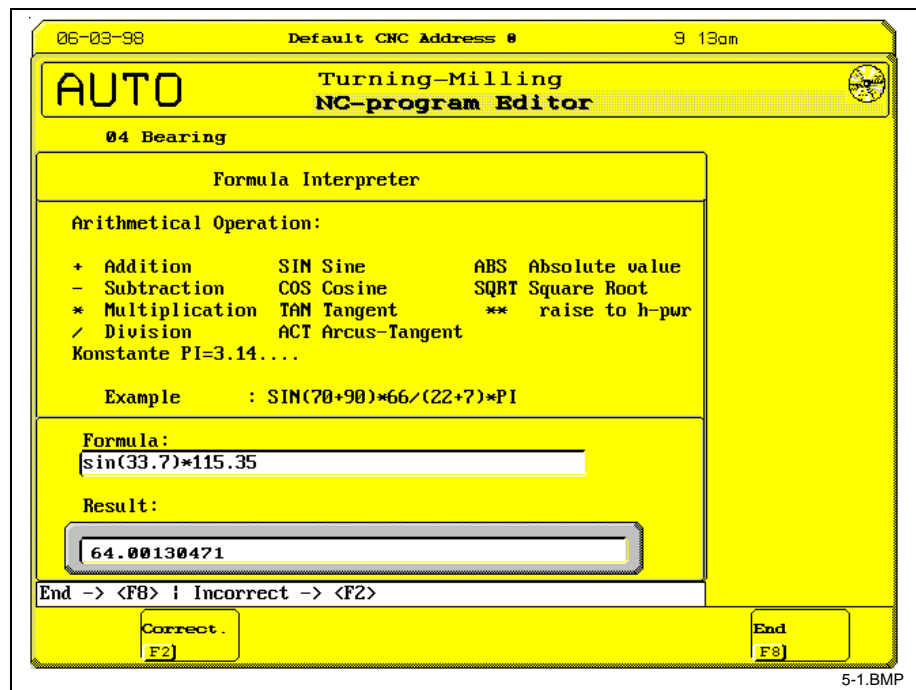


Fig. 5-1: Formula interpreter

Intersection points	Auxiliary contours can be calculated using the point-line-arc calculation. When accepting (see Abb. 5-3), only the values of the latest calculation step are transmitted. Example (see Fig. 5-2)
----------------------------	--

Example

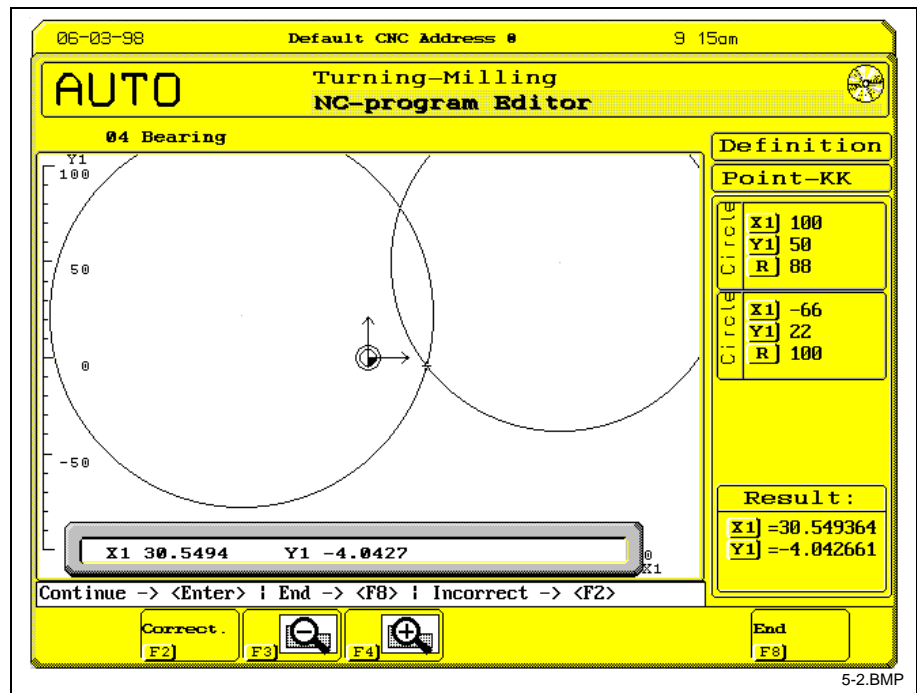


Fig. 5-2: Intersection of two circles

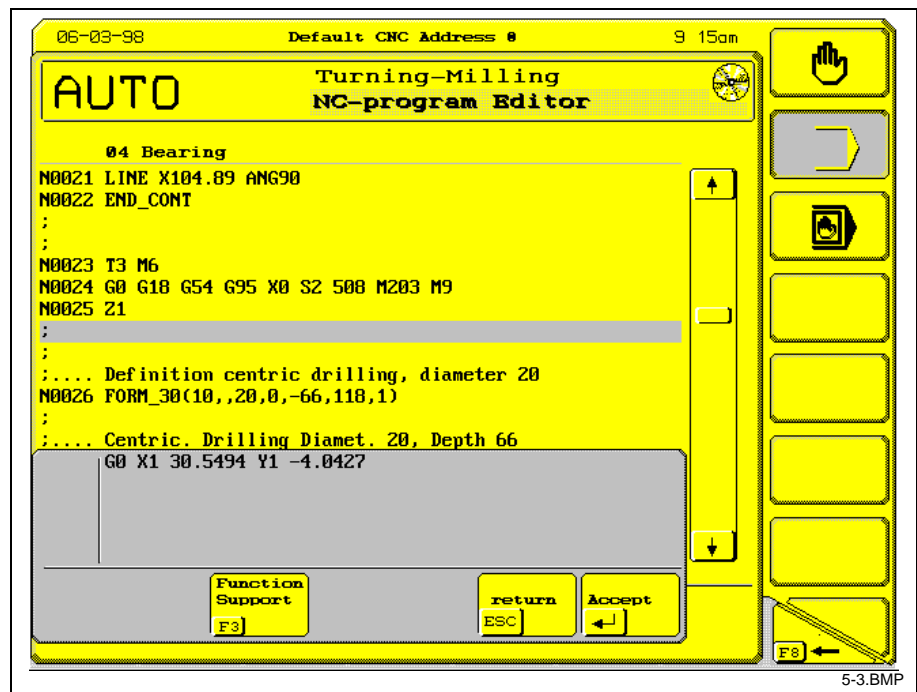


Fig. 5-3: Inserting the results

Special NC functions

Conversational guidance for special NC functions, i.e. reading and writing offset table data (OTD).

6 Error Messages

Message	Cause / ⇒ remedial action
Error when storing GNE data	Disk full or write-protected
Contour outside of initial form or window	There is no material to be machined ⇒ Wrong machining window or wrong initial contour
Contour cannot be machined completely	Tool shape does not allow complete machining ⇒ Modify tool parameter or rework the contour
Contour was modified	⇒ Confirm cycle in GNE dialogue analogously to „Modifications
Arc radius too small for the tool	⇒ Modify tool parameter or rework the contour
Arc radius too small for the tool or contour bottle-neck	Tool radius too small ⇒ Modify tool parameter or rework the contour
Deletion not permitted - Reference in block N...	The parts program contains instructions that refer to this instruction. ⇒ Clear those instructions first.
Check machining cycles of the modified contour	A cycle was defined within the NC program that relates to the modified contour. ⇒ Check if the cycle's parameters are still correct although the contour was modified
Corner rounding radius too large	Rounding does not fit between the defined elements. ⇒ Select smaller radius
Illegal change of allowance	Combination of contour and allowance cannot be specified ⇒ Change allowance
Tool radius modified	Inconsistency between tool radius NC program cycle and tool list ⇒ Change parameters within the NC program
Too many contour elements or no. of points	Geometry memory is full

7 Index

- Contour 3-2; 3-3; 3-6; 3-10; 3-13; 3-15; 3-16; 3-18
- Contour correction
 - Correct
 - Contour start point 3-18
 - Delete
 - Corner roundings, chamfers, relief grooves 3-15
- Copy 3-21
- Data 4-1; 4-2
- Drill 3-53; 3-54; 3-55; 3-56
- Drilling 3-21; 3-52
- Error 2-6; 3-20; 3-43; 6-1
- Exiting GNE 2-2
- Form 3-20
- General information
 - Application 2-1
 - Dialog 2-1
 - Function 1-1
 - Invocation 2-1
 - Ordering information 1-2
 - Result 2-1
- Geometry definition
 - (Contour definition, contour correction, form elements, hole patterns, copy instructions) 3-2
- GNE 2-2; 2-3
- GNE instruction
 - Deleting
 - CONT 2-6
 - COPY, FORM 2-6
 - CYCLE 2-6
 - Invocation 2-6
 - Modify
 - Multiple-line instruction 2-5
 - Modifying
 - First step 2-5
 - Second step 2-5
 - Single-line instruction 2-5
 - Programming
 - Exceptions 2-3
 - First invocation 2-3
 - First step 2-3
 - Menu 2-3
 - Second step 2-3
 - Subsequent invocation 2-3
 - Units 2-3
 - Zoom 2-3
- 2-6
- Machining 3-23; 3-24
- Milling 3-45; 3-46; 3-47; 3-49; 3-50; 3-51
- Milling cycles
 - Approach and run-out 3-47
- NC editor special function - F3
 - Application 5-1
 - Cutting velocity 5-1
 - Example 5-2
 - Fits 5-1
 - Formula interpreter 5-1
 - Intersection points 5-1
 - Speed calculation 5-1
- Turning 3-37; 3-39; 3-40
- Turning cycle
 - Center boring
 - Boring, reaming, thread cutting
 - Bore depth 3-42
 - Dwell 3-41
 - Feed rate 3-41
 - Tool 3-42

- Tool retrace 3-41
- Turning cycles
 - Cutting, contouring, residual cutting, form element 3-25
 - Approach and run-out 3-38
 - Chip breakage 3-37
 - Cutting cycle example 3-43
 - Cutting depth 3-30
 - Cutting variants 3-31
 - Feed rate 3-32
 - Initial form 3-35
 - Invocation 3-25
 - Machining tool 3-33
 - Machining window 3-28
 - Parameter list 3-27
 - Programming 3-27
 - Remaining stock 3-27
 - Retract variants 3-37
 - Roughing tool 3-33
 - Sequence 3-42
 - Single element 3-37
 - Tool retract 3-30
 - Undercut variant 3-31
- Window
 - Application 3-1
 - Clamping/chucking 3-1
 - Exceptions 3-1
 - Function 3-1
 - Invocation 3-1
 - Parameter 3-1
 - Programming 3-1
 - Validity 3-1

Kundenbetreuungsstellen - Sales & Service Facilities

Deutschland – Germany

vom Ausland:
from abroad:

(0) nach Landeskennziffer weglassen!!
don't dial (0) after country code!

Vertriebsgebiet Mitte Germany Centre	<input checked="" type="checkbox"/> SALES <input checked="" type="checkbox"/> Service	Vertriebsgebiet Ost Germany East	<input checked="" type="checkbox"/> SALES <input checked="" type="checkbox"/> Service	Vertriebsgebiet West Germany West	<input checked="" type="checkbox"/> SALES <input checked="" type="checkbox"/> Service	Vertriebsgebiet Nord Germany North	<input checked="" type="checkbox"/> SALES <input type="checkbox"/> Service
INDRAMAT GmbH Bgm.-Dr.-Nebel-Str. 2 D - 97816 Lohr am Main Telefon: +49 (0)9352/40-0 Telefax: +49 (0)9352/40-4885		INDRAMAT GmbH Beckerstraße 31 D - 09120 Chemnitz Telefon: +49 (0)371/35 55-0 Telefax: +49 (0)371/35 55-333		INDRAMAT GmbH Harkortstraße 25 D - 40849 Ratingen Telefon: +49 (0)2102/43 18-0 Telefax: +49 (0)2102/41 315		INDRAMAT GmbH Kieler Straße 212 D - 22525 Hamburg Telefon: +49 (0)40/85 31 57-0 Telefax: +49 (0)40/85 31 57-15	
Vertriebsgebiet Süd Germany South	<input checked="" type="checkbox"/> SALES <input type="checkbox"/> Service	Gebiet Südwest Germany South-West	<input checked="" type="checkbox"/> SALES <input checked="" type="checkbox"/> Service			INDRAMAT Service-Hotline INDRAMAT GmbH Telefon: +49 (0)172/660 04 06 oder/or Telefon: +49 (0)171/333 88 26	
INDRAMAT GmbH Ridlerstraße 75 D-80339 München Telefon: +49 (0)89/540138-30 Telefax: +49 (0)89/540138-10		INDRAMAT GmbH Böblinger Straße 25 D-71229 Leonberg Telefon: +49 (0)7152/9 72-6 Telefax: +49 (0)7152/9 72-727					

Kundenbetreuungsstellen in Deutschland - Service agencies in Germany

Europa – Europe

vom Ausland: (0) nach Landeskennziffer weglassen, 0 nach Landeskennziffer mitwählen!
from abroad: don't dial (0) after country code, dial 0 after country code!

<p>Austria <input checked="" type="checkbox"/> SALES <input type="checkbox"/> Service</p> <p>Mannesmann Rexroth Ges.m.b.H. Geschäftsbereich INDRAMAT Hägelingasse 3 A - 1140 Wien</p> <p>Telefon: +43 (0)1/9852540-400 Telefax: +43 (0)1/9852540-93</p>	<p>Austria <input checked="" type="checkbox"/> SALES <input type="checkbox"/> Service</p> <p>Mannesmann Rexroth G.m.b.H. Geschäftsbereich INDRAMAT Industriepark 18 A - 4061 Pasching</p> <p>Telefon: +43 (0)7221/605-0 Telefax: +43 (0)7221/605-21</p>	<p>Belgium <input checked="" type="checkbox"/> SALES <input checked="" type="checkbox"/> Service</p> <p>Mannesmann Rexroth N.V.-S.A. Geschäftsbereich INDRAMAT Industrielaan 8 B-1740 Ternat</p> <p>Telefon: +32 (0)2/5823180 Telefax: +32 (0)2/5824310</p>	<p>Denmark <input checked="" type="checkbox"/> SALES <input checked="" type="checkbox"/> Service</p> <p>BEC AS Zinkvej 6 DK-8900 Randers</p> <p>Telefon: +45 (0)87/11 90 60 Telefax: +45 (0)87/11 90 61</p>
<p>England <input checked="" type="checkbox"/> SALES <input checked="" type="checkbox"/> Service</p> <p>Mannesmann Rexroth Ltd. INDRAMAT Division Broadway Lane, South Cerney GB - Cirencester, Glos GL7 5UH</p> <p>Telefon: +44 (0)1285/863000 Telefax: +44 (0)1285/863030</p>	<p>Finland <input checked="" type="checkbox"/> SALES <input type="checkbox"/> Service</p> <p>Rexroth Mecman OY INDRAMAT division SF-017 40 Vantaa</p> <p>Telefon: +358 (0)9/84 91 11 Telefax: +358 (0)9/84 91 13 60</p>	<p>France <input checked="" type="checkbox"/> SALES <input checked="" type="checkbox"/> Service</p> <p>Mannesmann Rexroth S.A. Division INDRAMAT Parc des Barbanniers 4, Place du Village F-92632 Gennevilliers Cedex</p> <p>Telefon: +33 (0)141 47 54 30 Telefax: +33 (0)147 94 69 41 Hotline: +33 (0)6 08 33 43 28</p>	<p>France <input checked="" type="checkbox"/> SALES <input type="checkbox"/> Service</p> <p>Mannesmann Rexroth S.A. Division INDRAMAT 270, Avenue de Lardenne F - 31100 Toulouse</p> <p>Telefon: +33 (0)5 61 49 95 19 Telefax: +33 (0)5 61 31 00 41</p>
<p>France <input checked="" type="checkbox"/> SALES <input type="checkbox"/> Service</p> <p>Mannesmann Rexroth S.A. Division INDRAMAT 91, Bd. Irène Joliot-Curie F - 69634 Vénissieux – Cedex</p> <p>Telefon: +33 (0)4 78 78 53 65 Telefax: +33 (0)4 78 78 52 53</p>	<p>Italy <input checked="" type="checkbox"/> SALES <input checked="" type="checkbox"/> Service</p> <p>Mannesmann Rexroth S.p.A. Divisione INDRAMAT Via G. Di Vittoria, 1 I - 20063 Cernusco S/N.MI</p> <p>Telefon: +39 02/92 36 52 70 Telefax: +39 02/92 36 55 12</p>	<p>Italy <input checked="" type="checkbox"/> SALES <input checked="" type="checkbox"/> Service</p> <p>Mannesmann Rexroth S.p.A. Divisione INDRAMAT Via Borgomanero, 11 I - 10145 Torino</p> <p>Telefon: +39 011/7 71 22 30 Telefax: +39 011/7 71 01 90</p>	<p>Italy <input checked="" type="checkbox"/> SALES <input type="checkbox"/> Service</p> <p>Mannesmann Rexroth S.p.A. Divisione INDRAMAT Via del Progresso, 16 (Zona Ind.) I - 35020 Padova</p> <p>Telefon: +39 049/8 70 13 70 Telefax: +39 049/8 70 13 77</p>
<p>Italy <input type="checkbox"/> SALES <input checked="" type="checkbox"/> Service</p> <p>Mannesmann Rexroth S.p.A. Divisione INDRAMAT Via de Nicola, 12 I - 80053 Castellamare di Stabia NA</p> <p>Telefon: +39 081/8 72 30 37 Telefax: +39 081/8 72 30 18</p>	<p>Italy <input checked="" type="checkbox"/> SALES <input type="checkbox"/> Service</p> <p>Mannesmann Rexroth S.p.A. Divisione INDRAMAT Viale Oriani, 38/A I - 40137 Bologna</p> <p>Telefon: +39 051/34 14 14 Telefax: +39 051/34 14 22</p>	<p>Netherlands <input checked="" type="checkbox"/> SALES <input type="checkbox"/> Service</p> <p>Hydraudyne Hydrauliek B.V. Kruisbroeksestraat 1 (P.O. Box 32) NL - 5281 RV Boxtel</p> <p>Telefon: +31 (0)411/65 19 51 Telefax: +31 (0)411/65 14 83 e-mail: indramat@hydraudyne.nl</p>	<p>Netherlands <input type="checkbox"/> SALES <input checked="" type="checkbox"/> Service</p> <p>Hydrocare B.V. Kruisbroeksestraat 1 (P.O. Box 32) NL - 5281 RV Boxtel</p> <p>Telefon: +31 (0)411/65 19 51 Telefax: +31 (0)411/67 78 14</p>
<p>Poland <input checked="" type="checkbox"/> SALES <input type="checkbox"/> Service</p> <p>Mannesmann Rexroth Sp.zo.o. Biuro Poznan ul. Dabrowskiego 81/85 PL - 60-529 Poznan</p> <p>Telefon: +48 061/847 67 99 Telefax: +48 061/847 64 02</p>	<p>Russia <input type="checkbox"/> SALES <input checked="" type="checkbox"/> Service</p> <p>Tschudnenko E.B. Arsenia 22 RUS - 153000 Ivanovo Rußland</p> <p>Telefon: +7 093/223 96 33 oder/or +7 093/223 95 48 Telefax: +7 093/223 46 01</p>	<p>Spain <input checked="" type="checkbox"/> SALES <input checked="" type="checkbox"/> Service</p> <p>Mannesmann Rexroth S.A. Division INDRAMAT Centro Industrial Santiga Obradors s/n E-08130 Santa Perpetua de Mogoda Barcelona</p> <p>Telefon: +34 937 47 94 00 Telefax: +34 937 47 94 01</p>	<p>Spain <input checked="" type="checkbox"/> SALES <input checked="" type="checkbox"/> Service</p> <p>Goimendi S.A. División Indramat Jolastokieta (Herrera) Apartado 11 37 E - 20017 San Sebastian</p> <p>Telefon: +34 9 43/40 01 63 Telefax: +34 9 43/39 17 99</p>
<p>Sweden <input checked="" type="checkbox"/> SALES <input checked="" type="checkbox"/> Service</p> <p>Rexroth Mecman Svenska AB INDRAMAT Division Varuvägen 7 S - 125 81 Stockholm</p> <p>Telefon: +46 (0)8/727 92 00 Telefax: +46 (0)8/647 32 77</p>	<p>Slovenia <input checked="" type="checkbox"/> SALES <input checked="" type="checkbox"/> Service</p> <p>INDRAMAT elektromotorji d.o.o. Otoki 21 SLO - 64 228 Zelezniki</p> <p>Telefon: +386 64/61 73 32 Telefax: +386 64/64 71 50</p>	<p>Turkey <input checked="" type="checkbox"/> SALES <input checked="" type="checkbox"/> Service</p> <p>Mannesmann Rexroth Hidropar A.S. Fevzi Cakmak Cad No. 3 TR - 34630 Sefaköy Istanbul</p> <p>Telefon: +90 212/541 60 70 Telefax: +90 212/599 34 07</p>	
<p>Switzerland -East- <input checked="" type="checkbox"/> SALES <input checked="" type="checkbox"/> Service</p> <p>Mannesmann Rexroth Schweiz AG Geschäftsbereich INDRAMAT Gewerbstraße 3 CH-8500 Frauenfeld</p> <p>Telefon: +41 (0)52/720 21 00 Telefax: +41 (0)52/720 21 11</p>	<p>Switzerland -West- <input checked="" type="checkbox"/> SALES <input type="checkbox"/> Service</p> <p>Mannesmann Rexroth Suisse SA Département INDRAMAT Rue du village 1 CH-1020 Renens</p> <p>Telefon: +41 (0)21/632 84 20 Telefax: +41 (0)21/632 84 21</p>		

Europäische Kundenbetreuungsstellen (ohne Deutschland)
 European Service agencies (without Germany)

Außerhalb Europa - outside Europe

vom Ausland:

(0) nach Landeskenziffer weglassen!

from abroad:

don't dial (0) after country code!

<p>Argentina <input checked="" type="checkbox"/> SALES <input type="checkbox"/> Service</p> <p>Mannesmann Rexroth S.A.I.C. Division INDRAMAT Acassuso 48 41/7 RA - 1605 Munro (Buenos Aires)</p> <p>Telefon: +54 (0)1/756 01 40 +54 (0)1/756 01 36</p>	<p>Argentina <input checked="" type="checkbox"/> SALES <input checked="" type="checkbox"/> Service</p> <p>NAKASE Servicio Tecnico CNC Calle 49, No. 5764/66 RA - 1653 Villa Balester Prov. - Buenos Aires</p> <p>Telefon: +54 (0) 11/4768 36 43 Telefax: +54 (0) 11/4768 24 13 e-mail: nakase@usa.net nakase@infovia.com.ar</p>	<p>Australia <input checked="" type="checkbox"/> SALES <input checked="" type="checkbox"/> Service</p> <p>AIMS - Australian Industrial Machinery Services Pty. Ltd. Unit 3/45 Home ST Campbellfield , VIC 3061 AUS - Melbourne</p> <p>Telefon: +61 (0)3/93 59 02 28 Telefax: +61 (0)3/93 59 02 86</p>	<p>Australia <input checked="" type="checkbox"/> SALES <input type="checkbox"/> Service</p> <p>Mannesmann Rexroth Pty. Ltd. No. 7, Endeavour Way Braeside Victoria, 31 95 AUS - Melbourne</p> <p>Telefon: +61 (0)3/95 80 39 33 Telefax: +61 (0)3/95 80 17 33 Email: mel@rexroth.com.au</p>
<p>Brazil <input checked="" type="checkbox"/> SALES <input checked="" type="checkbox"/> Service</p> <p>Mannesmann Rexroth Automação Ltda. Divisão INDRAMAT Rua Georg Rexroth, 609 Vila Padre Anchieta BR - 09951-270 Diadema-SP [Caixa Postal 377] [BR-09901-970 Diadema-SP]</p> <p>Telefon: +55 (0)11/745 90 60 +55 (0)11/745 90 70 Telefax: +55 (0)11/745 90 50</p>	<p>Brazil <input type="checkbox"/> SALES <input checked="" type="checkbox"/> Service</p> <p>Mannesmann Rexroth Automação Ltda. Divisão INDRAMAT Rua Umberto Pinheiro Vieira, 100 Distrito Industrial BR - 09220-390 Joinville - SC [Caixa Postal 1273]</p> <p>Tel./Fax: +55 (0)47/473 55 833 Mobil: +55 (0)47 974 6645 e-mail: prochnow@zaz.com.br</p>	<p>Canada <input checked="" type="checkbox"/> SALES <input checked="" type="checkbox"/> Service</p> <p>Basic Technologies Corporation Burlington Division 3426 Mainway Drive Burlington, Ontario Canada L7M 1A8</p> <p>Telefon: +1 905/335 55 11 Telefax: +1 905/335-41 84</p>	<p>China <input type="checkbox"/> SALES <input checked="" type="checkbox"/> Service</p> <p>Mannesmann Rexroth (China) Ltd. Shanghai Parts & Service Center 199 Wu Cao Road, Hua Cao Minhang District PRC - Shanghai 201 103</p> <p>Telefon: +86 21/62 20 00 58 Telefax: +86 21/62 20 00 68</p>
<p>China <input checked="" type="checkbox"/> SALES <input type="checkbox"/> Service</p> <p>Mannesmann Rexroth (China) Ltd. 15/F China World Trade Center 1, Jianguomenwai Avenue PRC - Beijing 100004</p> <p>Telefon: +86 10/65 05 03 80 Telefax: +86 10/65 05 03 79</p>	<p>China <input checked="" type="checkbox"/> SALES <input type="checkbox"/> Service</p> <p>Mannesmann Rexroth (China) Ltd. A-5F., 123 Lian Shan Street Sha He Kou District PRC - Dalian 116 023</p> <p>Telefon: +86 411/46 78 930 Telefax: +86 411/46 78 932</p>	<p>Hongkong <input checked="" type="checkbox"/> SALES <input checked="" type="checkbox"/> Service</p> <p>Rexroth (China) Ltd. 1/F., 19 Cheung Shun Street Cheung Sha Wan, Kowloon, Hongkong</p> <p>Telefon: +852 22 62 51 00 Telefax: +852 27 44 02 78</p>	<p>India <input checked="" type="checkbox"/> SALES <input checked="" type="checkbox"/> Service</p> <p>Mannesmann Rexroth (India) Ltd. INDRAMAT Division Plot. 96, Phase III Peenya Industrial Area IND - Bangalore - 560058</p> <p>Telefon: +91 (0)80/8 39 73 74 Telefax: +91 (0)80/8 39 43 45</p>
<p>India <input checked="" type="checkbox"/> SALES <input checked="" type="checkbox"/> Service</p> <p>Mannesmann Rexroth (India) Ltd. INDRAMAT Division Plot. A-58, TTC Industrial Area Thane Turbhe Midc Road Mahape Village IND - Navi Mumbai - 400 701</p> <p>Telefon: +91 (0)22/7 61 46 22 Telefax: +91 (0)22/7 68 15 31</p>	<p>Indonesia <input checked="" type="checkbox"/> SALES <input type="checkbox"/> Service</p> <p>PT. Rexroth Wijayakusuma Jl. Raya Bekasi Km 21 Pulogadung RI - Jakarta Timur 13920</p> <p>Telefon: +62 21/4 61 04 87 +62 21/4 61 04 88 Telefax: +62 21/4 60 01 52</p>	<p>Japan <input type="checkbox"/> SALES <input checked="" type="checkbox"/> Service</p> <p>Rexroth Automation Co., Ltd. Nagoya Service Center Yutakagaoka 1810 Meito-ku, NAGOYA 465-0035 Japan</p> <p>Telefon: +81 (0)52/777 88 41 +81 (0)52/777 88 53 +81 (0)52/777 88 79 Telefax: +81 (0)52/777 89 01</p>	<p>Japan <input checked="" type="checkbox"/> SALES <input checked="" type="checkbox"/> Service</p> <p>Rexroth Automation Co., Ltd. INDRAMAT Division 1F, I.R. Building Nakamachidai 4-26-44 Tsuzuki-ku, YOKOHAMA 224-0041 Japan</p> <p>Telefon: +81 459/42-72 10 Telefax: +81 459/42-03 41</p>
<p>Mexico <input checked="" type="checkbox"/> SALES <input type="checkbox"/> Service</p> <p>Rexroth Mexico S.A. de C.V. Calle Neptuno 72 Unidad Ind. Vallejo MEX - 07700 Mexico, D.F.</p> <p>Telefon: +52 5 754 17 11 +52 5 754 36 84 +52 5 754 12 60 Telefax: +52 5 754 50 73 +52 5 752 59 43</p>	<p>Korea <input checked="" type="checkbox"/> SALES <input checked="" type="checkbox"/> Service</p> <p>Mannesmann Rexroth-Seki Co Ltd. 1500-12 Da-Dae-Dong ROK - Saha-Ku, Pusan, 604-050</p> <p>Telefon: +82 (0)51/2 60 06 18 Telefax: +82 (0)51/2 60 06 19</p>	<p>Korea <input checked="" type="checkbox"/> SALES <input checked="" type="checkbox"/> Service</p> <p>Seo Chang Corporation Ltd. Room 903, Jail Building 44-35 Yeouido-Dong Yeoungdeungpo-Ku C.P.O.Box 97 56 ROK - Seoul</p> <p>Telefon: +82 (0)2/7 80 82 08 +82 (0)2/7 80 82 09 Telefax: +82 (0)2/7 84 54 08</p>	<p>South Africa <input checked="" type="checkbox"/> SALES <input checked="" type="checkbox"/> Service</p> <p>HYTEC Automation (Pty) Ltd. 28 Banfield Road, Industria North RSA - Maraisburg 1700</p> <p>Telefon: +27 (0)11/673 20 80 Telefax: +27 (0)11/673 72 69</p>
<p>Taiwan <input checked="" type="checkbox"/> SALES <input type="checkbox"/> Service</p> <p>Rexroth Uchida Co., Ltd. No.1, Tsu Chiang Street Tu Cheng Ind. Estate Taipei Hsien, Taiwan, R.O.C.</p> <p>Telefon: +886 2/2 68 13 47 Telefax: +886 2/2 68 53 88</p>			

Kundenbetreuungsstellen außerhalb Europa - Service agencies outside Europe

Außerhalb Europa / USA - outside Europe / USA

<p>USA <input checked="" type="checkbox"/> SALES <input checked="" type="checkbox"/> Service</p> <p>Mannesmann Rexroth Corporation INDRAMAT Division 5150 Prairie Stone Parkway USA -Hoffman Estates, IL 60192-3707 Telefon: +1 847/6 45 36 00 Telefax: +1 847/6 45 62 01</p>	<p>USA <input checked="" type="checkbox"/> SALES <input checked="" type="checkbox"/> Service</p> <p>Mannesmann Rexroth Corporation INDRAMAT Division Central Region Technical Center USA - Auburn Hills, MI 48326 Telefon: +1 248/3 93 33 30 Telefax: +1 248/3 93 29 06</p>	<p>USA <input checked="" type="checkbox"/> SALES <input checked="" type="checkbox"/> Service</p> <p>Mannesmann Rexroth Corporation INDRAMAT Division Southeastern Technical Center 3625 Swiftwater Park Drive USA - Suwanee Georgia 30174 Telefon: +1 770/9 32 32 00 +1 770/9 32 19 03</p>	<p>USA <input checked="" type="checkbox"/> SALES <input checked="" type="checkbox"/> Service</p> <p>Mannesmann Rexroth Corporation INDRAMAT Division Northeastern Technical Center 99 Rainbow Road USA - East Granby, Connecticut 06026 Telefon: +1 860/8 44 83 77 +1 860/8 44 85 95</p>
<p>USA <input checked="" type="checkbox"/> SALES <input checked="" type="checkbox"/> Service</p> <p>Mannesmann Rexroth Corporation INDRAMAT Division Charlotte Regional Sales Office 14001 South Lakes Drive USA - Charlotte, North Carolina 28273 Telefon: +1 704/5 83 97 62 +1 704/5 83 14 86</p>			

Kundenbetreuungsstellen außerhalb Europa / USA
 Service agencies outside Europe / USA

Notes

278811

Printed in Germany