

Operation/Programming 08/2002 Edition

sinumerik

ManualTurn
SINUMERIK 840D/810D

SIEMENS

SINUMERIK 840D/810D

ManualTurn

Operation/Programming

Valid for

<i>Control</i>	<i>Software version</i>
SINUMERIK 840D	6
SINUMERIK 840DE (export version)	6
SINUMERIK 840D powerline	6
SINUMERIK 840DE powerline	6
SINUMERIK 810D powerline	6
SINUMERIK 810DE powerline	6

Introduction	1
Operation	2
Preparatory Functions for Machining	3
Turning Simple Contours	4
Turning with Cycles	5
Turning any Contours	6
Program Creation with EasyStep and G Code	7
Parts Program Management	8
General Functions	9
Intervention in the Machining Process	10
Alarms and Messages	11
Examples	12
Appendix	A

SINUMERIK® Documentation

Printing history

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

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

Status code in the "Remarks" column:

- A** New documentation.
- B** Unrevised edition with new Order No.
- C** Revised edition with new status.

If factual changes have been made on the page since the last edition, this is indicated by a new edition coding in the header on that page.

Edition	Order No.	Remarks
06.97	6FC5298-2AD00-0BP0	A
12.97	6FC5298-2AD00-0BP1	C
07.98	6FC5298-2AD00-0BP2	C
02.00	6FC5298-5AD00-0BP0	C
08.00	6FC5298-5AD00-0BP1	C
08.02	6FC5298-6AD00-0BP0	C

This manual is included in the documentation available on CD ROM (**DOCONCD**)

Edition	Order No.	Remarks
11.02	6FC5 298-6CA00-0BG3	C

Trademarks

SIMATIC®, SIMATIC HMI®, SIMATIC NET®, SIROTEC®, SINUMERIK® and SIMODRIVE® are registered trademarks of Siemens AG. Other product names used in this documentation may be trademarks which, if used by third parties, could infringe the rights of their owners.

Further information is available on the Internet under:
<http://www.a&d.siemens.de/sinumerik>

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

This publication was produced with WinWord V8.0 and Designer V7.0. The reproduction, transmission or use of this document or its contents is not permitted without express written authority. Offenders will be liable for damages. All rights, including rights created by patent grant or registration of a utility model or design, are reserved.

We have checked that the contents of this document correspond to the hardware and software described. Nonetheless, differences might exist and therefore we cannot guarantee that they are completely identical. The information contained in this document is, however, reviewed regularly and any necessary changes will be included in the next edition. We welcome suggestions for improvement.

© Siemens AG, 1997, 1998, 2000, 2002. All rights reserved

Subject to change without prior notice

Contents

Introduction	1-13
1.1 The ManualTurn product.....	1-14
1.2 Operator notes	1-15
1.3 Switching on/switching off	1-16
Operation	2-17
2.1 Operator panels	2-18
2.2 Machine control panel	2-21
2.3 Mini handheld unit	2-25
2.4 Graphics interface	2-27
2.5 Operating system	2-29
2.5.1 Operating modes.....	2-30
2.5.2 Important function keys	2-32
2.5.3 Important soft keys.....	2-33
2.5.4 Pocket calculator	2-34
2.5.5 Absolute and incremental dimensions	2-36
2.5.6 Angle reference system	2-37
2.5.7 Tool and cutting data.....	2-38
Preparatory Functions for Machining	3-41
3.1 Approach reference points	3-42
3.2 Setup	3-44
3.3 Incremental feed mode	3-45
3.4 Offsets.....	3-47
3.4.1 General notes.....	3-47
3.4.2 Preset.....	3-48
3.4.3 Manual offset.....	3-49
3.4.4 Delete manual offset	3-50
3.4.5 Zero offset.....	3-51
3.5 Spindle speed limitation	3-52
3.6 Oriented spindle stop	3-53
3.7 C axis mode	3-54
3.8 Tool	3-55
3.8.1 Enter tool offset data	3-55
3.8.2 Selecting/deselecting tool offset.....	3-57
3.8.3 Measure tool	3-58
3.8.4 Tool wear compensation	3-60
3.9 Measuring system changeover inch/metric.....	3-61

Turning Simple Contours	4-63
4.1 Turning in manual mode.....	4-64
4.2 Turning with path dimension input.....	4-66
4.2.1 Turning with STRAIGHT mode.....	4-67
4.2.2 Turning with CONICAL mode.....	4-69
4.2.3 Turning with CIRCLE mode.....	4-71
4.3 Turning with the contour handwheel and JOG keys +/-.....	4-75
Turning with Cycles	5-79
5.1 General notes	5-80
5.2 Turning cycles in CYCLE mode.....	5-81
5.2.1 Thread cutting	5-81
5.2.2 Controlling thread cutting operations.....	5-87
5.2.3 Re-working a thread	5-88
5.2.4 Undercuts Form E and F	5-89
5.2.5 Thread undercuts	5-91
5.2.6 Drilling in longitudinal axis (center).....	5-93
5.2.7 Hole circle drilling	5-97
5.3 Stock removal/grooving cycles in STOCK REMOVAL mode.....	5-100
5.3.1 Stock removal cycles.....	5-100
5.3.2 Grooving cycles	5-105
Turning any Contours (Free Contour Input)	6-111
6.1 General notes	6-112
6.2 Create new contour	6-113
6.3 Symbolic representation of contour	6-114
6.4 Graphic representation of contour	6-115
6.5 Create contour elements	6-116
6.6 Editing contour elements	6-119
6.7 Stock removal against contour	6-122
6.8 Machining residual material.....	6-126
6.9 Single-cycle machining	6-127
Program Creation with EasyStep and G Code	7-129
7.1 General notes	7-130
7.2 Creating a machining sequence	7-131
7.3 Display machining sequence	7-132
7.4 Program steps	7-133
7.4.1 Inserting new program step	7-133
7.4.2 Special functions	7-134

7.4.3	Insert G code step	7-135
7.4.4	Changing program steps.....	7-135
7.4.5	Program editor	7-136
7.5	Switching off the program	7-137
7.6	Start machining sequence.....	7-137
7.7	Single-step mode (single block).....	7-138
7.8	Block search.....	7-138
7.9	Tool nose radius compensation	7-139
7.10	G code programming	7-141
7.10.1	Select program view.....	7-141
7.10.2	G code editor.....	7-143
7.10.3	Create new parts program	7-145
7.10.4	Inserting program blocks.....	7-145
7.10.5	Editing program blocks.....	7-146
Parts Program Management		8-149
8.1	General	8-150
8.2	Select a file.....	8-151
8.3	Delete a file	8-151
8.4	Storing thread undercut and thread cycles	8-152
8.5	Insert a contour in EasyStep machining sequence	8-152
8.6	Rename/copy file	8-153
8.7	Read out a file to an external medium	8-153
8.8	Read in a file	8-154
8.9	Error/transmission log	8-154
General Functions		9-155
9.1	Simulation and simultaneous recording	9-156
9.1.1	Simulation	9-158
9.1.2	Simultaneous recording	9-159
9.1.3	Dry run.....	9-159
9.2	Teach In	9-161
9.2.1	Selecting the Teach In function.....	9-161
9.2.2	Deselecting "Teach In"	9-162
9.2.3	Continuing "Teach In".....	9-162
9.2.4	Transferring machining steps to system	9-163
9.2.5	Transferring auxiliary functions to system.....	9-165
9.3	Standard CNC operation.....	9-166
Intervention in the Machining Process		10-167
10.1	Aborting a machining operation	10-168

10.2	Repositioning.....	10-169
10.3	Saving manual offset with "Store offset" function.....	10-170

Alarms and Messages 11-171

11.1	Alarms and messages in ManualTurn cycles.....	11-172
11.1.1	Error handling in cycles	11-172
11.1.2	Overview of cycle alarms.....	11-172
11.1.3	Messages in cycles	11-173
11.2	Alarms with ManualTurn.....	11-174
11.2.1	Overview of alarms.....	11-174
11.2.2	Selecting the alarm/message overview.....	11-174
11.2.3	Description of alarms.....	11-175

Examples 12-183

12.1	Example 1: External machining with groove and thread	12-184
12.2	Example 2: External machining with sphere	12-189
12.3	Example 3: External machining with thread undercuts and grooves.....	12-194
12.4	Example 4: External machining with thread undercut and groove	12-200

Appendix A-207

A	Abbreviations.....	A-208
B	Terms	A-211
C	References	A-214
D	Index.....	A-227

Preface

Organization of documentation	The SINUMERIK documentation is organized on 3 different levels: <ul style="list-style-type: none">• General Documentation• User Documentation• Manufacturer/Service Documentation
Reader group	This manual is intended for users (operators) of turning machines with SINUMERIK 840D/810D.
Validity	The Operator Control/Programming Guide is valid for ManualTurn SW 6.2 with <ul style="list-style-type: none">• SINUMERIK 810D (SW 6.3 and higher)• SINUMERIK 840D (SW 6.3 and higher)
Hotline	Please address any questions to the following hotline: A&D Technical Support Phone: ++49-(0)180-5050-222 Fax: ++49-(0)180-5050-223 E-Mail: adsupport@siemens.com
	If you have any questions (suggestions, corrections) concerning the documentation, please fax or e-mail them to the following address: Fax: ++49-(0)9131-98-2176 Fax form provided at the end of the document E-mail: motioncontrol.docu@erlf.siemens.de
Internet address	http://www.ad.siemens.de/sinumerik
SINUMERIK 840D powerline	From 09.2001 onwards, the SINUMERIK 840D powerline and SINUMERIK 840DE powerline will be available with enhanced performance. See the hardware description below for a list of the available powerline modules: References: /PHD/, Configuring Manual SINUMERIK 840D
SINUMERIK 810D powerline	From 12.2001 onwards, the SINUMERIK 810D powerline and SINUMERIK 810DE powerline will be available with enhanced performance. See the hardware description below for a list of the available powerline modules: References: /PHC/, Configuring Manual SINUMERIK 810D
Standard scope	This Operating/Programming Guide describes only the functionality of the ManualTurn user interface. A description of add-on features or modifications made by the machine builder are not included in this guide. For more detailed information on SINUMERIK 840D/810D publications and other publications covering all SINUMERIK controls (e.g. universal interface, measuring cycles...), please contact your local Siemens office.

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

Basis

Your SIEMENS 840D/810D with ManualTurn is state of the art and is manufactured in accordance with recognized safety regulations, standards and specifications.

Add-on equipment

Using special add-on equipment and expanded configurations from SIEMENS, SIEMENS controls can be adapted to suit your specific application.

Personnel

Only **authorized and reliable personnel with the relevant training** must be allowed to handle the control. Nobody without the necessary training must be allowed to work on the control, not even for a short time.

The **responsibilities** of the personnel employed for setting, operating and maintenance must be clearly **defined** and **supervised**.

Behavior

Before the control is started up, it must be ensured that the Operator's Guide has been read and understood by the personnel responsible. The operating company is also responsible for **constantly monitoring** the overall technical state of the control (faults and damage apparent from the outside and changes in response).

Service

Repairs must only be carried out in accordance with the information given in the Service and Maintenance Guide by **personnel trained and qualified** in the relevant field. The relevant safety regulations must be observed.

The following is **contrary to the intended purpose** and **exonerates the manufacturer from any liability**:

- Any use whatsoever beyond or deviating from the application stated in the above points.
- If the control is **not in perfect technical condition**, or is operated without awareness for safety or the dangers involved or without observing the instructions given in the instruction manual.
- If faults that can reduce safety are not remedied **before** the control is started up.
- Any **modification, overriding or deactivation** of equipment on the control used for the perfect functioning, unrestricted use or active and passive safety.



This can result in **unforeseen dangers** for:

- the health and life of people,
- the control, machine and other property of the operating company and user.

Structure of the document



The following information blocks marked by symbols are used in this document:



Function



Sequence of operations



Explanation of parameters



Additional notices



Software option

The function described is a software option, i.e. the function is only executable on the control if you have acquired that option.

Warnings



The following five warnings are used with graded severity.



Danger

Indicates an imminently hazardous situation which, if not avoided, **will** result in death or serious injury or in substantial property damage.



Warning

Indicates a potentially hazardous situation which, if not avoided, **could** result in death or serious injury or in substantial property damage.



Caution

Used with the safety alert symbol indicates a potentially hazardous situation which, if not avoided, **may** result in minor or moderate injury or in property damage.

Caution

Used without safety alert symbol indicates a potentially hazardous situation which, if not avoided, **may** result in property damage.

Notice

Used without the safety alert symbol indicates a potential situation which, if not avoided, **may** result in an undesirable result or state.

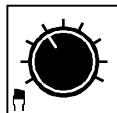
Reference to other literature

This marking appears wherever specific information can be found in more detailed reference literature.

References:

The Appendix in this Operator's Guide contains a complete list of references.

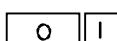
The following symbols are used for the operating elements:



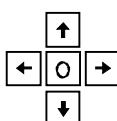
Selection of an operating mode

Soft key

Selection via soft key



Feed Stop/Start keys



Axis/direction selection, e.g. using control stick



JOG keys

Introduction

1.1	The ManualTurn product.....	1-14
1.2	Operator notes	1-15
1.3	Switching on/switching off	1-16

1.1 The ManualTurn product

The ManualTurn product with SINUMERIK 840D or SINUMERIK 810D is a CNC (Computerized Numerical Control) for turning machines used predominantly for conventional machining operations. The 810D is simple and reliable to operate so as to facilitate the task of the skilled machinist. All inputs are made in plain text in interactive dialog and displayed graphically for checking purposes, i.e. the lathe operator can examine the motional path of the tool before he starts the program.

The operator panel of the CNC allows you to implement the following basic functions (in conjunction with a machine lathe):

- Setting up and conventional turning with handwheels
- Longitudinal and taper turning, facing with feedrate per revolution and per minute
- Execution of finishing or roughing cuts on elementary contours
- Machining with cycles in single-cycle mode
- Creation of complex contours with the option of removing stock and finishing against the contour (option)
- Creation of parts programs for complete machining using EasyStep programming.
- Automatic creation of parts programs in the Teach In operating mode.



It is advisable to read Chapter 2 "Operation" carefully before working through the other sections.

All further sections are written on the premise that you have read and understood Chapter 2!

1.2 Operator notes



Caution

The operator panel/machine control panel may only be opened for servicing purposes by properly qualified personnel.



Danger

Fatal injury may occur if the operator panel/machine control panel is opened when the power supply is still connected.



Warning

Electronic components inside the operator/machine control panel may sustain irreparable electrical damage if they are not handled in the correct manner.



Before you touch any control elements on this operator panel:
Please read through the explanations in this document carefully!

1.3 Switching on/switching off**Function****Switching on**

There are various methods by which the control system or the entire installation can be switched on.



It is very important to read the information supplied by the machine manufacturer!

A power-up display specific to the machine manufacturer appears several seconds after the control has powered up.

Switching off

Before you switch off the control system or the entire installation, please remember:



It is very important to read the information supplied by the machine manufacturer!

Operation

2.1	Operator panels.....	2-18
2.2	Machine control panel	2-21
2.3	Mini handheld unit	2-25
2.4	Graphics interface	2-27
2.5	Operating system	2-29
2.5.1	Operating modes.....	2-30
2.5.2	Important function keys.....	2-32
2.5.3	Important soft keys.....	2-33
2.5.4	Pocket calculator.....	2-34
2.5.5	Absolute and incremental dimensions	2-36
2.5.6	Angle reference system	2-37
2.5.7	Tool and cutting data.....	2-38

2.1 Operator panels

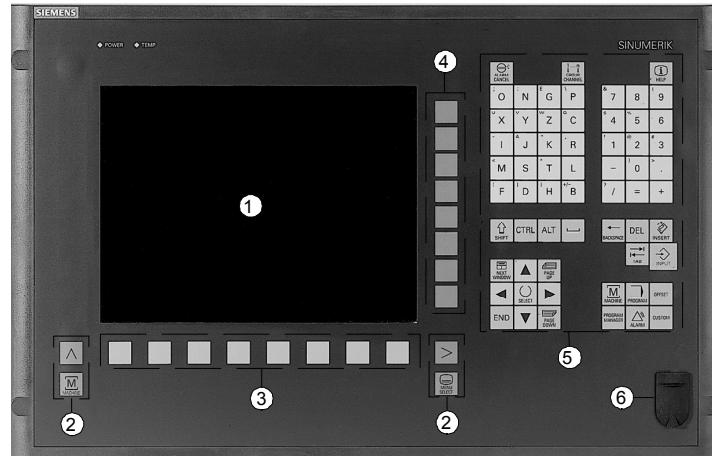
Alternately, you can use one of the following operator panels for the PCUs:

OP 010

OP 010C

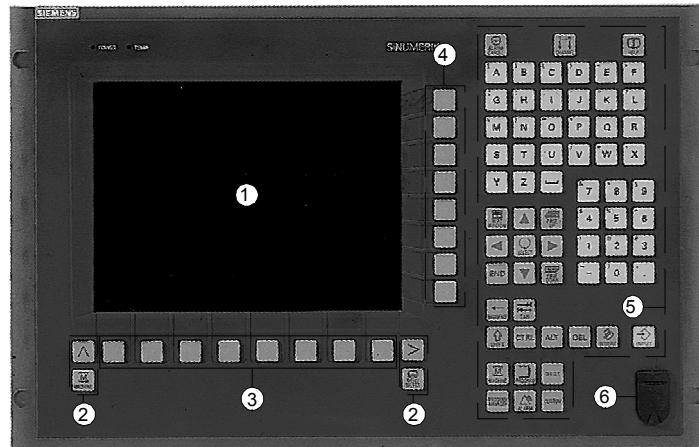
OP 010S with CNC full keyboard OP 032S

Operator panel OP 010

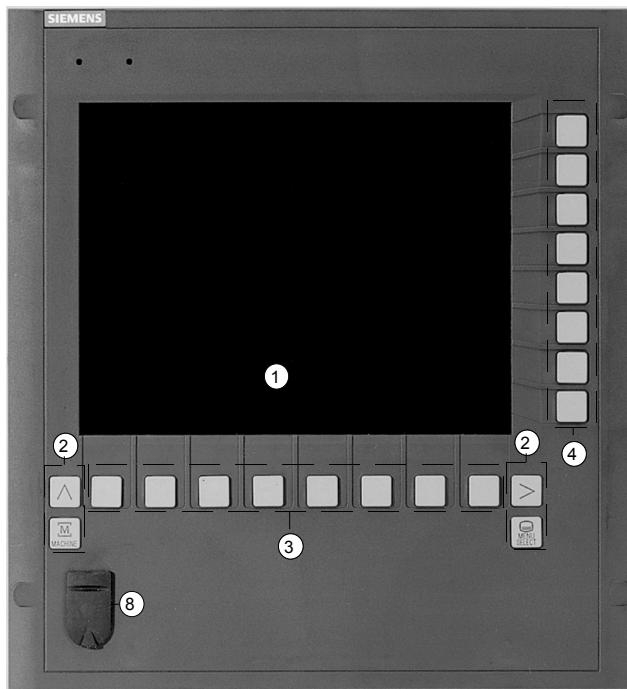


Operator panel OP 010

- 1** Monitor
- 2** Monitor keys
- 3** Horizontal soft key menu
- 4** Vertical soft key menu
- 5** Alphanumeric pad
Correction/cursor pad with control keys and input key
- 6** USB interface

Operator panel OP 010C*Operator panel OP 010C*

- 1 Monitor
- 2 Monitor keys
- 3 Horizontal soft key menu
- 4 Vertical soft key menu
- 5 Alphanumeric pad
Correction/cursor pad with control keys and input key
- 6 USB interface

Slimline operator panel**OP 010S***Operator panel OP 010S***With CNC full keyboard****OP 032S***CNC keyboard OP 032S*

- 1** Monitor
- 2** Monitor keys
- 3** Horizontal soft key menu
- 4** Vertical soft key menu
- 5** Alpha pad
- 6** Correction/cursor pad with control keys
- 7** Numeric pad
- 8** USB interface

2.2 Machine control panel

General notes

Operations on the machine tool such as axis traversal or program start can only be initiated via a machine control panel.

The machine control panel is configured and supplied by the machine tool manufacturer.

Please refer to the operating manual supplied by the machine tool manufacturer for details of which panel control elements are required for your application and a description of their functionality.

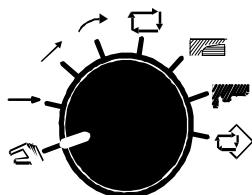
Example

The following description is based on an example configuration.

Operating modes

Depending on your requirements, operating modes **MANUAL**, **STRAIGHT**, **CONICAL**, **CIRCLE**, **CYCLE**, **STOCK REMOVAL**, **CONTOUR** and **PROGRAM** can be activated via

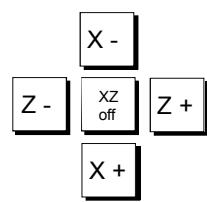
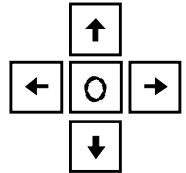
- an operating mode switch or
- the vertical soft key menu on the operator panel or
- illuminated keys.



Traversing directions

Control stick with rapid traverse key

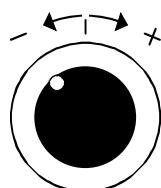
The control stick allows axes X and Z to be traversed paraxially and at angles of 45°. The control stick is active in Setup mode and in the above mentioned operating modes.



- Illuminated keys.

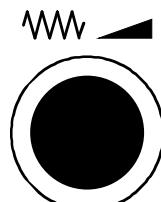
As an alternative to the control stick, you can also use the illuminated keys to select the traversing direction.

The traversing velocities can be selected by means of a fixed feedrate and feed key, whereby the preselected feed axes traverse for as long as the JOG key is pressed. The set working feedrate can be substituted by this function.



Contour handwheel

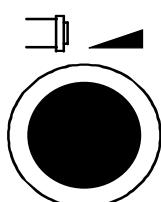
When the contour handwheel function is activated, the handwheel controls the feedrate along a programmed contour.



Feedrate override switch

The feedrate override switch can be used as required to make fine feedrate adjustments to suit the machining process.

The feed control is displayed as a percentage in the status field.



Spindle control

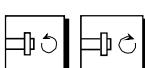
- Spindle speed override switch

The speed override switch is used to change the speed or peripheral speed during machining within speed limits for the selected gear step. The new value is displayed.



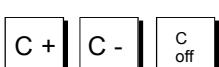
- Keys

The programmed spindle speed S (corresponds to 100%) can be decreased/increased with Spindle -/Spindle +.



Spindle counterclockwise/clockwise rotation

These keys start the spindle in the desired direction of rotation.



Illuminated key "C axis"

This key selects the rotational direction of the C axis. The selection is disabled again by means of the traversing direction key "Off".



Illuminated keys "Spindle start" and "Spindle stop" key

This key is for starting the spindle.

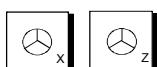


This key is for stopping the spindle.



Illuminated key "Incremental dimension On/Off"

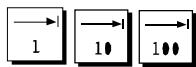
The incremental dimension display of the control system is selected/deselected with this key.

**Illuminated keys "Handwheels X, Z On/Off"**

These keys enable/disable the handwheel functions for the X and Z handwheels.

**Illuminated pushbutton "Contour handwheel On/Off"**

This key switches the contour handwheel on and off.

**Illuminated keys "Traverse by handwheel"**

The handwheel factor is set with keys 1, 10 and 100.

**JOG keys for fixed feedrates**

Fine traverse/creep feed/moderate traverse/rapid traverse button
When an operating mode is active, axes are not traversed at the programmed feedrate but at a fixed feedrate setting (override has no effect).

When an operating mode has been interrupted or not started at all, these keys act as JOG keys for the feed or C axis. The travel direction is selected with the control stick or the illuminated key for the C axis.

**JOG key for feedrate**

When an operating mode has been interrupted or not started at all, the feed key acts as a JOG key for the feed or C axis. The override is active. The travel direction is selected with the control stick or the illuminated key for the C axis.

**Illuminated key "Cycle Start/Stop"**

You use the Start key to activate the function selected via the operating mode switch, e.g. single positioning step or complete machining cycle.



The Stop key can be pressed to halt a motion in progress.

The keys light up correspondingly to indicate the current operational status. The possible operational states are listed below:

- **No key illuminated**

The selected operating mode has not been started. You can select another operating mode or start setup.

- **Start key illuminated, Stop key not illuminated**

The displayed operating mode has been started. The axes move according to the way they have been selected or programmed. Setup is not possible.

- **Stop key illuminated**

The displayed operating mode has been started, but the motion sequence has been interrupted. Setup is possible. You can continue an interrupted movement by pressing the Start key.



"Plus/Minus direction" keys

You can use these JOG keys for traversing in plus or minus direction along the contour if the contour handwheel is activated.



Key "TEACH feed"

When this key is actuated, a manually approached position is transferred to the TEACH IN memory as a feed block (G01).



Key "TEACH rapid traverse"

When this key is actuated, a manually approached position is transferred to the TEACH IN memory as a rapid traverse block (G00).



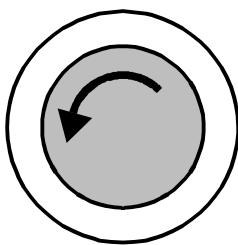
Single-step mode key

By activating the single-step mode key, you can select/deselect single-step mode in the PROGRAM operating area.



RESET key

You can cancel a program with the RESET key.



Emergency stop key

This red key must be actuated in emergency situations, i.e.

1. when human life is at risk,
2. when there is a risk of damage to the machine or workpiece.

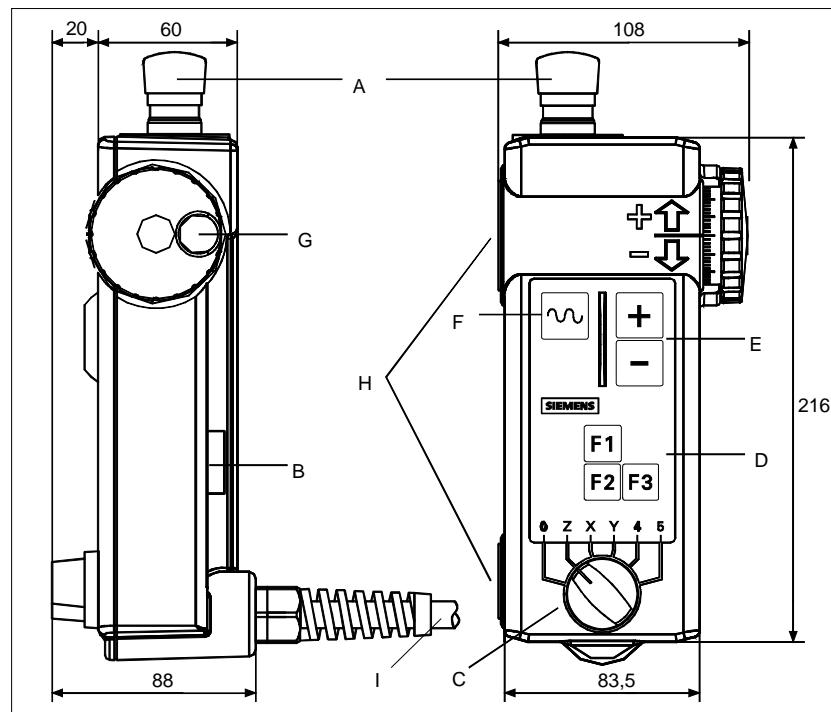
Generally speaking, an EMERGENCY STOP command shuts down all drives with the highest possible braking torque in a controlled manner.



For further or different reactions to EMERGENCY STOP:

See data supplied by machine manufacturer!

2.3 Mini handheld unit



- A EMERGENCY STOP button, two-channel
- B Enable key, two-channel
- C Axis selection key for 5 axes and neutral position
- D Function keys F1, F2, F3
- E Traversing keys direction +, -
- F Rapid traverse keys for fast traversal with traversing keys or handwheel
- G Handwheel
- H Magnetic clamps to attach to metal parts
- I Connection cable 1.5m ... 3.5m

Control elements

EMERGENCY STOP button

The EMERGENCY STOP button must be activated in the event of an emergency

1. when human life is at risk,
2. when there is a risk of damage to the machine or workpiece.

Enable key

The enable key is designed with two positions. It must be pressed to enable triggering of traversal movements.

Axis selector switch

You can select up to five axes with the axis selector switch.

Function keys

You can trigger machine-specific functions with the function keys.

Traversing keys

By activating traversing keys +, – you can initiate traversal movements on the axis selected with the axis selector switch.

Handwheel

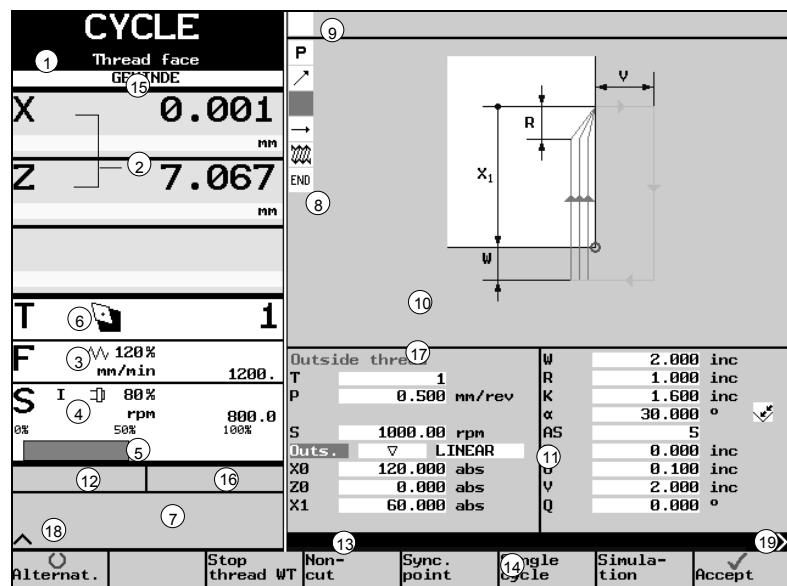
By activating the handwheel, you can initiate traversal movements on the axis selected with the axis selector switch. The handwheel returns two track signals with 100 l/rev.

Rapid traverse key

The traversing velocity of the axis selected via the axis selection key can be increased by means of the rapid traverse key. The rapid traverse key affects the travel commands of the +/- keys, as well as the handwheel signals.

2.4 Graphics interface

Screen layout



Explanation of display elements

- 1 Name of the operating modes:
MANUAL, STRAIGHT, TAPER, CIRCLE, CYCLE, STOCK REMOVAL, CONTOUR, PROGRAM together with any applicable submenu; applies to CYCLES and CONTOUR only (e.g. Thread face, Undercut, Stock removal)
- 2 Position displays
- 3 Feed display
- 4 Speed display with rotational direction
- 5 Output display
- 6 Tool data
 - Tool number
 - Tool position
- 7 Status field:
This field contains the following information depending on the current machining situation:
 - Test run
 - TNRC left, TNRC right (TNRC=tool nose radius compensation)
 - Dwell
 - Ack aux. command (acknowledge auxiliary command)
 - Travel command
 - Manual offs. (manual offset)
 - Current zero offset
 - Data trans. (data transmission)

- 8 Machining sequence of programmed steps
 - EasyStep sequence
 - Special commands (dwell time, comment, etc.)
- 9 Current block or status line
- 10 Graphic display area

When you press the information key, you switch between

 - the EasyStep flowchart and contour display in PROGRAM operating mode
 - in the other operating modes, between contour display or direction arrow and help display (if available).
- 11 Parameter input field
- 12 Status field for alarms and messages
- 13 Dialog line
- 14 Horizontal soft key menu with eight soft key functions
- 15 Program name
- 16 Teach-in steps displayed (e.g. Teach1, etc.)
- 17 Cursor text display: Plaintext for the parameter underneath the cursor
- 18 Recall: Return jump to the higher-level menu
- 19 ETC: Extended soft key menu

2.5 Operating system

General notes

Options for manual intervention as well as step-by-step machining of lathed parts are the principle features of turning machines controlled in the conventional manner (X, Z; one spindle).

With the ManualTurn system, you enter travel commands in plaintext via simple input screen forms in a graphics-assisted dialog.

The following machining modes are at your disposal:

- MANUAL
- STRAIGHT
- TAPER
- CIRCLE
- CYCLES
- STOCK REMOVAL
- CONTOUR
- PROGRAM

Machining possibilities

With ManualTurn, workpieces can be machined as follows:

- conventionally with single-cycle machining
- automated with step chain programming using EasyStep

Single cycle machining

You can parameterize the above modes (except for MANUAL and PROGRAM) as a single cycle and process them immediately with NC start, i.e. you can create a contour and then cut without having to create an entire EasyStep program.

Prerequisite for single-cycle machining is that no program is selected. An active program can be deselected by activating the "Program ON" soft key (PROGRAM mode). The "Accept" soft key is then no longer available in the single cycles.

Step chain programming with EasyStep

When generating an EasyStep program, each single cycle/individual element is created as a separate step in a machining chain (step chain) by "accepting" the parameters.

Each step is stored on one line and consists of the entered parameterization data with the associated element-specific icon.

The completed machining sequence can be modified later.

Once all the machining sequence parameters have been set, the NC start key can be pressed to execute the sequence.

Under the Directory menu, PROGRAM mode offers a program management facility in which you can store the machining sequences you have created.

2.5.1 Operating modes



MANUAL

"Manual" mode in this case means conventional feed mode travel for longitudinal and taper turning and as well as facing. The travel direction is determined by the position of the control stick. The reference points of the machine can also be approached in this mode.



STRAIGHT

Longitudinal turning and facing with automatic shutdown on arrival at the specified target position. The C axis can also be traversed in this mode.



TAPER

Taper turning; the taper can be defined in three different ways. The C axis can also be traversed in this mode.



CIRCLE

Radius machining; circular movements can be defined in three different ways.



CYCLE

The CYCLE machining mode provides you with DIN-based cycles for threads, undercuts and drilling operations in the form of simple parameterization displays.

The machine manufacturer may have added other special customized cycles to this group.

The program management function allows you to store your thread and thread undercuts and call them up again whenever you need them.



STOCK REMOVAL

Stock removal mode allows machining operations based on special stock removal and grooving cycles. The machine manufacturer may also incorporate special customized cycles.



CONTOUR

In CONTOUR mode you can create and cut free contours, as well as remove residual material.

With the program management function you can store contours and call them up again whenever you need them.



PROGRAM

The purpose of the EasyStep function is to allow users with no G-code programming knowledge to create a complete, executable program by chaining individual elements such as the following in a step-by-step process:

- Straight, chamfer, circle, cycles
(e.g. thread, cutting, contour cycles)
- Special commands (e.g. tool change, auxiliary commands, dwell times, programmed stop, tool nose radius compensation, zero offsets and comments).

2.5.2 Important function keys



Information key

With this key you switch the display area

- between the help displays (if available) and contour display or direction arrow in operating modes STRAIGHT, CONICAL, CIRCLE, CYCLES, STOCK REMOVAL and CONTOUR
- in the EasyStep flowchart and contour display in PROGRAM operating mode
- in MANUAL mode between help menus via the most important soft keys and function keys.



Message overview key

With this key you can activate an overview display of all active messages.

You exit this overview by selecting the message overview key again or by selecting the Recall key.



Operating area switchover key

With the operating area switchover key you can switch over to standard CNC operation.

Please follow the advice of the machine manufacturer.



Toggle key

If an input field offers several parameter setting options, you switch between them with the toggle key.



Home key

This key moves the cursor to the beginning of the line on the opened page in input fields or the editor window.



Edit key

With this key you can activate the calculator function when the cursor is positioned on an input field.



Recall key

Return jump to the higher-level menu. The Recall key closes a window without transferring edited values to the system.



"ETC" key

An extended soft key menu is displayed in every operating mode when you press the "ETC" key (see Section "Important soft keys").

2.5.3 Important soft keys



The following soft keys are incorporated into ManualTurn. Each of them has the same function and screen position whatever the operating mode.



General soft key functions



"Alternat." soft key

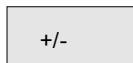
This soft key appears when the cursor is positioned on an input field with several parameter setting options.

Examples of these are as follows:

- Feed F in mm/min, in mm/rev or in rapid traverse
- Spindle speed S in rev/min or m/min

Please note: Speed in m/min only applies in conjunction with feedrate in mm/rev (constant cutting speed)

- Absolute and incremental programming for axis motions (X, Z)
- Clockwise/counterclockwise direction of rotation
- Position of the coordinate system and machining types (roughing, etc.) for cycles



+/- soft key

You can change the sign of the value on which the cursor is positioned with this soft key.



Accept soft key

The soft key Accept is displayed with all parameter screens that can be transferred to an EasyStep machining sequence or a contour. The parameters are integrated as a step in the sequence with this soft key.



"OK" soft key

The parameters entered are accepted and saved.



"Back" soft key

Return jump to the higher-level menu.



"Abort" soft key

Return jump to the higher-level menu. The Abort key closes a window without transferring the entered values to the system.



Extended soft key menu

You can display the extended soft key menu incorporating the following soft keys in all modes by pressing the "ETC" key:

	For defining the control zero in the machine coordinate system; inch/metric changeover
	Menu with inputs for tool offsets/wear and select/deselect tool
	Menu for additional spindle inputs
	Menu with display of functions: Zero point and manual offsets
	Run diagram to trace a program as it runs
	Selection and deselection of Teach-in function

2.5.4 Pocket calculator



Function

The calculator function is activated by pressing the edit key. Any number of calculations can then be performed using the four basic arithmetic operations (+, -, *, /) and the equal sign (=). The value you enter is calculated with the previously entered value.



You can terminate the calculation with the input key which also closes the input field.

Example 1

Starting position:

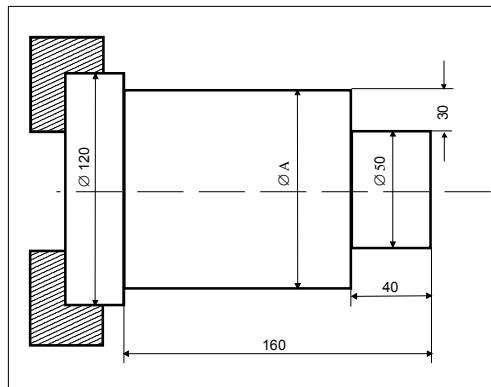
Number	X	Z
1	0.050	0.000



Open input field X using the edit key and enter value +0.1.

Result:

Number	X	Z
1	0.150	0.000

Example 2**Calculate diameter A**

Open input field X with Edit key



Position cursor at the end of the input field



Delete old value with cursor

Enter this example: $2 * 30 + 50$

Result:

X	110.000 ABS
---	-------------

2.5.5 Absolute and incremental dimensions



Function

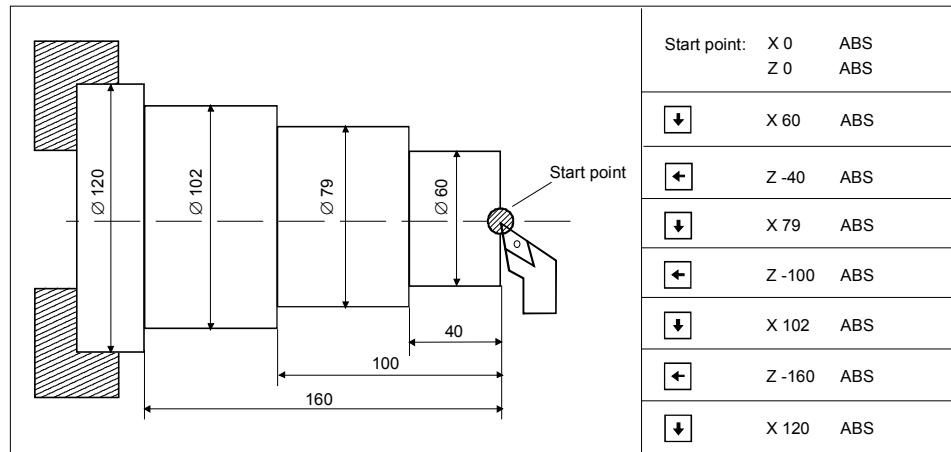
You can define the traversing movements of the tool in incremental and absolute dimensions.

You can mix incremental and absolute dimensions, i.e. one coordinate in incremental dimensions, the other in absolute dimensions.

Absolute dimension (ABS)

Absolute dimensions refer to the workpiece zero.

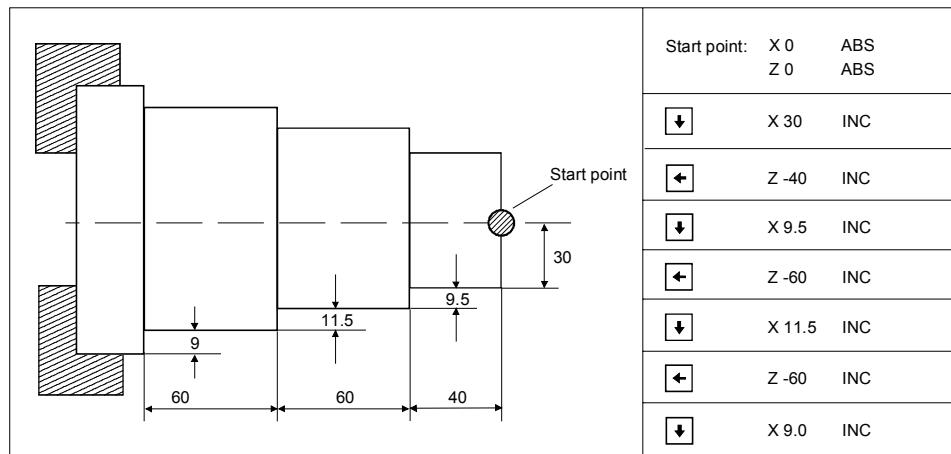
Example



Incremental dimension (INC)

Incremental dimensions refer to the momentary position.

Example



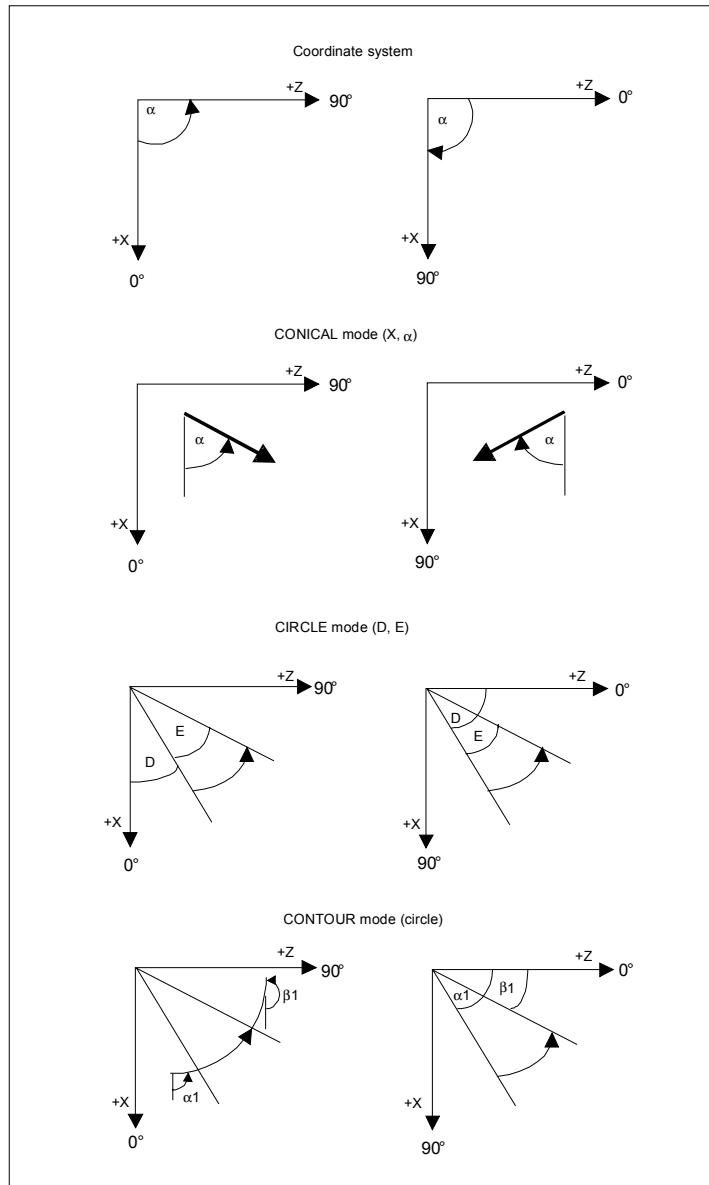
2.5.6 Angle reference system



Function

With the elements CONICAL, CIRCLE and CONTOUR, the angle specifications can refer to both the 1st geometry axis (Z) and the 2nd geometry axis (X). The direction of rotation changes, however.

Examples



The angle reference system is set up via machine data. Please refer to your machine manufacturer's specifications for information on which angle reference system is used on your machine.

2.5.7 Tool and cutting data

The main tool and cutting data are as follows:

- Tool **T**
- Feedrate **F**
- Spindle speed **S** or pitch **P**

Parameter input field

With the exception of the parameter **P** for the thread lead (only in the "Thread" cycle), the above mentioned parameters are displayed in the parameter input fields with all turning cycles (in MANUAL, STRAIGHT, CONICAL, CIRCLE, CYCLE, MACHINING and CONTOUR modes).

Display window

The display windows for tool, feed and spindle speed showing the current information about the machine state are located underneath the actual-value display.

These are defined with the following address letters and units in the parameter input field.

T

Set this parameter to define the tool that you wish to use for the respective turning cycle.

If you change the tools on your machine manually, you must enter the number of the required tool from the tool table.

Example: Select tool 2 from the table |T

If your machine is equipped with a turret and tools are changed automatically, you must enter a 3 or 4 digit number. The first 1 or 2 digits correspond to the turret number and the last two digits to the number of the tool from the tool table. You must always enter the tool number as a 2-digit number.

Example: Select tool 2 from the table in turret location 6

|T

With the STRAIGHT, CONICAL and CIRCLE modes, tool selection is modal, that is, if several cycles with the same tool occur in sequence, it is necessary to program a tool for the 1st cycle only. The T field in the input screens for the following cycles remains empty.

The currently active tool (as tool number) and the tool point direction (as symbol) are displayed in the **tool window**.

- F** This parameter shows the feed either in mm/rev or mm/min. With "Hole circle drilling" using an external spindle, the feed can only be specified in mm/min. With STRAIGHT and CONICAL modes, you can also select rapid traverse. A straight or conical programmed with rapid traverse is displayed as a broken-line arrow in the program graphics.
- P** With the "thread cutting" cycle, you can specify the thread lead for **P** either in mm/rev, inch/rev, module and in threads."/".
- The **feedrate window** displays the setpoint and actual-value of the feedrate, as well as the position of the feedrate correction switch. The setpoint that must actually be traversed depends on the feedrate correction switch.
- S** Parameter **S** can be set to one of the following spindle speed units: m/min (constant cutting speed) and in rev/min. If the machine manufacturer has included the feature, you can also accept a constant cutting rate defined in the tool table with the selection S(T).
- The **spindle window** displays the setpoint and actual values of the spindle speed, the position of the spindle, the position of the spindle correction switch and the spindle power.

Notes

Preparatory Functions for Machining

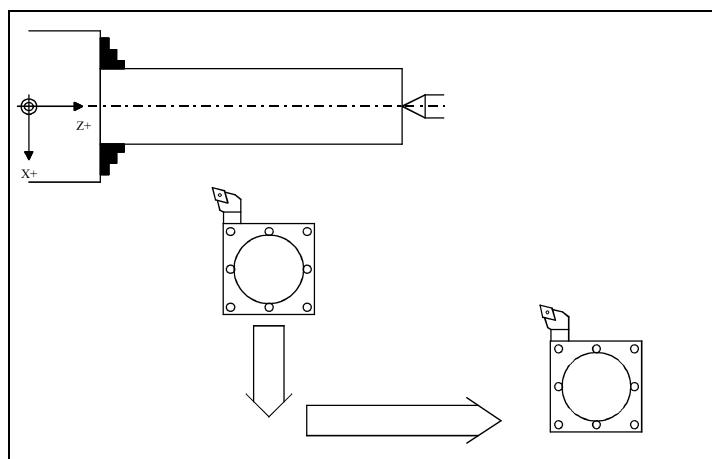
3.1	Approach reference points	3-42
3.2	Setup	3-44
3.3	Incremental feed mode	3-45
3.4	Offsets	3-47
3.4.1	General notes	3-47
3.4.2	Preset	3-48
3.4.3	Manual offset	3-49
3.4.4	Delete manual offset	3-50
3.4.5	Zero offset	3-51
3.5	Spindle speed limitation	3-52
3.6	Oriented spindle stop	3-53
3.7	C axis mode	3-54
3.8	Tool	3-55
3.8.1	Enter tool offset data	3-55
3.8.2	Selecting/deselecting tool offset	3-57
3.8.3	Measure tool	3-58
3.8.4	Tool wear compensation	3-60
3.9	Measuring system changeover inch/metric	3-61

3.1 Approach reference points**Function**

The axes will need to be referenced depending on how the turning machine is configured and the requirements of the application. For this purpose, the axes are traversed to a certain position that the control system recognizes. After the axes have been referenced, the control knows the position of the machine working point.



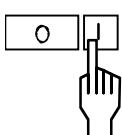
Any previously set reference points are re-approached when the control is switched on again.

**Sequence of operations**

Select MANUAL mode (via soft key or operating mode switch)

Ref.
point

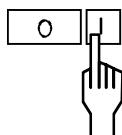
Select soft key labeled "Ref. point"



Activate reference point approach for X axis by pressing NC Start key



Select parameter "Z ref. pnt" by means of soft key labeled "Alternat."



Activate reference point approach for X axis by pressing "NC Start" key



Before starting the reference approach process, check that the reference points can be approached without risk of collision. Approach a suitable start position in setup mode if necessary.

You can stop the movement at any time by pressing the Stop key. The Start lamp goes out when the relevant reference position has been reached. The machine position is then set to reference dimension, i.e. synchronized with the machine.

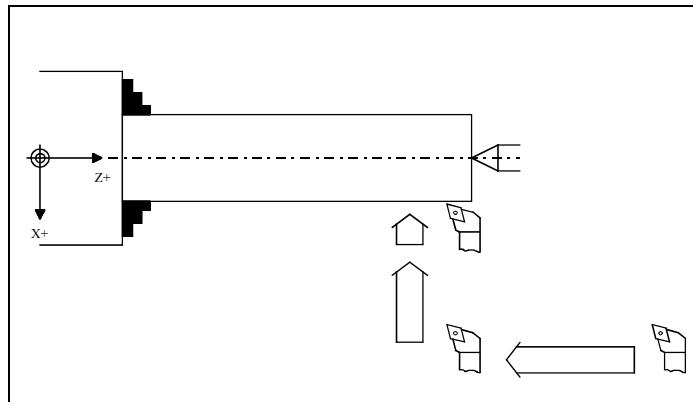
3.2 Setup



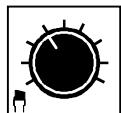
Function

In setup mode, you move the axes with

- the direction and JOG keys or
- the handwheels.



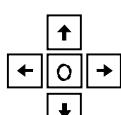
Sequence of operations



Any operating mode



Start lamp is off or Stop lamp is on
(axes are not in motion)



Select axis direction



Trigger a movement with the JOG keys, e.g. with rapid traverse



Rapid traverse



Normal speed



Creep speed



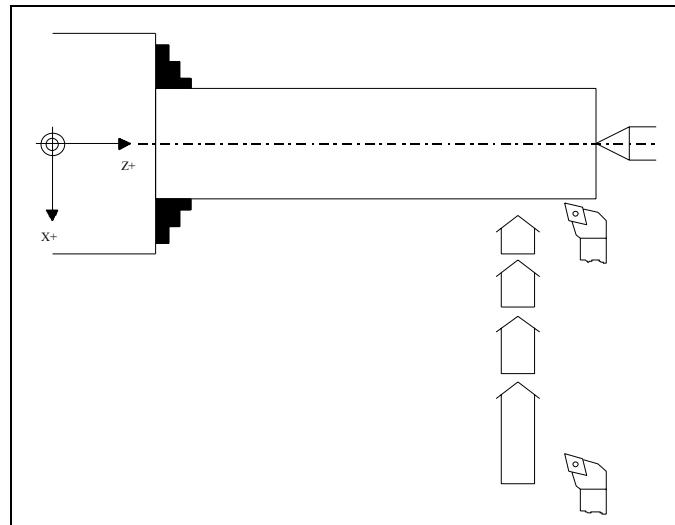
Fine speed

3.3 Incremental feed mode

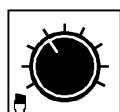


Function

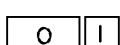
In incremental feed mode, the axes of the machine tool are traversed by a set incremental distance in the selected direction.



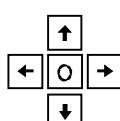
Sequence of operations



Any operating mode



Start lamp is off or Stop lamp is on
(axes are not in motion)



Select axis direction, e.g. X



Key labeled "Increment" (incremental function is displayed)



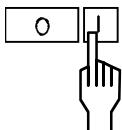
Select increment size (0.001-0.01-0.1-1-10) with soft key labeled "Back" or "Forward", e.g. 0.1mm

Alternative

Enter incremental dimension, e.g. 0.175.



If necessary, use the cursor keys to select the infeed speed and overwrite the value.



Activate incremental feed by pressing NC Start key.

If required, repeat the process with the same or different values.

To deselect incremental feed mode, select the key labeled "Increment" again.

You can interrupt axis feed at any time by pressing the Stop key or abort the process by deselecting the axis direction.



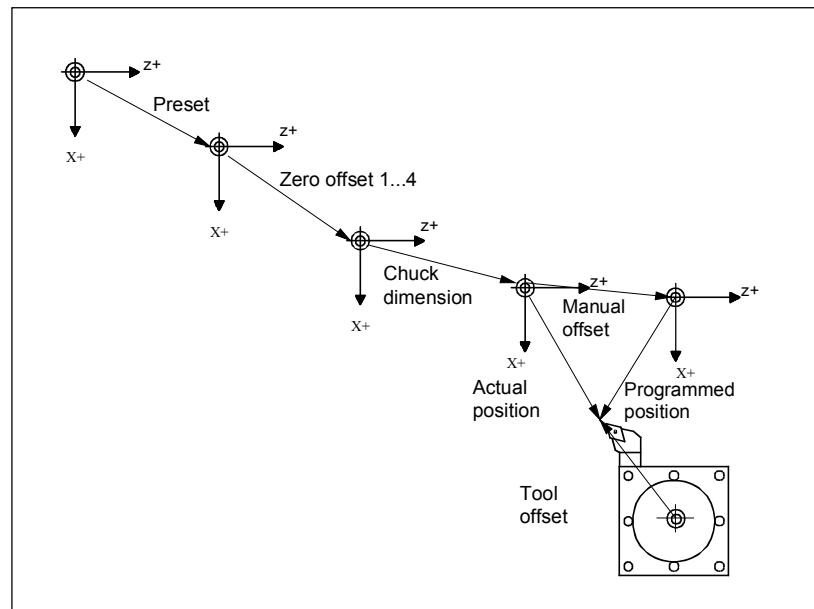
3.4 Offsets

3.4.1 General notes



Overview

Function

**Preset**

You can redefine the reference point for the axis movements and the actual value display with the function "Preset".

Zero offset N1...N4

You can numerically enter four zero offsets. If required, you can then activate one of the zero offsets.

Manual offset

A manual offset is automatically stored if the axes are additionally traversed using the handwheel during a cycle or if an offset is stored during an interruption.

Chuck dimension

Chuck dimension is an additive zero offset and refers to the zero offset currently set.

Tool offset

The tool offset has the same effect as a zero offset.

3.4.2 Preset**Function**

This function can be used, for example, to set the X axis to a specific (measured) diameter or the Z axis to the desired value (after slight contact with the plane surface). The preset values act on machine axes. The axes are not moved in the course of a "Preset".

**Sequence of operations****Preset**

Select soft key labeled "Preset" in the extended soft key menu of any operating mode.

Enter Preset

Select the axis position, e.g. "Z", in the position display by means of the cursor keys.

Type in position value, e.g. -100.000 and transfer to system with "Input" key.

Z=0

You can set the Z axis position display to 0 via the soft key "Z=0".

Delete Preset**Delete Preset**

You can cancel the preset offset again via the menu "Offset" with the soft key labeled "Delete Preset".



Preset is an available option if no start command has been given.

3.4.3 Manual offset



Sequence of operations

Precondition

You have started a cycle, elementary contour or EasyStep program.

1. Manual offset via handwheel

- Activate Handwheel key
- Traverse with handwheel

"Manual offs." appears in the status field to inform the operator that a manual offset is active. The offset is included in the actual position display.



Additional notes

Programmed absolute positions are traversed relative to the manual offset.

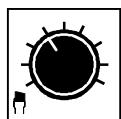
Example

- Straight line in Z programmed with: 100mm (abs)
Manual offset in Z active: 5mm
- At the beginning of the straight line
Actual position display, e.g. 35mm (incl. manual offset)
Distance to go 70mm
- After the straight line has been traversed
Actual position display 105mm
Distance to go 0mm



2. Storing the manual offset via Offset

(see Section "Intervention in the Machining Process")

3.4.4 Delete manual offset**Sequence of operations**

Select any operating mode



Select the extended soft key menu by pressing the ">" key

Select the soft key labeled "Offset" in the soft key menu.
All position data are then displayed on the screen.

Select soft key labeled "Delete man. offs."



The function is not executed.

or



The function is executed.



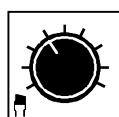
Offset can be deleted if no Start command has been given.

3.4.5 Zero offset



Sequence of operations

Reselect zero offset



Select any operating mode



Press the soft key labeled "Offset" in the extended soft key bar



and select a new zero offset with the relevant soft key

"> <" marks the zero offset you have activated

Deselect zero offset



Press the soft key for the selected zero offset again.

The zero offset is deselected.

Input zero offset



Select the soft key labeled "Offset" in the extended soft key menu in any mode.



Use the cursor keys to select a zero offset, e.g. ZO1

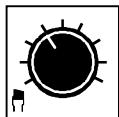
Enter value for "X" (e.g. 100) and "Z" (e.g. 200).

Confirm your settings with "Input".

Enter other data if required.

You can enter data if no start command has been given.



3.5 Spindle speed limitation**Sequence of operations**

Select any operating mode



Select the extended soft key menu by pressing the ">" key



Spindle

Select the soft key labeled "Spindle"

In this menu you can

- define a speed **S**, either in rev/min or m/min
- define a max. spindle speed in rev/min
- enter a speed limitation for the constant cutting speed (m/min) in rev/min.

The constant cutting speed is also displayed as a set speed in rev/min.

**V = const
act. val.**

The "V=const act. val." soft key displays the spindle speed in place of the cutting rate in the spindle display window if the cutting rate is programmed in the machining steps.

3.6 Oriented spindle stop



Function

When the oriented spindle stop function is active, the spindle is always stopped in a preselected position (e.g. position for chuck wrench).



Spindle

Sequence of operations

Select the soft key labeled "Spindle" in the extended soft key menu in any mode.

Enter oriented stop position,
e.g.: 11.0°



Confirm your selection by pressing the "Input" key

Orient ON

Press the "Orient ON" soft key (if necessary, selectable with separate switch on machine).

Stop spindle.

The spindle stops at the preselected position.

3.7 C axis mode



Function

The spindle must be operated as a C axis if the spindle is to be used for positioning and machining (e.g. for drilling and hexagonal milling). In C axis mode, the spindle is operated as a position-controlled rotary axis. It can then be operated in interpolation with other axes.



Notes

Display C always

- The traversing range in C axis mode ranges from 0° to 359.999° (absolute), or ± 99 999.999 (incr.).
- You select the display of the C axis position with the "Preset" menu in the extended soft key menu and the "Display C always" soft key. The position of the C axis appears in the position display. The C axis is also offered in the appropriate operating modes (e.g. STRAIGHT, CONICAL, CIRCLE) as a further input parameter when programming.

Example

Programming C-axis mode

To program C-axis mode under EasyStep programming, please proceed as follows:

Precondition

PROGRAM mode is selected.

Press the "Special function" and "Auxiliary commands" soft keys.

Set oriented spindle stop and the STOP position, e.g. 0°.

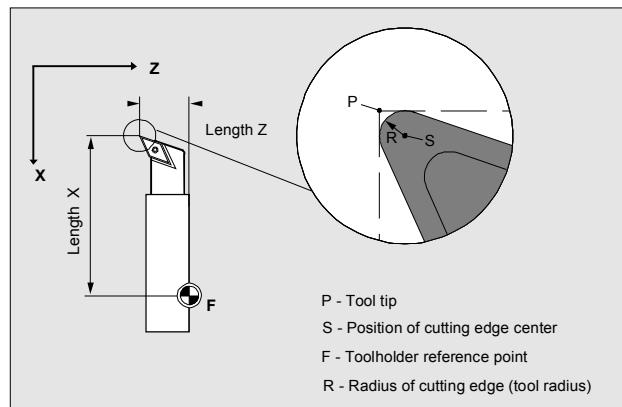
Validate the setting by pressing the soft key "Accept".

Program the end position (100°) of the C axis by means of the "C" soft key in STRAIGHT mode,
e.g. C = 100

Exit the input screen by pressing the "Accept" soft key.

3.8 Tool

3.8.1 Enter tool offset data



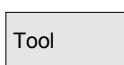
Sequence of operations



Select any operating mode.



Select the extended soft key menu by pressing the ">" key.



Select the soft key labeled "Tool".



Select the soft key labeled "Tool offset".

A table is now displayed into which you can enter the tool offset data for your tools.

Example

Number	Length X	Length Z	Radius	Pos.	$v = \text{const}$ m/min
1	127.830	24.580	0.200	◀	200.000
2	142.620	24.330	0.400	▶	250.000
3	260.450	24.880	0.500	▼	300.000
4	0.000	350.000	15.000	▲	0.000
5	0.000	300.000	10.000	✎	0.000



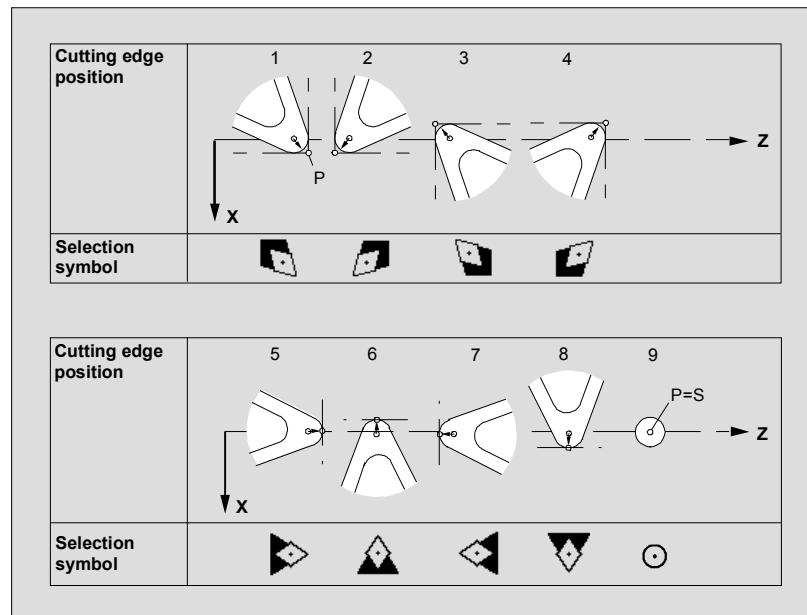
Use the cursor keys to select the tool number you want.

Supply the tool number with your tool's offset data:

- Length X, length Z,
- Radius R and the
- Position of the tool edge



The "Alternat." soft key sets 9 tool point directions depending on the machining direction for the tool (also applies to mushroom):



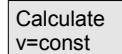
The specifications length X and length Z refer to point P with cutting edge positions 1-8; but with 9 to S (S = P).

The tool point direction 9 is not supported by ManualTurn cycles, but can be used in programs that were externally generated (G-code programs).

Constant cutting rate

If the machine manufacturer has included the feature, you can also assign a constant cutting rate v_{const} in m/min to each tool with the selection S(T) which you can then call in each machining step with selection S(T) as an alternative to the spindle speed.

Either enter the value for constant cutting rate in the table.



Or press the "Calculate $v=\text{const}$ " soft key while the spindle is rotating. ManualTurn then calculates the constant cutting rate for the selected tool from the current speed velocity and automatically transfers the value to the table.

You can enter offset data for other tools if necessary.

Offset data can be input provided that no start command has been given. Offset data can be overwritten.

3.8.2 Selecting/deselecting tool offset



Function

When the tool is selected, the control includes the programmed tool lengths in the X and Z directions in its calculations. The values for the tool tip position and tool nose radius are activated at the same time.



Sequence of operations

Tool offset

Select the soft key labeled "Tool offset".

Select an offset

Select

Select tool to be offset with cursor and press soft key "Select".

The activated tool is identified by the marker ">".

Deselect an offset

Deselect

Select the soft key labeled "Deselect".

This deselection causes tool number 0 to be selected. This has no programmed offset values.



Offsets can be selected/deselected provided that no Start command is active. The offset data are calculated in the absolute display.

This process does not involve any axis motions.

3.8.3 Measure tool



Function

The "Measure tool" function measures the length compensation values of the selected tool in the X and Z directions. The offset is calculated on the basis of

- the stored or current position and
- the workpiece dimensions that you have specified.

You can gauge tools as follows:

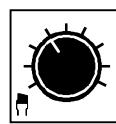
Example 1: Scratch the workpiece with the tool and measure the diameter of the workpiece.

Example 2: Approach a known reference point on the workpiece with the tools and make measurements.



Example 1

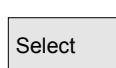
Sequence of operations



Select any operating mode.



Use the cursor to select the tool to be measured in the menu "Tool offset".



Select the soft key labeled "Select".



Select the soft key labeled "Measure".

Select setup mode or use handwheel to traverse tool up to workpiece, scratch workpiece.



Store the position in the X direction with soft key "Store Pos. X".

Retract tool from workpiece with Setup Measure the actual diameter of the workpiece and enter it under "Workpiece dimensions: X", e.g.: 60 mm



Confirm your settings with "Input".

The tool length X is automatically calculated and entered for the selected tool.

Follow the same procedure for the Z axis.





Example 2

Precondition

Sequence of operations

- Any mode is selected and you are in "Tool offset" mode.
- A workpiece with known dimensions is clamped in the chuck.



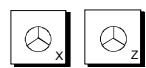
Use the cursor to select the tool to be measured in the menu "Tool offset".



Select the soft key labeled "Select".



Select the soft key labeled "Measure".



Approach known reference point on the workpiece with the handwheel (e.g. in X direction).

Enter the value of the known reference point in X direction under "Workpiece dimensions: X",
e.g. 50.239mm.



Confirm your settings with "Input".

Length X is automatically calculated and entered for the selected tool.
Select the next tool in the tool list and change to the "Measure" menu.
Approach the workpiece with the tool.



Press the "Last value" soft key to accept the known reference point in the input field.

Length X is automatically calculated and entered for the newly selected tool.

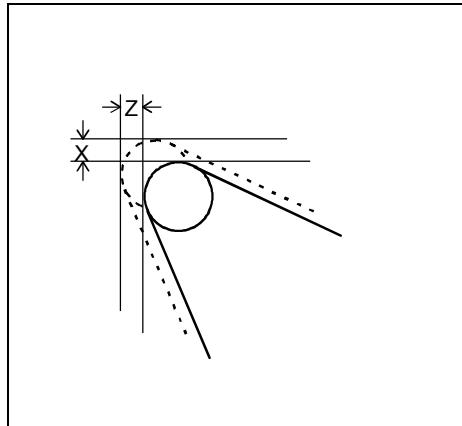
Use the same procedure to measure further tools.

3.8.4 Tool wear compensation



Function

Changes (tool wear) of the active tool form can be taken into account in length X and Z.



Sequence of operations

Tool

Select the soft key labeled "Tool" in the extended soft key menu in any mode.

Enter the current wear,
e.g. for Z: 0.1 (choice of mm or inches)



Activate the pocket calculator function by pressing the "Edit" key.
When you enter an offset value of + 0.1, 0.1 (either mm or inches) is added to the current wear value.

3.9 Measuring system changeover inch/metric



Function

This function allows you to change over between the two measuring systems (inch/metric) according to the drawing's measurement specifications.

You can change over the measuring system for the entire machine via the "Inch" soft key in the extended soft key menu. After each measuring system changeover your machine will show all dimensions in "inch" or "metric".

At changeover, all necessary specifications are converted into the new measuring system, e.g.

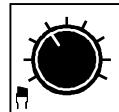
- positions
- tool offsets
- zero offsets.



Input values in the dialog boxes are not taken into account for the measuring system changeover.



Sequence of operations



Select any operating mode



Select the extended soft key menu by pressing the ">" key



Press the "Preset" and "Inch" soft keys

- Changeover metric to **inch**: Soft key is active
- Changeover inch to **metric**: Soft key is not active

After activating the "Inch" soft key, a dialog box queries whether you want to perform the changeover.



After you confirm by pressing the "OK" soft key, the measuring system is adapted accordingly.

Notes

Turning Simple Contours

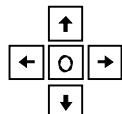
4.1	Turning in manual mode	4-64
4.2	Turning with path dimension input	4-66
4.2.1	Turning with STRAIGHT mode	4-67
4.2.2	Turning with CONICAL mode.....	4-69
4.2.3	Turning with CIRCLE mode	4-71
4.3	Turning with the contour handwheel and JOG keys +/–	4-75

4.1 Turning in manual mode



Function

Conventional feed traversal for facing, longitudinal and taper turning operations that have no specific end point is performed in MANUAL mode.

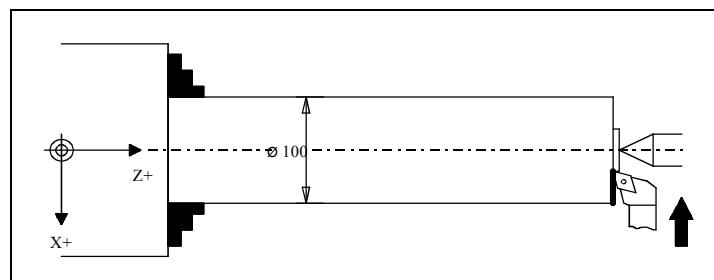


After a tool number, spindle speed and feedrate have been entered, the direction of motion is preset, for example, with a control stick.

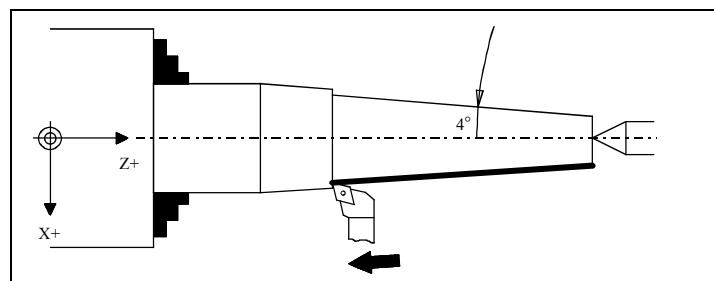
The direction indicator on the operator interface always indicates the direction of travel.

For taper machining operations, it is possible to preselect a 45° angle with the control stick and to define additional angles using the "Conical" function.

Example of facing



Example of taper turning



In addition, in MANUAL mode you can traverse the axes to a certain position and execute simple machining sequences or start the program from a specific position.



Sequence of operations

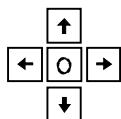


Select MANUAL mode.



a) Facing/longitudinal turning

Select the soft key labeled "Straight".

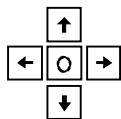


Select the desired axis direction.

X Facing

Z Longitudinal turning

b) Taper turning



Select reference direction Z.

Enter angle and confirm your selections by pressing the "Input" key.

e.g. α 4°

Enter tool number of tool currently in use.

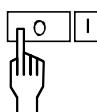
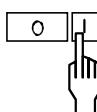
Enter feed mode and speed and confirm your settings by pressing the "Input" key.

e.g. F 0.3 mm/rev

S 320 rev/min

Feed movement with

- Press Start key

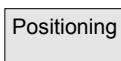


- Press Stop key

- You can interrupt the feed motion at any time by pressing the Stop key or turning the control stick back to the center position.
- If manual mode is active you can also traverse the axes using the handwheels. However, if you move the transverse axis using the handwheel, the travel path generated by the handwheel is not included in the constant cutting speed.



c) Positioning



Select the soft key labeled "Positioning".

Select the axis to be traversed and enter the target position.

Enter the desired value for feed F, for example, 1000 mm/min.

The selected feed is only effective for positioning and not for the elements straight or conical.

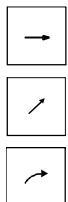


4.2 Turning with path dimension input



Function

The following geometry elements are provided as modes for turning simple contours:



- STRAIGHT for facing and longitudinal turning
 - CONICAL for taper turning
 - CIRCLE for radius machining
-
- You can specify absolute or incremental path dimensions in any of the available modes.
 - The direction pointer in the graphic display area indicates the current standard direction of travel for the selected geometry element.



4.2.1 Turning with STRAIGHT mode

You can perform the following steps in STRAIGHT mode:

- face and longitudinal turning in feed mode
- paraxial positioning in rapid traverse mode with specific target point
- positioning the rotary axis (e.g. C).



Soft key X, face turning over a certain diameter (abs), or face turning along a specific path (inc).



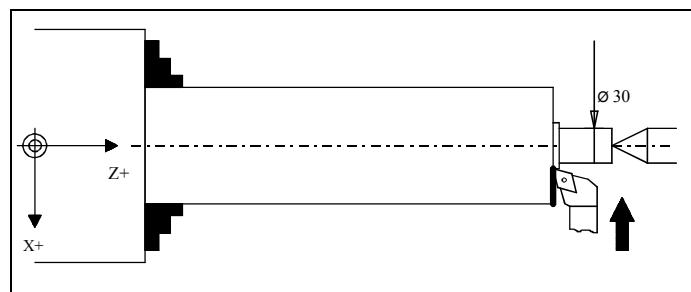
Soft key Z, longitudinal turning up to a certain position from zero (abs) or longitudinal turning along a specific path (inc).



Soft key C, rotary axis

The motion stops automatically when the target is reached.

Example of facing



Sequence of operations

a) Face turning

Select STRAIGHT mode



Select the soft key for the desired axis, here the X axis.



Before entering a value, enter "abs/incr." with the "Alternat." soft key.

Enter the traversing path (incr.) or end position (abs) and confirm by pressing the "Input" key

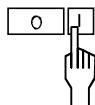
e.g. 30 abs.

Enter the feedrate or spindle speed in the appropriate input field

e.g. F 0.3 mm/rev

S 700 rev/min.

4.2 Turning with path dimension input



Initiate feed motion by pressing the Start key.



- The feed motion can be interrupted at any time by means of the Stop key.
- Longitudinal turning is performed similarly to face turning by means of soft key "Z".

b) Positioning the rotary axis (C)



Select the soft key for the desired axis, here the C axis.



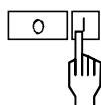
Before entering a value, enter "abs/incr." with the "Alternat." soft key.

Enter the traversing path (incr.) or end position (abs) and confirm by pressing the "Input" key

e.g. 30 abs.

Enter feed type (rapid traverse or mm/min) and confirm by pressing the "Input" key

e.g. F 2 mm/min



Initiate feed motion by pressing the Start key.



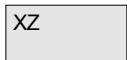
The input value mm/min is traveled as °/min.

4.2.2 Turning with CONICAL mode



Function

The following parameter setting options are available to you for taper turning operations:



Input of XZ coordinates

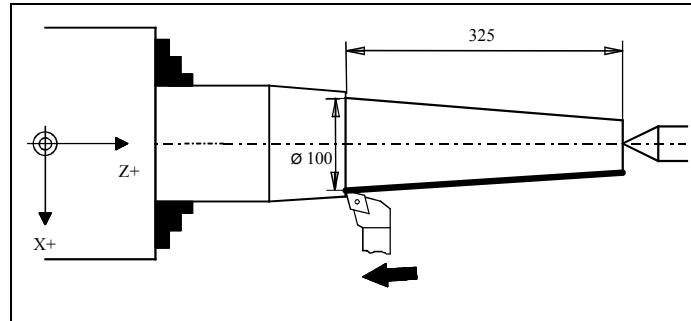


Input of an X or Z coordinate and an angle

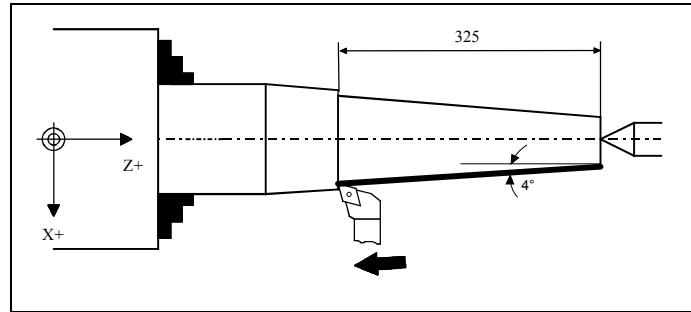


Input of an X or Z coordinate and a rotary axis (e.g. C)

Example of taper turning with input path dimension



Example of taper turning with input path dimension and angle



4.2 Turning with path dimension input



Sequence of operations



Select the CONICAL mode.

a) Taper turning with specified path dimension

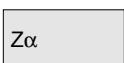


Select soft key labeled "XZ".

Enter traversing paths and confirm your selections by pressing the "Input" key

e.g. X 100 abs
 Z -325 inc.

b) Taper turning with specified path dimension and angle



Select desired soft key with path dimension and angle specifications

Enter traversing path and angle and confirm by pressing the "Input" key

e.g. Z -325 abs
 α 4°.

c) Positioning with linear axis (X/Z) and rotary axis (C)



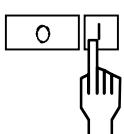
Select the soft key labeled "XZC".

Enter traversing paths and confirm your selections by pressing the "Input" key

e.g. Z 100 abs
 C 20 inc.

Enter feed type (rapid traverse or mm/min) and confirm by pressing the "Input" key

e.g. F 2 mm/min.



Initiate feed motion by pressing the Start key.



The feed motion can be interrupted at any time by means of the Stop key. When you switch over between XZ, Xα and Zα, a value conversion is performed if permitted by the existing values.

4.2.3 Turning with CIRCLE mode



Function

- Angle
- Radius
- IK

CIRCLE mode provides you with a selection of three alternative input options for machining radii:

Radius machining with angle specification

Radius machining with final dimension specification

Radius machining with IK and final dimension specification

Direction of rotation

The tool moves from the start to the end point along a circular path. Set the direction in parameter **Direction of rotation** using soft key "Alternat.".



Means counterclockwise circular direction



Means clockwise circular direction



Sequence of operations



Select CIRCLE mode.

a) Radius machining with angle specification

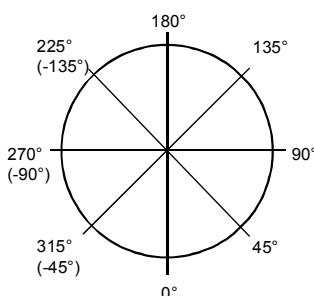
For radius machining with angle specification option, the **start position** can be defined either by the

- absolute angular position **D** or
 - the tangential run-in angle **α**
- and the **end position** either by means of the
- arc length in degree **E** or
 - tangential run-out angle **B**.

Mixed input is possible, e.g. tangential run-in angle **α** and length of arc **E**.

Both input types refer to the following angular system.

Angular system

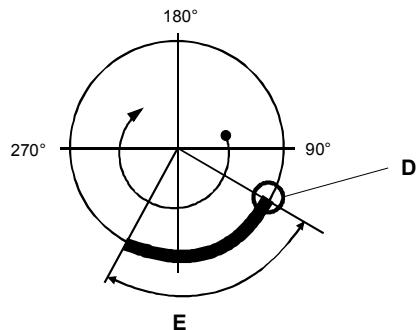


4.2 Turning with path dimension input

Example 1

$D = 60^\circ$

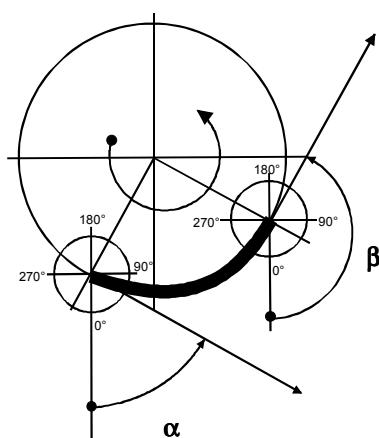
$E = 90^\circ$



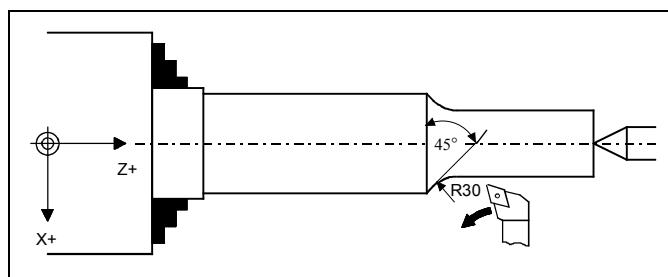
Example 2

$\alpha = 60^\circ$

$\beta = 150^\circ$



Example 3



Angle

Select the soft key labeled "Angle".

Enter circle data and confirm each entry with the "Input" key

e.g. D 180°

 E 45°

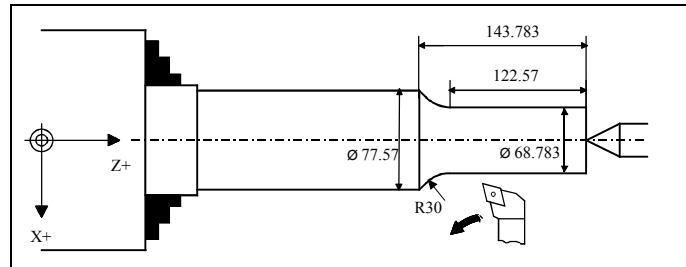
 R 30.

Alternat.

Select alternative circle data by means of the "Alternat." soft key.

b) Radius machining with final dimension specification

Simple radius definition by specifying the end point in X and Z and the radius and the direction of rotation.

Example

Radius

Alternat.

Select the soft key labeled "Radius".

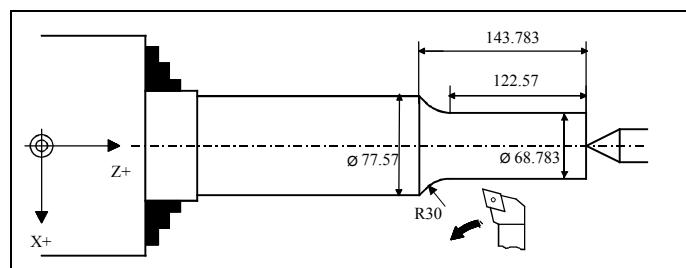
Before entering the value, set "abs/inc" and the direction of rotation "↺" with the soft key labeled "Alternat.".

Enter circle data and confirm each setting by pressing the "Input" key

E.g. X 8.787 inc
 Z -21.213 inc
 R 30.

c) Radius machining with IK and final dimension specification

Radius specification by entering the end point in X and Z and the position of the arc center with parameters I (in the X direction) and K (in the Z direction). The direction of rotation must also be specified.

Example

IK

Alternat.

Select the soft key labeled "IK".

Before entering the value, set "abs/inc" and the direction of rotation "↺" with the soft key labeled "Alternat.".

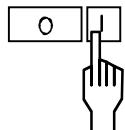
Enter circle data and confirm each setting by pressing the "Input" key

e.g. X 8.787 inc
 Z -21.213 inc
 I 30.0 inc
 K -0.000 inc.

4.2 Turning with path dimension input

If required, select and overwrite Feed and Speed using the cursor keys.

e.g. F 0.25 mm/rev
 S 530 rev/min.



Initiate feed motion by pressing the Start key.



The feed motion can be interrupted at any time by means of the Stop key.

When you switch over between the "Angle", "Radius" and "IK" soft keys, a value conversion is performed provided that a valid circle is described by the existing values.

4.3 Turning with the contour handwheel and JOG keys +/-



Function

The contour handwheel functions as a third handwheel and allows you to traverse programmed motions manually.



These movements can be executed with the programmed feedrate using the JOG keys +/- . All travel movements started with NC start can be traversed manually using the contour handwheel or the JOG keys +/- .

The programmed motion can consist of a single element, a contour sequence, a cycle or a complete EasyStep program.



Selection and deselection of contour handwheel

The contour handwheel is generally selected/deselected via an illuminated key, i.e.:

- Illuminated key OFF = contour handwheel and JOG keys +/- deselected
- Illuminated key ON = contour handwheel and JOG keys +/- selected

Selection

The handwheel can be **selected** both before an NC start command as well as during a motion.

Selection of the handwheel interrupts any motion that is already in progress. The motion can be continued either with the handwheel or with the JOG keys.

Deselection

If the contour handwheel function is **deselected** (illuminated key OFF) while a machining operation is in progress, the control system switches to "Stop" mode. The "NC stop" key lights up because there is still a distance to go.

The "NC start" key must be pressed to continue the machining operation with the programmed feed values.

Notes on the contour handwheel

The contour handwheel is active as soon as you select it and press the "NC Start" key.

When you select the direction of rotation of the handwheel, the axes travel at a velocity that depends on

1	10	100
---	----	-----

- the preselected increment and

- the rotational speed of the handwheel (as is the case for the handwheels of the X and Z axes)

Traversing direction

The traversing direction is determined by the rotational direction:

- Clockwise:** Forwards motion according to contour path. The next block is inserted as soon as the end of the block or element is reached.
- Counterclockwise:** Reverse motion according to contour path. In this case, the axis can only travel as far as the beginning of the relevant block or element.

Programmed feedrates are not active!

The contour handwheel has no effect in the "Incremental" and "Reference approach" modes.



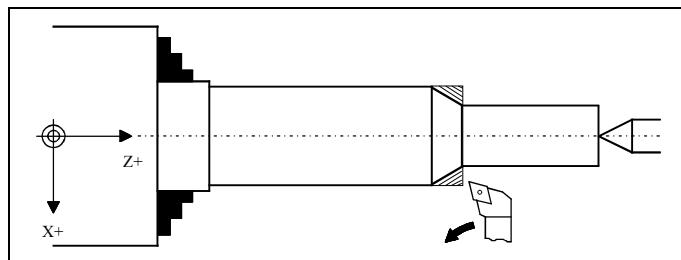
Notes on the JOG keys +/-

As soon as you have selected the contour handwheel and pressed the "NC start" key the JOG keys +/- are active.

- Press the JOG key + and keep it pressed to move the axis forwards at the programmed feedrate along the contour. The next block is inserted as soon as the end of the block or element is reached.
- Press the JOG key – and keep it pressed to move the axis backwards at the programmed feedrate to the beginning of the block or element.

Example

Place a chamfer on a contour with the contour handwheel.



Sequence of operations



Approach the Z dimension with the handwheel.



Select MANUAL mode.



Select direction required for the chamfer.



Activate contour handwheel.



Press the "NC start" key.



Activate handwheel Z.



Move tool in Z direction.



Machine the chamfer with the contour handwheel.



Retract the tool with the contour handwheel.



Use the handwheel to move X in the X direction again.

Repeat the steps described above until the chamfer is the correct size.

Notes

Turning with Cycles

5.1	General notes.....	5-80
5.2	Turning cycles in CYCLE mode	5-81
5.2.1	Thread cutting	5-81
5.2.2	Controlling thread cutting operations	5-87
5.2.3	Re-working a thread.....	5-88
5.2.4	Undercuts Form E and F.....	5-89
5.2.5	Thread undercuts	5-91
5.2.6	Drilling in longitudinal axis (center).....	5-93
5.2.7	Hole circle drilling	5-97
5.3	Stock removal/grooving cycles in STOCK REMOVAL mode.....	5-100
5.3.1	Stock removal cycles	5-100
5.3.2	Grooving cycles.....	5-105

5.1 General notes



The following section describes how you can program turning cycles using the ManualTurn system. Its purpose is to help you in selecting the turning cycles and assigning parameters to them. The menu tree for the turning cycles can be altered by the machine manufacturer. This section describes the menu tree in its original form.



All turning cycles can also be executed in single-block mode (see also section "Program generation with EasyStep and G code").

Calling turning cycles



You can select the following turning cycles using, for example, the operating mode switch in the CYCLE and STOCK REMOVAL modes:

Operating mode	Cycle	Function
	Thread	Transversal thread Longitudinal thread Taper thread
	Undercut	Form F Form E DIN thread Thread
	Drilling	Deep hole drilling Tapping Hole circle face/peripheral
	Stock removal	    
	Grooving	  

Programming and displaying turning cycles

You set the parameters of the turning cycles in a dialog in the parameter input field. You can either call the programming graphics of the parameterized cycle or the help display into the graphic display area by pressing the information key.

Cycle end point

At the end of the cycle, the tool returns to the position it was in before the cycle started.

5.2 Turning cycles in CYCLE mode

5.2.1 Thread cutting



Function

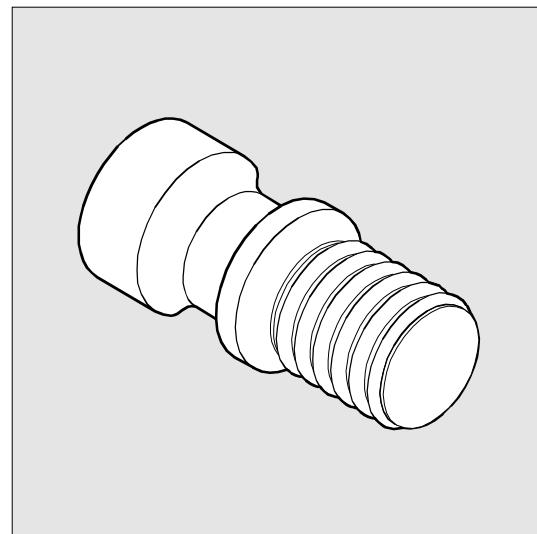
The "Thread" cycle allows you to produce cylindrical/tapered external and internal threads with a variable lead, as well as transversal threads. The threads may be single or multi-turn.

Infeed is automatic. You can select one of two infeed variants, i.e. constant infeed per cut and constant cutting cross-section.

Whether the thread is left-handed or right-handed is determined by the rotational direction of the spindle and the feed direction.



This cycle can only be applied on machine tools that have a speed-controlled spindle with position measuring system.



Parts program management When you select the cycle "Thread" from the extended soft key bar, the menu "Directory" offers you the option Parts program management. You can store parameterized dialog forms in this directory as a file and later call them up again (see Section "Parts Program management").



Sequence of operations

Precondition

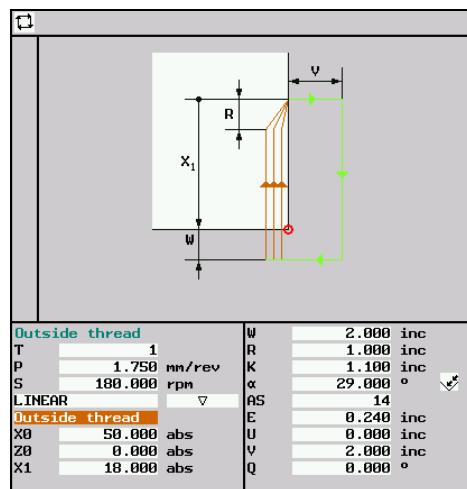
Thread

CYCLE mode is selected.

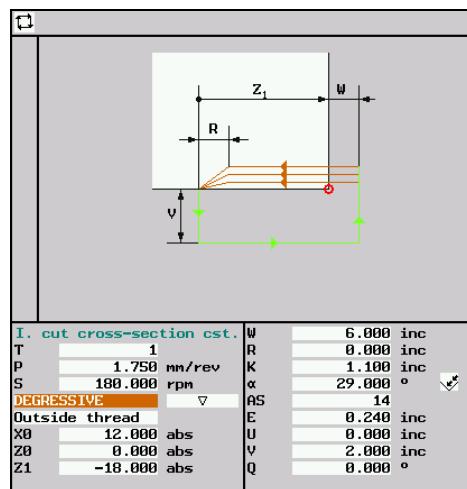
Select the soft key labeled "Thread".

Select the type of threading you require by pressing the associated soft key.

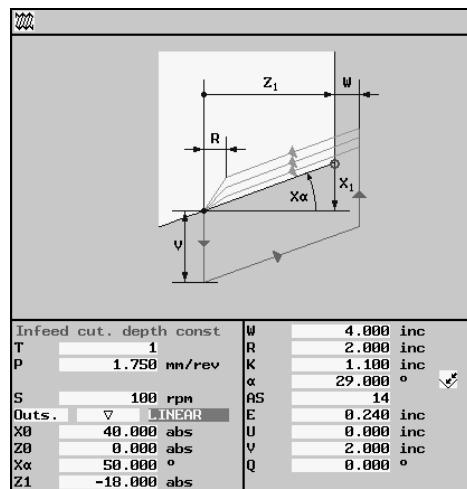
Face
thread



Longitud.
thread



Taper
thread





Explanation of parameters based on example of tapered thread

P

Thread pitch

G

Lead change

G = 0 The lead does not change.

G > 0 The lead is increased by the value G.

G < 0 The lead is reduced by the value G.

If the start and end lead of the thread are known, the lead change to be programmed can be calculated as follows:

$$G = \frac{|P_e^2 - P^2|}{2*Z_1} \text{ [mm/rev}^2]$$

Where:

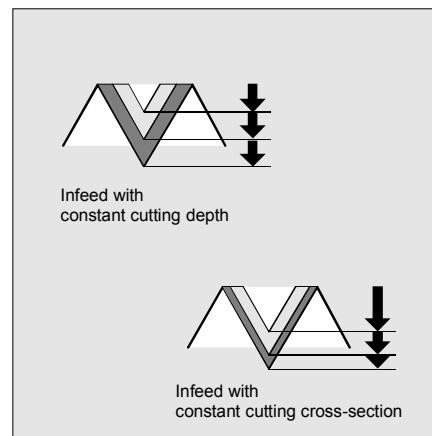
P_e is the end lead of the thread [mm/rev]

P is the start lead of the thread [mm/rev]

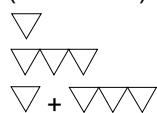
Z₁ is the thread length [mm]

A greater lead means a greater distance between thread turns on the workpiece.

Cut segmentation (alternative)

LINEAR
DEGRESSIVEInfeed with constant cutting depth
Infeed with constant cutting cross-section

Machining mode (alternative)



You can choose between the types of machining:

Roughing

Finishing

Complete machining (roughing and finishing)

Internal/external

Internal/external thread

(alternative)

X0, Z0

Reference points for dimensioning

X1, X α

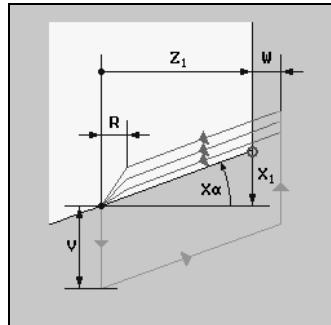
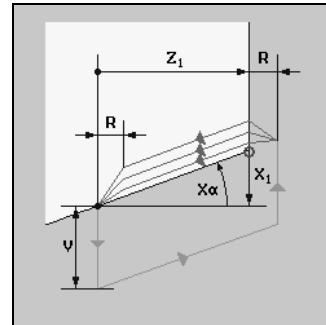
Length (inc) or thread end position in X direction or thread taper (degrees) (only with tapered thread)

Z1

Length (inc) or end point of thread in Z direction

W

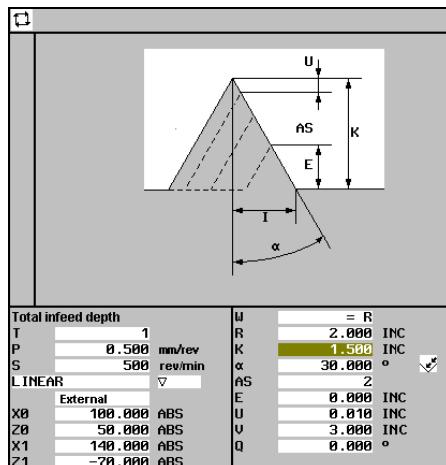
Thread run-in (enter without sign)

Thread run-in **W**Thread run-in **W = R**

The start point used in the cycle corresponds to the start point shifted forward by a distance corresponding to the thread run-in path **W**. You can also select a thread run-in with the value **W = R** via soft key "Alternat.".

R

Thread run-out (enter without sign)

**K**

Total infeed depth

α

Incline of infeed as angle

I (alternative)

Incline of infeed as angle

- Infeed along rear flank $\alpha > 0$
- Infeed along front flank $\alpha < 0$

Alternatively, you can select the incline of infeed as width **I**.

Parameter α is set to determine the angle of infeed in the thread. If you require an angle of infeed of 90° to the cutting direction, then this parameter must be set to zero. If infeed is to take place along the flanks, then the value of this parameter may not exceed half the flank angle of the tool.



Infeed along flank



(alternative)

Infeed along alternating flanks

You can infeed along alternate edges instead of just one edge to avoid loading the same tool cutting edge every time. That increases tool life.

$\alpha > 0$: Start at rear flank

$\alpha < 0$: Start at front flank

AS

Number of roughing cuts or first infeed (mm)

When changing between the number of roughing cuts and the first feed, the associated value is displayed respectively.

E

Insertion depth (enter without sign)

E is the depth already reached in the previous machining operation, i.e. the insertion depth only has to be entered if thread finishing is intended.

U

Finishing allowance (enter without sign)

The programmed finishing allowance **U** is subtracted from the specified thread depth **K**, the remainder being divided up into the number of roughing cuts. The cycle automatically calculates the individual, current infeed depths as a function of the programmed cut segmentation.

When the thread depth to be machined is divided into infeeds with constant cut cross-section, all roughing cuts are made with the same cutting pressure. The tool is then fed in at different infeed depths.

A second possible variant is to divide the total thread depth into constant infeed depths. In this case, the cut cross-section increases with every cut, but this technology results in improved cutting conditions for low thread depth values.

The finishing allowance **U** is removed in one cut after the roughing process.

V

Return distance (enter without sign)

Q

Angular offset for single-turn threads ($-360^\circ < Q < 360^\circ$)

With this parameter you can program the angular value that determines the point at which the thread starts on the circumference of the part.

e.g. $Q = 30.0$

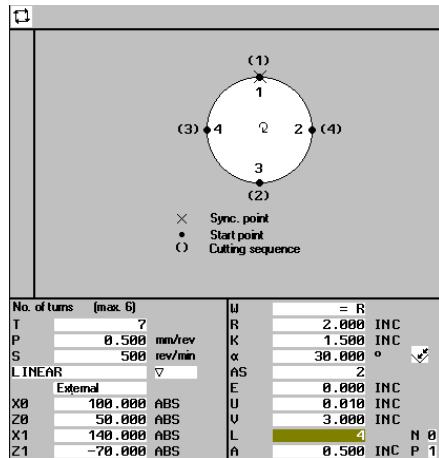
The thread start point is at 30° .

Parameter for multiple thread

Q (alternative)

Example showing how to parameterize a multi-turn thread

Selection of parameters via Q and soft key "Alternat."



L

Number of turns (max. 6)

The number of threads for a multiple-start thread is set in parameter **L**.

The threads are evenly distributed along the circumference of the turned part; the 1st thread start is always at 0°.

Should you require a multi-turn thread with a different start point on the circumference, then the cycle must be called individually for each thread when the appropriate starting point offset **Q** is programmed.

N

1 of L turns.

- Machine all thread starts: Value = 0
- Machine only this thread start: Value = 1 ... L

P

First turn P = 1 ... L

If P > 1, the threads that to be machined beforehand are omitted!

A

Turn-change depth (enter without sign)

In the case of deep threads, specify the infeed depth of the thread before each thread change with **A**. If the depth at which turn changes is to be ignored, simply enter 0.

5.2.2 Controlling thread cutting operations



Function

Stop
Turn CD

If the soft key "Stop Turn CD" is selected, then the cycle is stopped after the thread change depth is reached and the thread is machined (e.g. for measuring the thread depth).

To continue the cycle, press the NC Start key.

Noncut

If the soft key "Noncut" is selected, no further cutting depth infeed takes place, e.g. for smoothing the flanks.

Single
cycle

If the soft key "Single cycle" is selected, only one thread cut is executed per thread.

The next thread cut takes place with NC Start.

Single
block

If the "Single block" soft key (PROGRAM mode) is selected, the thread is fully machined.



The NC stop key can be pressed at any time to interrupt tool feed. An NC Stop during thread cutting triggers a retraction from the thread cutting. With NC Start you can continue machining from the start point where the thread cutting was interrupted.

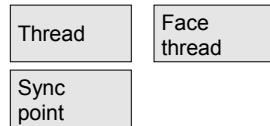
5.2.3 Re-working a thread**Function**

The angular offset of a thread caused by tool breakage or re-measurement is taken into account and compensated by the "synchronization point" function.

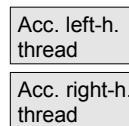
**Sequence of operations****Precondition**

The spindle is stationary.

Insert the thread cutting tool along the thread.



Press the "Sync Point" soft key in the appropriate thread screen form (thread face, longitudinal, or taper) when the threading tool is precisely in the thread turn.

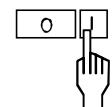


Select the appropriate soft key to choose when the thread is left-handed or right-handed.



By selecting the soft key labeled "Back", you can return to the higher-level soft key menu without activating the "Sync point" function.

Then retract the thread cutting tool such that the reference point can be approached without collision.



Press NC start to activate the thread cycle.

When the thread cycle is started, the angular offset is taken into account.



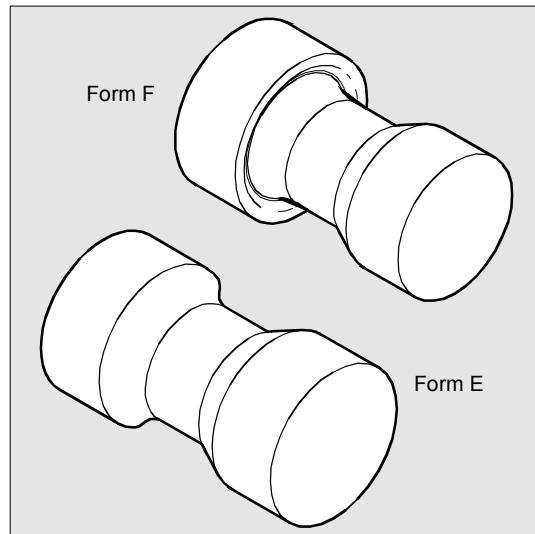
Entering an insertion depth **E** (depth already reached in the previous machining operation) avoids unnecessary noncuts during thread reworking.

5.2.4 Undercuts Form E and F



Function

These cycles allow you to machine form E and F undercuts to DIN 509.



Special cycles are available (see Section "Thread undercuts") for machining thread undercuts.



Sequence of operations

Precondition

CYCLE mode is selected.

Undercut

Select the soft key labeled "Undercut".

Form F
Undercut

or

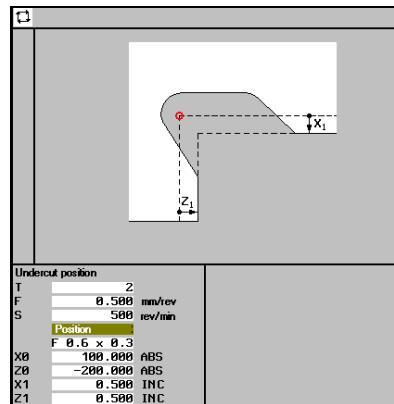
Form E
Undercut

Select the appropriate soft key for the undercut you require.

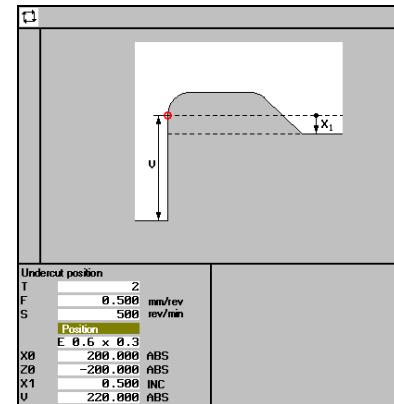


Explanation of parameters

Form F undercut

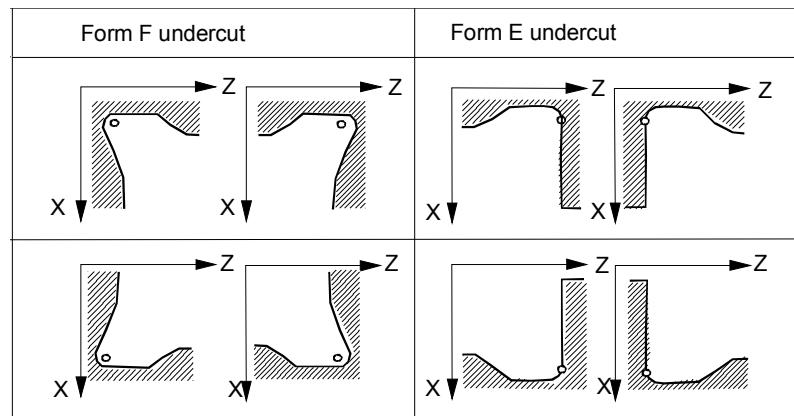


Form E undercut



Position (alternative)

This setting determines the position of the undercut in the coordinate system.



Undercut size

Selection according to DIN table: Radius/depth, e.g. F0.6 x 0.3 (Form F undercut). Can be set via the "Alternat." soft key.

X0, Z0

Reference points for dimensioning

X1

Allowance in X direction

Z1

Allowance in Z direction (for Form F undercut)

V

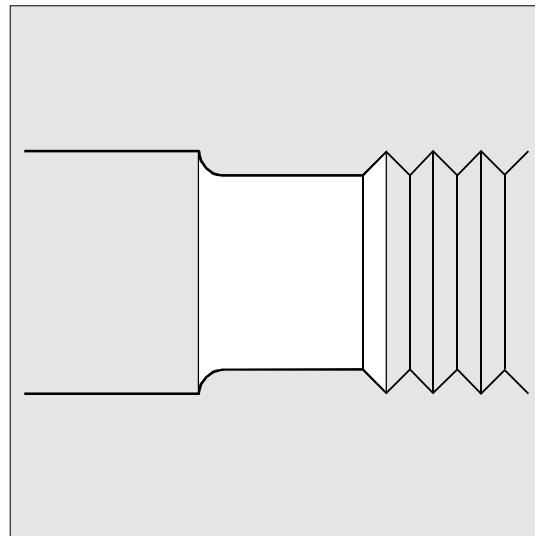
Cross-feed X

5.2.5 Thread undercuts



Function

These cycles are suitable for machining freely definable thread undercuts as well as thread undercuts to DIN 76 for parts with a metric ISO thread.



Parts program management

When you select the thread undercut cycle "Undercut Thread" on the extended soft key menu a Parts program management function is activated in the "Directory" menu. You can store parameterized dialog forms in this directory as a file and later call them up again (see Section "Parts program management").



Sequence of operations

Precondition

CYCLE mode is selected.

Undercut
DIN thread

You can either select a thread undercut according to DIN 76:

Select the soft key labeled "Undercut DIN thread".

Undercut
thread

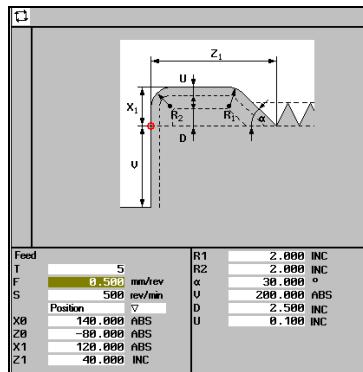
or select the freely definable thread undercut:

Select the soft key labeled "Undercut thread".

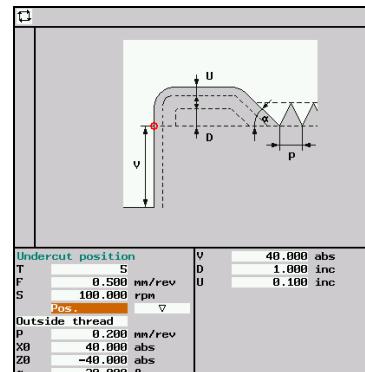


Explanation of parameters

Freely definable thread undercut:

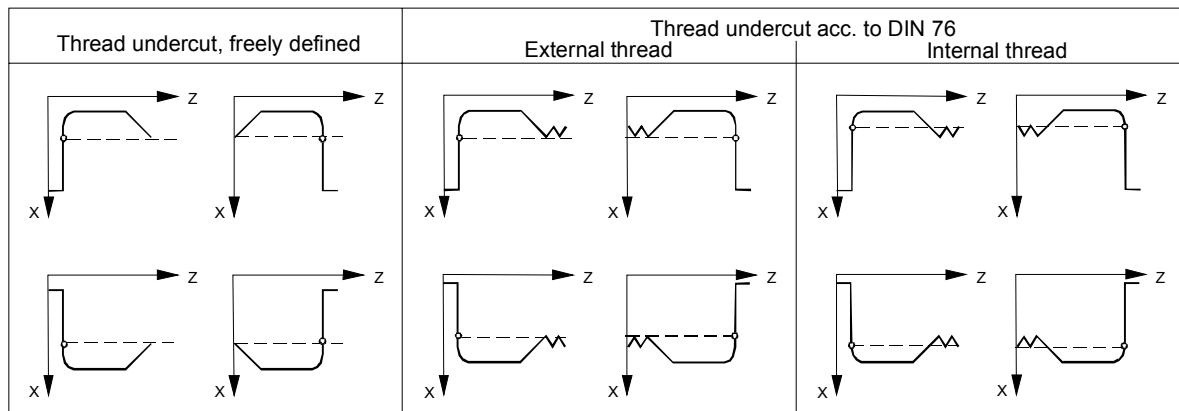


Thread undercut acc. to DIN:

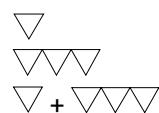


Position

This setting determines the position of the undercut in the coordinate system.



Machining mode



Select one of the following machining modes

Roughing

Finishing

Complete machining (roughing and finishing)

P

Lead in mm/rev: selection for DIN table (only for thread undercut DIN 76), e.g. 0.2

With other undercuts, enter the lead via parameter F.

X0, Z0

Reference points for dimensioning

X1

Undercut depth

Z1

Undercut width

R1, R2

Radius 1, radius 2 (only for freely definable thread undercut)

α

Approach angle

V

Cross-feed X

D

Infeed

U

Finishing allowance

5.2.6 Drilling in longitudinal axis (center)



You can perform deep hole drilling and thread drilling under the "Drilling center" function.

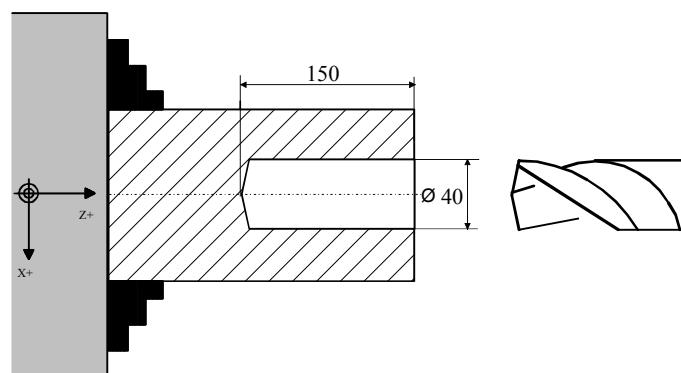


Deep hole drilling

The workpiece rotates at the programmed spindle speed. The tool drills in the longitudinal axis at the programmed feedrate until the specified final drilling depth is reached.

The deep hole is drilled by repeated, stepwise depth infeeds (depth value programmable) until the final drilling depth is reached.

As an option, the drill can be retracted after every infeed depth to the reference plane for swarf removal or by a distance which can be entered for chipbreaking.



Precondition

Sequence of operations

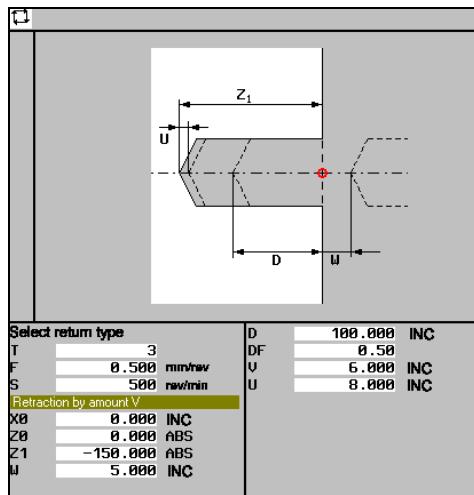
CYCLE mode is selected.

Drilling center Drilling

Select the soft keys "Drilling center" and "Drilling".



Explanation of parameters



Retraction planes
(alternative)

You can select one of two retraction types for deep hole drilling:

- Retraction by amount **V**
- Retraction to start position (reference point + safety clearance **W**)

X0, Z0

Reference point **X, Z**

Z1

Final drilling depth

W

Enter preprocessing (safety clearance), incremental and without leading sign.

D

1st infeed, enter without sign.

DF

Degression factor (0.001 ... 1.0)

This is the factor with which every further infeed is multiplied (DF=1.0: Infeed is always the same).

U

Minimum drilling depth, enter without sign.

The minimum drilling depth specifies the minimum amount of infeed, the depth must not be less than this amount.

V

Retraction amount, enter without sign.

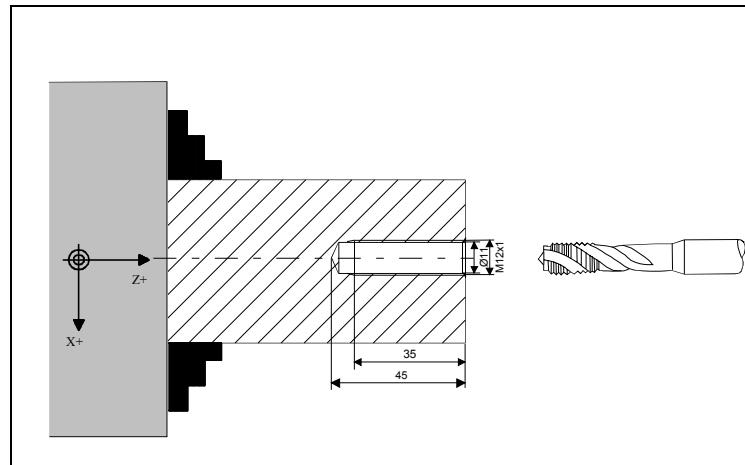
Amount by which the drill is retracted for chipbreaking.



Center tapping (rigid tapping)

This function is for drilling right and left-handed threads at turning centre. The tool drills in the longitudinal axis at the programmed spindle speed and feedrate (lead) until the specified final drilling depth is reached. The spindle rotation is reversed when the thread depth is reached, and the tool is retracted from the workpiece.

You can use this cycle for rigid tapping.



Rigid tapping is only possible with a second controlled spindle. You can specify in the machine data whether you want to use tapping with floating tapholder or rigid tapping.

Please follow the advice of the machine manufacturer.



Precondition

Drilling center

Thread tapping

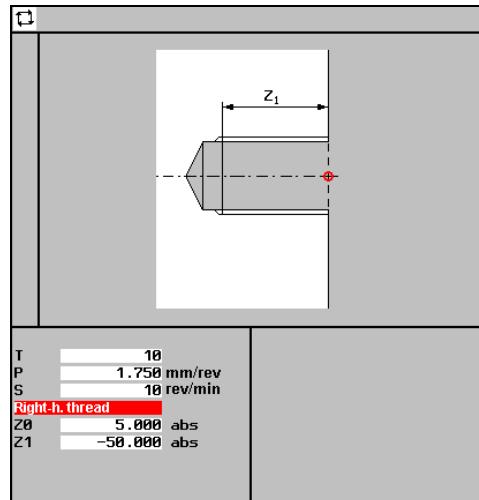
Sequence of operations

CYCLE mode is selected.

Select the soft keys "Drilling center" and "Thread tapping".



Explanation of parameters



P (alternative)

Thread lead, freely definable

mm/rev

inch/rev

Module

Turns/inch (thread turns per inch)

Thread direction

**Right-hand/left-hand
thread** (alternative)

Z0

Reference point Z

Z1

Thread depth

5.2.7 Hole circle drilling



Function

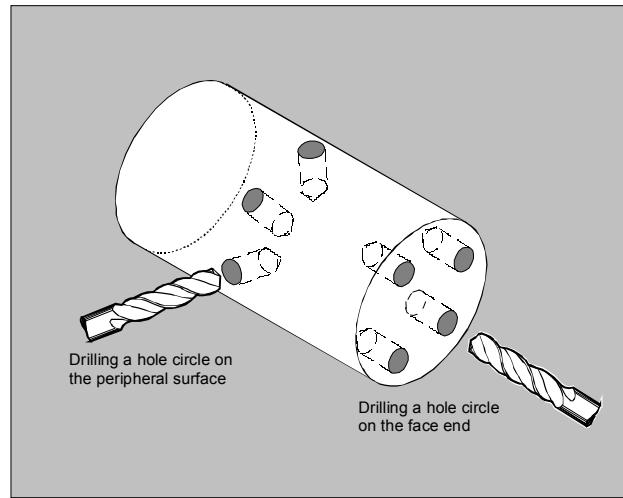
With this cycle you can create a hole circle on the face or peripheral end in front of or behind the turning center. Here you can select either deep hole drilling or tapping.

Sequence of operations

1. The workpiece rotates to the first position α_0 on the hole circle.
2. Initial machining is performed with the drill (2nd spindle) rotating and the workpiece immobile.
3. The workpiece advances through α_1 to the next position.
4. The second drilling operation is performed.
5. Steps 3 and 4 are repeated until all positions on the hole circle have been machined.



Please follow the advice of the machine manufacturer.



Sequence of operations

Precondition

Drilling hole circle

CYCLE mode is selected.

Drilling face

Drilling peripheral

Select the soft key "Drilling hole circle" and define the machining cycle with:

Deep hole drilling on hole circle

The hole circle can be machined at the face or peripheral end.

Tapping face

Tapping peripheral

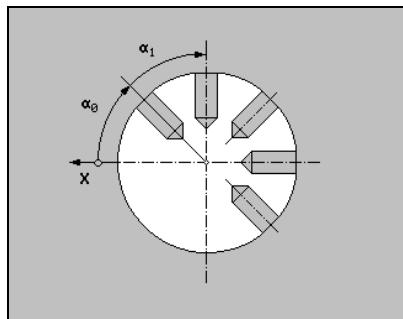
Tapping on hole circle

The hole circle can be machined at the face or peripheral end.

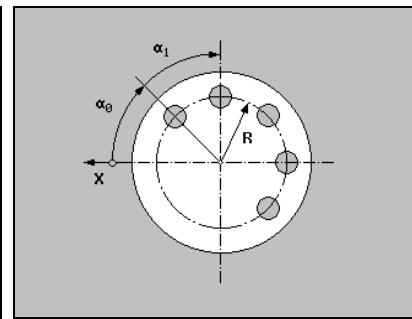


Parameters for hole circle drilling

Peripheral end



Face plane

**F**

Feedrate (only with drilling peripheral/face)

If you are drilling with a second spindle that is not supported by ManualTurn, the feedrate can only be entered in mm/min.

Please follow the advice of the machine manufacturer.

S2 ↗ (alternative)

Spindle speed of the second spindle with specification of the rotation direction.

P

Tool lead (only for peripheral/face thread).

Full/circle segment (altern.) Selection circle patterns

Peripheral end

Z0

Absolute coordinate Z

N

Number of holes

α0

Base rotation angle

This parameter defines the location of holes in the hole circle.

Parameter **α0** defines the base rotation angle of the first hole in reference to the X axis.

α0 >0: full circle/circle segment is rotated counterclockwise

α0 <0: full circle/circle segment is rotated clockwise

α1

Advance angle

Parameter **α1** contains the angle of rotation from one hole to the next and is only needed for setting parameters for a circle segment. If the value of parameter **α1** is zero, the advance angle is calculated in the cycle from the number of holes, such that they are evenly distributed along the circle.

α1 >0: further positions are rotated counterclockwise.

α1 <0: further positions are rotated clockwise.

X0

Reference point; corresponds to the diameter of the peripheral which is to be drilled into.

For drilling behind the turning center, you must enter a negative parameter value.

X1 Drilling depth (inc) or end diameter (abs) where the final drilling depth is.

Face end

R	Radius of hole circle For drilling behind the turning center, you must enter a negative parameter value.
N	Number of holes
α0	Base rotation angle (see peripheral end)
α1	Advance angle (see peripheral end)
Z0	Reference point
Z1	Final drilling depth.



For setting the parameters for the deep hole drilling cycle/thread drilling cycle contained in the hole circle cycle, please proceed as described in section "Drilling in the longitudinal axis (center)".

5.3 Stock removal/grooving cycles in STOCK REMOVAL mode



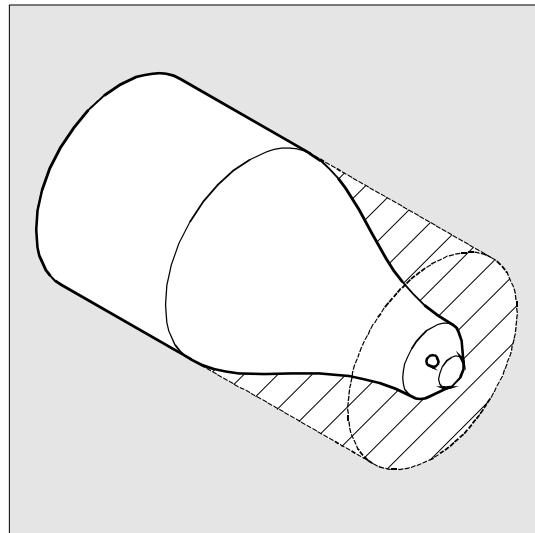
The menu tree for STOCK REMOVAL mode can be modified by the machine manufacturer.

This section describes the menu tree in its original form.

5.3.1 Stock removal cycles



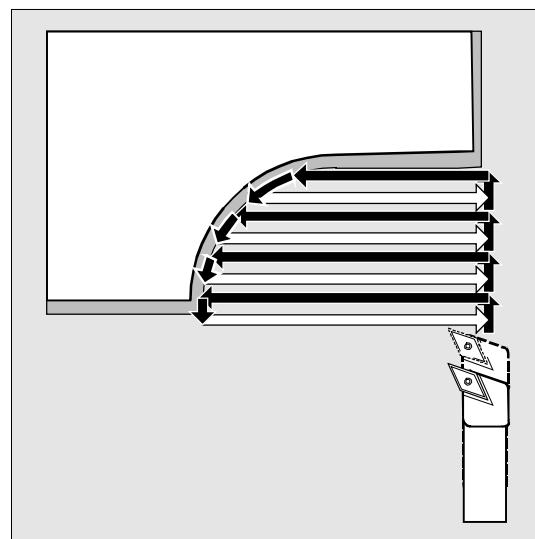
Function



You can select one of five stock removal cycles which are then selected via the soft keys displayed in STOCK REMOVAL mode.



The stock removal cycles allow internal and external contours to be faced and machined longitudinally. You can select one of three technologies, i.e. roughing, finishing and complete machining. For roughing of the contour, paraxial cuts are generated for the programmed infeed depth. Once a contour intersection point has been reached, the resulting residual corners are removed immediately parallel to the contour. The workpiece is rough-cut down to the programmed finishing allowance.

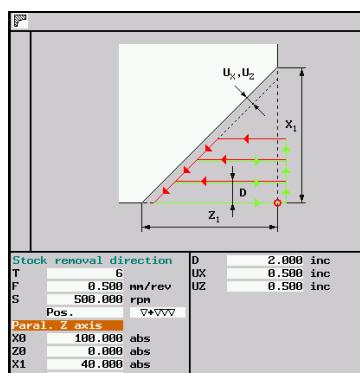
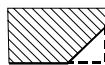


The part is finished in the same direction as it is rough-cut. The cycle automatically selects and deselects tool radius compensation as required during finishing.

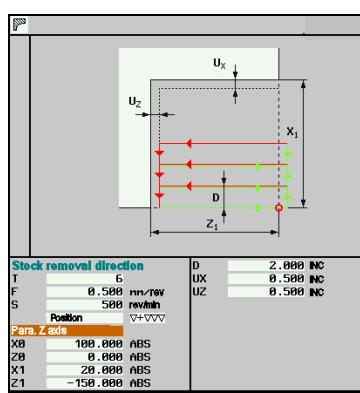
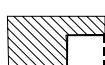


Precondition

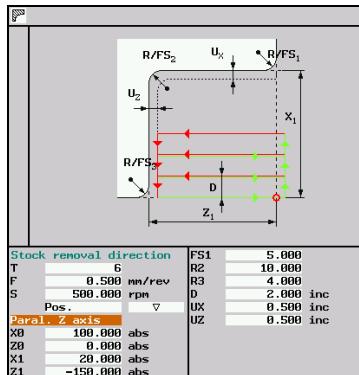
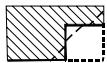
Select STOCK REMOVAL mode and call the required stock removal cycle by pressing the soft key.



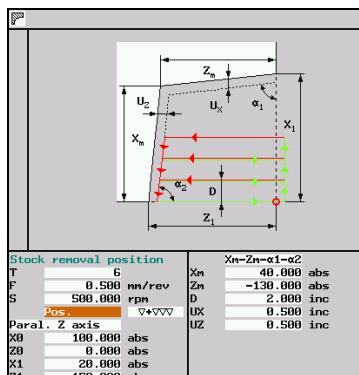
Simple stock removal cycle conical



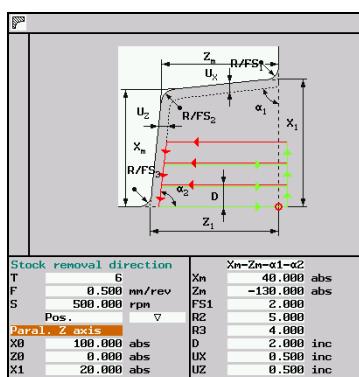
Simple stock removal cycle straight



Stock removal cycle straight with radii and chamfers



Stock removal cycle with conicals



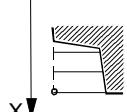
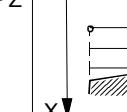
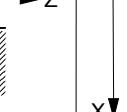
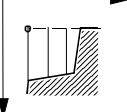
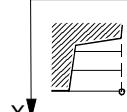
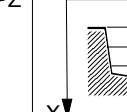
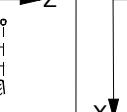
Stock removal cycle with conicals, radii and chamfers



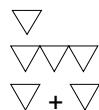
Explanation of parameters

Position parallel Z axis/ X axis

You can define the stock removal position and the stock removal direction in the coordinate system by means of these settings.

Parallel to X axis (longitudinal)		Parallel to Z axis (plane)	
External	Internal	Face end	Back plane
			
			

Machining mode



(alternative)

Select one of the following machining modes:

Roughing

Finishing

Complete machining (roughing and finishing)

X0, Z0

Reference points for dimensioning

X1, Z1

Final dimension (only the amount for "inc")

Xm - Zm - α1 - α2

You can set the specified parameters to other combinations in this parameter field.

Xm, Zm

Intermediate point (only the amount for "inc")

α1

Angle of 1st path

α2

Angle of 2nd path

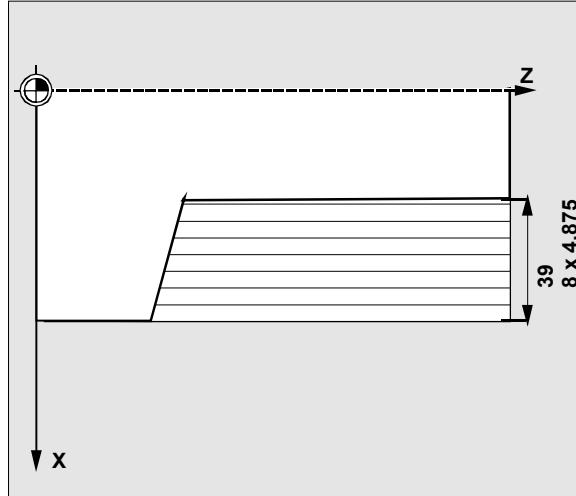
Rn, FSn

Input of roundings and chamfers ($n = 1 \dots 3$)

D

Infeed depth (enter without sign)

The maximum possible infeed depth for the roughing process is defined in parameter D. The cycle automatically calculates the infeed depth to be applied for roughing.

Example

The entire machining depth amounts to 39mm. When the infeed depth is a maximum of 5mm, 8 roughing cuts must be made. These are carried out with a constant infeed of 4.875mm.

 U_x , U_z **Finishing allowance**

A finishing allowance for the roughing operation is specified by means of parameters U_x and U_z . These parameters are not necessary in finishing machining mode, therefore they are not displayed in the input screen.

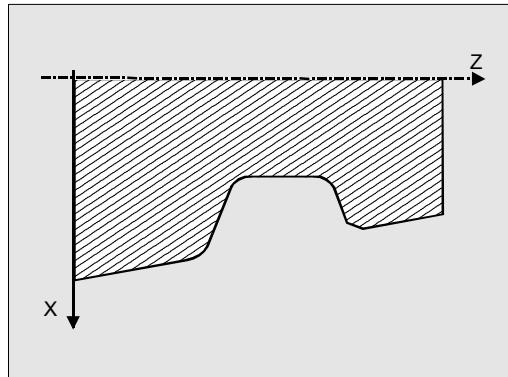
The roughing cuts are always made down as far as these finishing allowances. After every paraxial roughing operation, the residual corner produced is immediately removed to obviate the need for a special residual corner cutting operation on completion of roughing. If no finishing allowances are programmed, then the workpiece is rough cut down to the final contour.

5.3.2 Grooving cycles

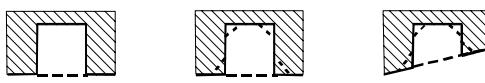


Function

With grooving cycles you can produce symmetrical and asymmetrical grooves for longitudinal and face machining on any straight contour elements. You can machine internal or external grooves.



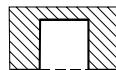
You can select one of three groove cycles with the soft keys in operating mode STOCK REMOVAL.



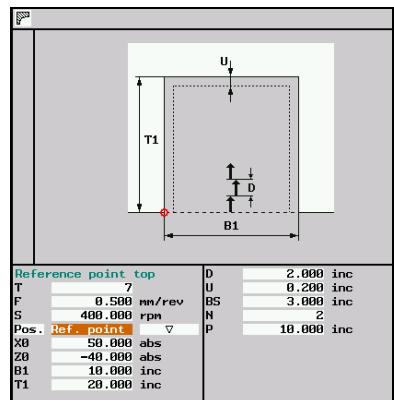


Sequence of operations

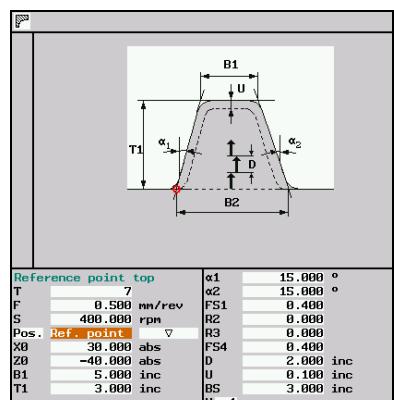
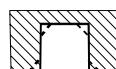
Precondition



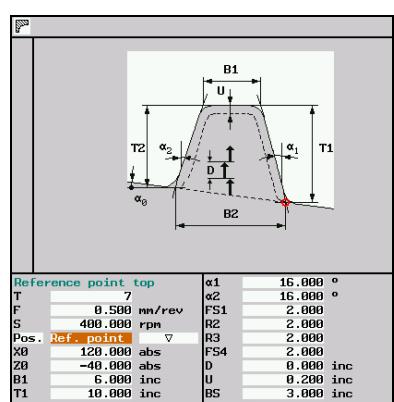
Select STOCK REMOVAL mode and call the required grooving cycle by pressing the soft key.



Simple grooving cycle



Grooving cycle with conicals, radii and chamfers



Grooving cycle in a conical with conicals, radii and chamfers



Explanation of parameters

Position

As an example for all grooving cycles, the parameters for the cycle stored under soft key 8 are explained below. The parameter description applies equally to the remaining two grooving cycles.

The position of the groove in the coordinate system is defined by means of this setting.

Position	Longitudinal groove	Face groove
External		
Internal		

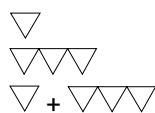
Reference point

You define the reference point on the groove base (reference point bottom) or the groove edge (reference point top) depending on the **position** in the coordinate system.

e.g. for an external longitudinal groove, the following options can be selected:

Position	Longitudinal groove	
	Reference point top	Reference point bottom
External		

Machining mode



Select one of the following machining modes:

Roughing

Finishing

Complete machining (roughing and finishing)

X0, Z0

Reference points for dimensioning

These coordinates define the point of a groove from which the cycle calculates the groove form.

B1, B2

B₁ = Groove width, bottom

B₂ = Groove width, top

T1, T2

T₁ = Settable depth at reference point

T₂ = Depth opposite reference point

Set the groove width **B1, B2** and groove depth **T1, T2** parameters to define the groove form. The cycle always bases its calculations on the point programmed under **X0** and **Z0**.

If the groove is wider than the active tool, then the width is cut in several steps.

 α_0

Angle of slope

The oblique angle in relation to the reference axis on which the groove is to be machined is programmed with parameter α_0 . The angle can be set to values of between -180° and $+180^\circ$.

- Longitudinal groove: $\alpha_0 = 0^\circ \Rightarrow$ parallel to Z axis
- Face groove: $\alpha_0 = 0^\circ \Rightarrow$ parallel to X axis
- α_0 is positive when rotational direction of X axis \Rightarrow Z axis

 α_1, α_2

Edge angle

Flank angles are separately entered so that asymmetrical grooves can be described. The angles can be set to values of between 0 and $< 90^\circ$.

 R_n, FS_n

The groove form is modified through the input of radii/chamfers on the edge or base ($n = 1 \dots 4$).

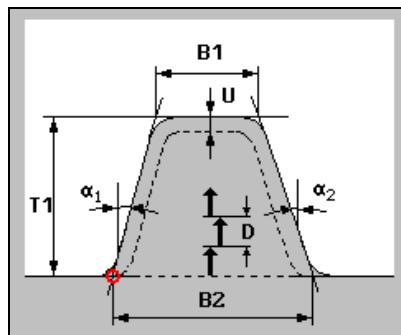
D

Infeed depth 1st cut

D=0 : 1st cut is executed straight to the final depth **T1**.

D>0 : The 1st and 2nd cut are executed at the infeed depth **D** on alternate sides, to ensure a better chip clearance and prevent tool breakage.

All other cuts are executed straight to the final depth **T1**.



The lateral infeed for the alternate-sided cutting is automatically defined in the cycle.

Alternate-sided cutting is not possible when the tool can only reach the groove base at one position.

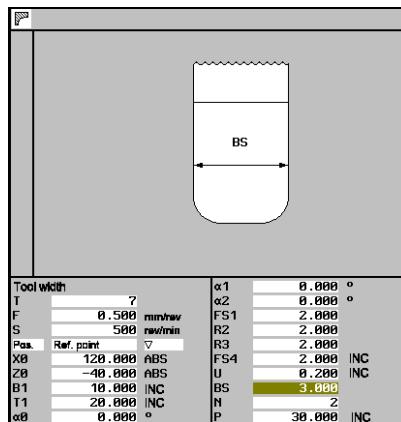
U

Finishing allowance

You can specify a finishing dimension for the groove base and flanks. The workpiece material is rough-cut down to this finishing allowance during roughing. Finally, a cut is made along the final contour with the same tool.

BS

Tool width



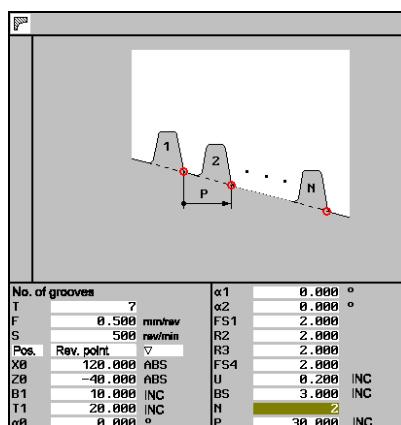
N

Number of grooves; with N = 1, P is not displayed.

P

Distance of grooves

- Longitudinal groove: parallel to Z axis
- Face groove: parallel to X axis



Notes

Turning any Contours (Free Contour Input)

6.1	General notes.....	6-112
6.2	Create new contour.....	6-113
6.3	Symbolic representation of contour.....	6-114
6.4	Graphic representation of contour.....	6-115
6.5	Create contour elements.....	6-116
6.6	Editing contour elements.....	6-119
6.7	Stock removal against contour.....	6-122
6.8	Machining residual material	6-126
6.9	Single-cycle machining.....	6-127

6.1 General notes

6.1 General notes



Function

You can create and machine complex contours in CONTOUR mode. The integrated contour calculator calculates the points of intersection of the individual contour elements taking account of geometric relationships. You can link up to 100 contour elements with one another. The contour transition elements radius and chamfer are available in addition.

Contour elements are:

- Start point
- Straight line (face, longitudinal, conical)
- Circular arc

Machining the contour can take account of the blank contour to be defined before the finished part contour.

With stock removal, residual material is automatically detected and an updated blank contour generated. The excess material can be removed using a suitable tool with the "Residual material" function.

Parts program management



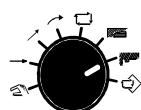
Operating mode CONTOUR offers you a parts program management function under soft key "Directory" (see Section "Parts program Management"). Existing contours can be called up again and edited in operating mode CONTOUR.

Operating mode CONTOUR is a software option.

6.2 Create new contour



Sequence of operations



New contour

Activate the CONTOUR machining mode.

Activate the "New contour" soft key to create a new contour.

Enter a name for the new contour and confirm.

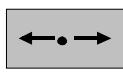
Start point

The input screenform for the **starting point** of the contour is overlaid.

Enter the start coordinates of the contour under X and Z.

You can enter the feed for all transitions (chamfers/roundings) under FRCM. If no feed is specified, the feed entered under stock removal is used instead. The feedrate units (mm/rev or mm/min) set under stock removal can also be used.

Contour elements



The following contour elements are available for defining a contour:

Straight line in Z direction



Straight line in X direction



Conical in X/Z direction



Arc in any direction of rotation

Contour transition elements

You can select either radius **R** or chamfer **FS** as the transition element between any two contour elements. The transition element is always placed at the end of the contour element. You always select a contour transition element in the parameter input form of the contour element.

6.3 Symbolic representation of contour**Function**

The contour is symbolically described in the order in which it was programmed in a continuous contour sequence next to the graphic window.

Contour element	Symbol	Definition
Start point		Start point of contour
Straight line to left/right		Straight lines in 90° grid
Up/Down		
Any straight line		Straight line with optional gradient
Circular arc		Circle
Contour termination	END	End of contour definition

Display elements

The symbols are displayed in different colors to indicate their status:

Foreground	Background	Definition
-	red	Cursor on new element
black	red	Cursor on current element
black	white	Normal element
red	white	Element currently separated (model of remaining elements)

6.4 Graphic representation of contour



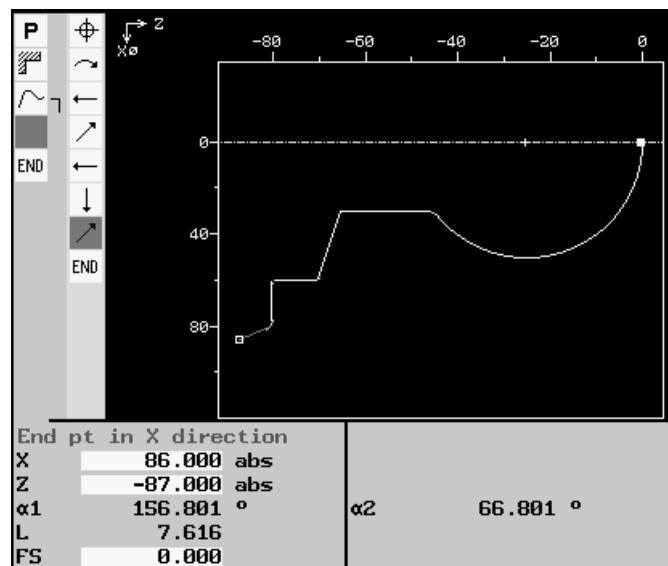
Function

In parallel to the sequential parameterization of the contour elements, each step in the creation of the contour is displayed graphically in the graphic window.

A selected element is displayed in the graphic window highlighted in red.

The coordinate system scale is adapted to any in the overall contour.

The symmetrizing axis of the contour is represented by a dot-and-dash line.



The position of the coordinate system is shown in the graphic window.

6.5 Create contour elements**Function**

Make your entries in the screen forms in the way described for operating modes STRAIGHT, CONICAL and CIRCLE.

The following soft keys are available to you for programming a contour:

All para-meters

This soft key switches the display over to the extended input form which contains the full range of input parameters for a contour element.

Tangent prev. elem.

Pressing the "Tangent prev. elem." soft key presets angle $\alpha 2$ to 0, i.e. transition to the preceding element is tangential.

Delete value

When you select the soft key labeled "Delete value", the value in the selected parameter input field is deleted.

✓ Accept

When you select the soft key labeled "Accept", the current settings in the parameter input form are transferred to the system and you return to the main soft key menu of CONTOUR mode.

○ Alternat.

This soft key only appears if the cursor is positioned on an input field with several parameter setting options.

+/-

You can change the sign of the value on which the cursor is positioned with this soft key.

Λ

Press soft key "Recall" to return to the basic soft key menu of CONTOUR mode **without** accepting the values you have just edited.

ⓘ HELP**Help displays**

Press the "Information" key to call a help display for every parameter input form showing the geometric meaning of the input parameters.



Explanation of contour elements

Straight



Direction/ Parameter			
X	—	End point in X -direction (+,-)	End point in X and Z directions
Z	End point in Z -direction (+,-)	—	
L	Length of straight line		
α_1	Angle of lead		
α_2	Transition angle: Preceding/current contour element $\alpha_2 = 0$: Tangential transition to preceding element		
Transition to following element	"Radius" or "Chamfer" as transition element can be selected with the "Alternat." soft key.		
FB	Feed of the straight line		

Circular arc



Parameters	Definition
Rotational direction	In clockwise/counterclockwise direction
R	Radius
X	End point in X direction (+,-)
Z	End point in Z direction (+,-)
I	X coordinate of circle center
K	Z coordinate of circle center
α_1	Start angle
β_1	End angle
α_2	Transition angle: Preceding element/current contour element
β_2	Arc angle
Transition to following element	"Radius" or "Chamfer" as transition element can be selected with the "Alternat." soft key.
FB	Feed of the arc

6.5 Create contour elements



A separate feed can be entered for each element. If no value is specified, the feed is taken over from the cutting cycle.

Dialog selection

Dialog selection

You are given a dialog selection if the parameters can belong to two different contours.

By toggling the soft key, you can display the two possible selections. The selection you have made is displayed by an unbroken line (white) and the alternative by a dotted line (green) in the graphic display area.

Accept dialog

By selecting the soft key labeled "Accept dialog", you can transfer your dialog selection to the system.

Change selection

You can cancel your dialog selection with this soft key. Both options are displayed again.

If previous input values make further selection superfluous, the dialog is stopped!

Notes on entering contours

- **Straight line/Circle or Chamfer/Radius?**

You can use a transition element whenever the two adjoining elements form an intersection point which can be calculated from the input values. Otherwise, you must use the contour elements Straight line/Circle.

- **Input value has already been calculated!**

If a contour is overdefined, a value to be entered may already have been calculated from other values.

Problems can result if the dimensions entered are inexact and an input value which should be precise does not correspond to the calculated value. In this case, values from which the value to be entered has been calculated must be deleted again. You can then enter the precise value.

6.6 Editing contour elements



Sequence of operations

Selecting a contour element



Select the contour element you require with the appropriate soft key.



The parameters for the selected element will then appear on the screen. The element name will be output on the top left of the menu.



As soon as the contour element can be displayed geometrically, it is highlighted in the graphic display area, i.e. the color of the contour element changes from white to yellow.



Adding a contour element



Select the element before the end marking using the cursor keys.



Select the contour element you want with the soft keys and enter the values known to you in the element input form.

Confirm your entries by selecting the soft key labeled "Accept".

Modifying a contour element



Use the cursor keys to select the contour element you want to modify.



When you press the "Input" key, the associated input form is activated with the existing settings and parameters.

The selected element is displayed zoomed in the programming graphics.

You can now make the changes.

The effects of the changes you have made are displayed immediately in the graphic display area, new calculated values are displayed immediately.



Transfer the modifications you have made to the system by selecting the "Accept" soft key.

6.6 Editing contour elements

Rejecting the modifications made to a contour element



Select the key "Recall" to exit the input form without accepting the changes.

Inserting a contour element



Use the to select the contour element **after** which you wish to insert another element.

Then select the contour element you wish to insert from the soft key menu.



After you have entered the parameters for the new contour element confirm the insertion by pressing the soft key labeled "Accept".

The contour elements downstream of the insertion are automatically updated in keeping with the new contour status.

Model of remaining elements

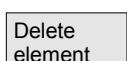
When an element is inserted in a contour, the remaining contour elements are ignored until the model produced in this manner is selected with the cursor and accepted!

If the model of remaining elements does not fit into the contour it is kept as a remaining model. However, you can append it **again and again** by inserting a non-parameterized conical!

Deleting a contour element



Select the contour element you wish to delete with the cursor keys



followed by the soft key labeled "Delete element"

Before you can delete the contour element you are asked for confirmation:



The contour element is deleted when you select the soft key "OK".



You can cancel the delete action with the "Back" soft key.

Creating a new contour

Contour
new

Press soft key "Contour new".

Once you have confirmed the prompt again (soft key "OK" or "Return") by pressing the "OK" soft key, you can start a new contour. The contour you have already defined is stored under the name you have given it.

6.7 Stock removal against contour**Function**

With the function "Stock removal", contours can be machined in longitudinal/facing direction. Stock removal can be performed for any blank. You can select one of three technologies, i.e. roughing, finishing and complete machining.



This function is a software option.

Remove stock

In the basic soft key menu of CONTOUR mode, confirm the soft key labeled "Remove Stock".

**Explanation****Longitudinal direction**

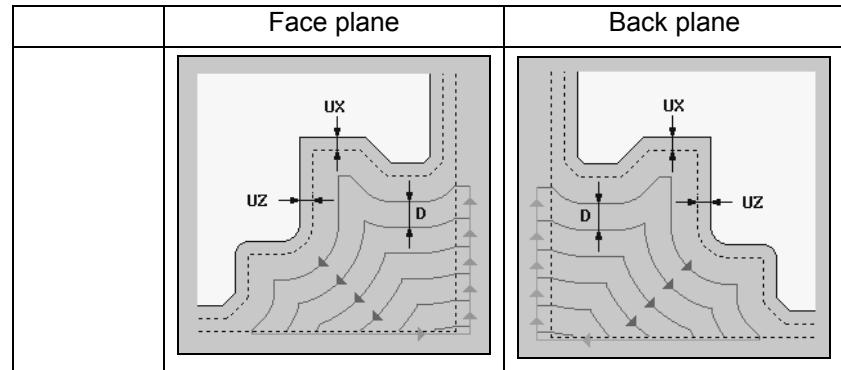
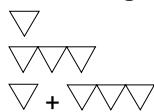
	External	Internal
parallel to Z axis		

Facing direction

	Face end	Back plane
parallel to X axis		

Parallel to contour

	External	Internal
parallel to contour		

**Machining mode**

Roughing
Finishing
Complete machining (roughing and finishing)

Direction of machining

Machining can be performed from the face end to the rear end, or from inside out towards the exterior, as well as vice versa.
The direction of machining depends on the tool selected.

D

Infeed depth

With contour roughing, parallel cuts are made using the maximum programmed infeed depth **D**.

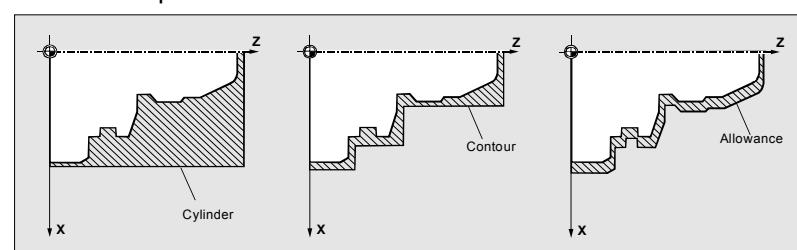
 U_x, U_z

Finishing allowance

The contour is roughed to the programmed finishing allowance U_x/U_z .
The part is finished in the same direction as it is rough-cut.

Blank

Blank description

 **X_D, Z_D**

Blank definition

Cylinder: The cylinder's dimensions can be entered in absolute values or incrementally as allowance in relation to the smallest possible cylinder that can be positioned around the contour.

For inside contours, the blank cylinder is typically equivalent to the drill-hole diameter.

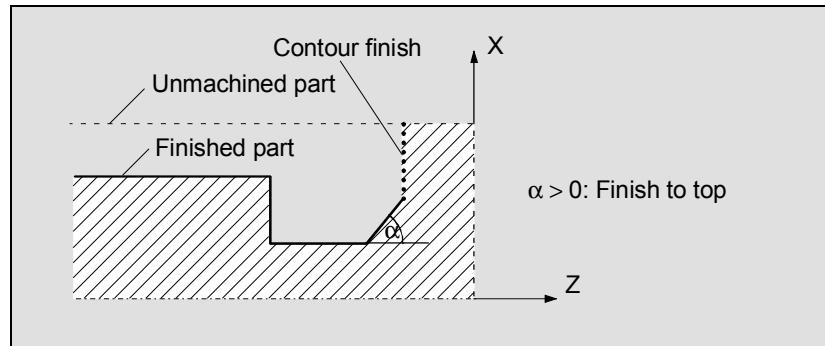
Allowance: The allowance for the finished-part contour is specified incrementally.

Contour: The blank contour must be defined as a separate closed contour definition before the finished-part contour.

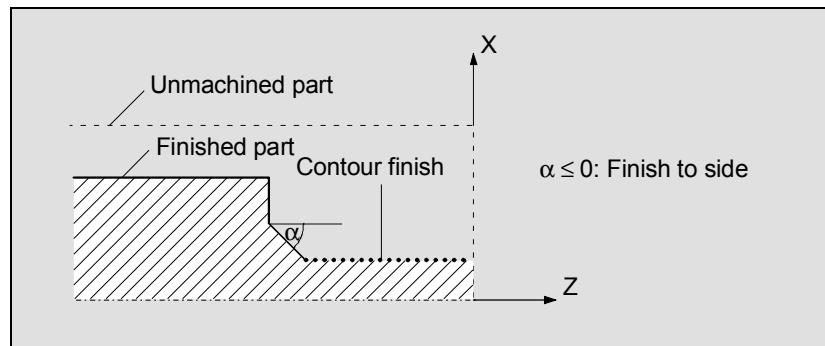
6.7 Stock removal against contour

If the blank and finished-part contours do not coincide, ManualTurn will define the connection between the blank and the finished part.

If the angle between the straight lines and the Z axis is greater than zero, the connection is upward, if the angle is less than or equal to zero, the connection is to the side.



Connection between the blank and finished part upward



Connection between the blank and finished part to the side

Relief cuts

Process machine relief cuts (yes/no)

A free contour can contain relief cut elements which is not the case for stock removal cycles in STOCK REMOVAL mode. You can specify whether the relief cut elements are to be machined or not.

Tool angle

Machine tool angle (yes/no)

The tool angle can be included in machining, for example, to avoid contour violations on relief cut elements.

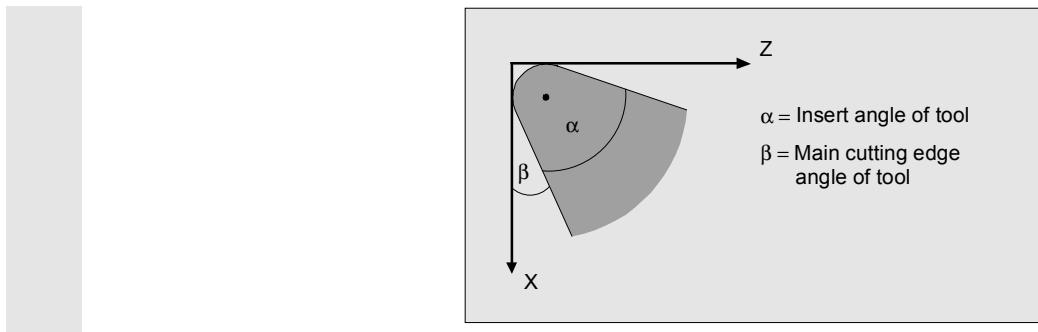
If the tool angle is not taken into account, ManualTurn will assume that a mushroom is used for stock removal.

α

Tool insert angle

β

Tool main cutting edge angle



6.8 Machining residual material**Function**

With stock removal against the contour, residual material is automatically detected and an updated blank contour generated. The excess material can be removed using a suitable tool with the "Residual material" function.



This function is a software option.



Stock removal of free contours is usually programmed as described below:

1. Stock removal (roughing)
2. Residual material (roughing)
3. Stock removal (finishing)



Residual
material

In the basic soft key menu of CONTOUR mode, confirm the soft key labeled "Residual material".

**Explanation**

A more detailed description of the parameters can be found in Section "Stock removal against the contour".

Machining

Parallel to contour or axis (facing/longitudinal)

External/internal/face end/rear side

Machining mode ▽

Roughing

Direction of machining

Machining can be performed from the face end to the rear end, or from inside out towards the exterior, as well as vice versa.

D

Infeed depth

U_x, U_z

Finishing allowance

Relief cuts

Machine relief cuts

Tool angle

Machine tool angle

α

Tool insert angle

β

Tool main cutting edge angle

6.9 Single-cycle machining



Function

A contour can also be machined as a single cycle. Prerequisite is that no program is selected (see Section "Switching off program").



Sequence of operations

Finish

Create a contour via soft key "New contour".

Define the individual contour elements (see Section "Create contour elements").

Complete the contour with "Finish". You return to the basic soft key menu of CONTOUR mode.

Select "Stock removal" and enter the parameters.

Start machining by pressing the "NC Start" key.

Activate soft key "Residual material" if necessary, enter the values in the parameterization screenform and start machining by pressing "NC Start".

Modify
contour

In order to modify the created contour, press soft key "Modify contour" in the basic soft key menu. The R parameter input screenform is overlaid.

Notes

Program Creation with EasyStep and G Code

7.1	General notes.....	7-130
7.2	Creating a machining sequence.....	7-131
7.3	Display machining sequence	7-132
7.4	Program steps.....	7-133
7.4.1	Inserting new program step.....	7-133
7.4.2	Special functions	7-134
7.4.3	Insert G code step.....	7-135
7.4.4	Changing program steps.....	7-135
7.4.5	Program editor	7-136
7.5	Switching off the program	7-137
7.6	Start machining sequence.....	7-137
7.7	Single-step mode (single block).....	7-138
7.8	Block search.....	7-138
7.9	Tool nose radius compensation	7-139
7.10	G code programming	7-141
7.10.1	Select program view.....	7-141
7.10.2	G code editor.....	7-143
7.10.3	Create new parts program	7-145
7.10.4	Inserting program blocks.....	7-145
7.10.5	Editing program blocks.....	7-146

7.1 General notes**Function****EasyStep
programming**

You can create parts programs in two different ways with ManualTurn:

The EasyStep function allows you to create a complex, executable program by chaining elements such as straight lines, tapers, circles and cycles together. You do not require G code programming knowledge to use the function. This type of program is called machining sequence in the following description. This sequence consists of different program steps.

G code programming

The G code function (in the extended soft key menu) allows you to write parts programs in G code with your programming knowledge. The ManualTurn cycles are available in the G code editor to assist you in a way you are familiar with.

Main soft key bar

The basic soft key bar offers the soft key functions described below.

Program on	New program	Program editor	Special function	Block search	Single block	Simulation	Directory
------------	-------------	----------------	------------------	--------------	--------------	------------	-----------

Parts program management Operating mode PROGRAM offers you a parts program management function under soft key "Directory" (see Section "Parts program management"). The programs you create are stored there. Existing programs can be transferred back to the editor for editing.

The parts programs can be imported and exported via the RS-232 interface.

7.2 Creating a machining sequence**Sequence of operations**

New
program

Select PROGRAM mode.



Press the soft key labeled "New program".

Enter a name for the machining sequence and confirm. The name must not contain more than 24 characters. All letters (excluding accented characters, such as German umlauts), figures, and the underscore (_) are allowed.

The machining sequence is stored in the Program directory.



END

Switch to the operating mode in which you want to start your new machining operation. Insert the required program step into the machining sequence. Repeat this procedure for further program steps. The individual program steps are inserted into the program via "Accept".

When a new EasyStep program is created, the machining symbol "END" is automatically inserted in the end of program step in the machining sequence.

7.3 Display machining sequence

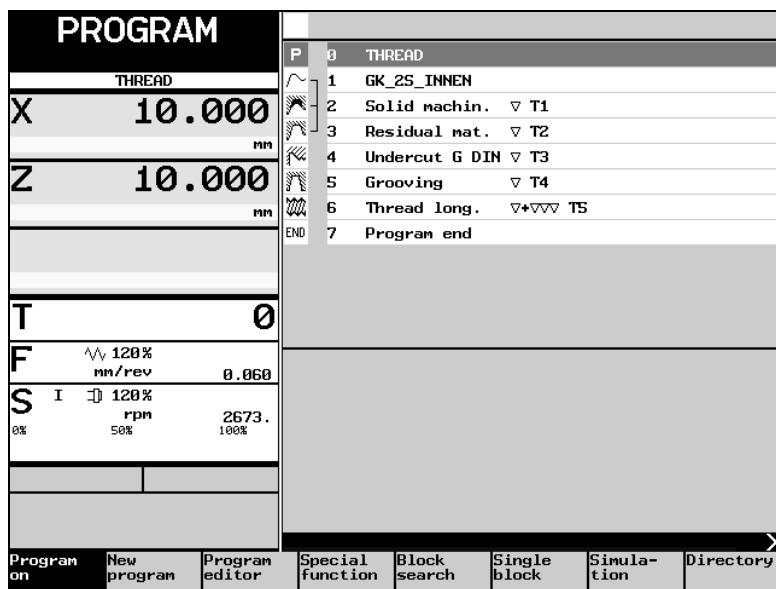


Sequence of operations



You can call the EasyStep flow chart from every mode (except for MANUAL) via the information key.

The machining sequence is displayed with the individual program steps.



The contour steps contour, stock removal and residual material are linked in the flow chart via a square bracket next to the machining symbol.



You can switch between the contour display and the EasyStep flow chart by activating the "Information" key.

You can navigate in the machining sequence using the following keys:

Move one line up or down.



Page one screen forward or back.



Jump to the beginning or end of the machining sequence.



7.4 Program steps

7.4.1 Inserting new program step



Function

An EasyStep program step consists of program instructions such as

- Geometry and cutting data of an element (e.g. straight, conical, circle)
- Special functions
- Program end instructions

You can test the execution of the individual program steps on the machine tool before accepting them into the machining sequence (tool approach).



Sequence of operations



Call the EasyStep flow chart via the information key.



Position the cursor at the desired location in the machining sequence.

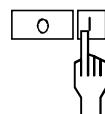


Select the required operating mode and enter the parameters for the program step.

Select the soft key "Accept". The input screenform is closed and the new program step is inserted in the machining sequence behind the current cursor position.

A maximum of 150 program steps can be chained.

Tool approach



You can test the execution of the program step on the machine tool before accepting it into the machining sequence by pressing "NC Start". The "Dry run" function is also available for testing (see Section "Dry run"). The program step parameters can be adapted if necessary and tested again on the machine.



Warning

With tool approach the tool traverses in rapid traverse directly to the setpoint position of the program step.

Please ensure that there is no obstacle in the traversing path.

7.4.2 Special functions

Special
functions

Function

If you select the soft key labeled "Special functions" in the main soft key menu of PROGRAM mode, you can insert the following functions into your machining sequence:

Soft key	Symbol	Function
G code block	G	Insert G code blocks and comments (max. 241 characters)
Auxiliary commands	M	Spindle: Spindle right Spindle left Oriented spindle Stop 0 Spindle off - Coolant: On, Off, - Gear step: 0, I, II, III, - Other M commands: See data supplied by machine manufacturer 2nd spindle: Direction of rotation Speed rev/min
Dwell		Input in revolutions
Stop		Programmed stop (NC stop)
		Tool nose radius compensation on, in machining direction right of the contour
		Tool nose radius compensation on, in machining direction left of the contour
		Tool nose radius compensation off

Please read the instructions on tool nose radius compensation in Section "Tool nose radius compensation".



You can also use the special functions outside programs, e.g. to start your 2nd spindle (tool spindle).

7.4.3 Insert G code step



Function

G code steps and comments can be inserted into a machining sequence.

Special functions

G code block

In PROGRAM mode, select soft keys "Special functions" and "G code block".

Programming G code step

- A G code block cannot be inserted after the program end.
- It is not possible to display the G code block in the contour view.



For a detailed description of G code programming, please refer to: /PG/, Programming Guide, Fundamentals

Comments

Comments can be inserted in the program to provide information about the individual program steps. Comments start with a semicolon ";".



You insert the step into the machining sequence by pressing the "Input" key and the "Accept" soft key.

7.4.4 Changing program steps



Sequence of operations



You can call the EasyStep flow chart from every mode (except for MANUAL) via the information key.



Select the desired program step in the machining sequence with the cursor.



Press the "Input" key.

The dialog screen form for the selected program step now appears on the screen. The selected cycle is displayed zoomed in the programming graphics.

You can now alter the parameters.



Select the soft key labeled "Accept" to update all the data in the selected program step of the machining sequence.



Press the soft key "Recall" to return to the machining sequence without saving your changes.

7.4.5 Program editor



Function

Program steps can be deleted, copied and inserted again with the "Program editor" function.

Program editor

Press the soft key "Program editor" in PROGRAM mode.

Changing a program step

Use the cursor keys to select the step you want to change. Then press the appropriate soft key:

Cut

Deleting a program step

Copy

Insert

Copying and inserting a program step

Cut

Insert

Moving a program step

A program step is always inserted after the current program step.

Changing multiple program steps

Use the cursor keys to select the first or last program step you want to change and then press the "Select" soft key. Then, using the cursor keys, select the rest of the program steps you want to edit.

Cut

Deleting program steps

Copy

Insert

Copying and inserting program steps

Cut

Insert

Moving program steps

Copying program steps into another program

First, copy the steps you want to insert in another program (please refer to the instructions for selecting and copying).

Directory

INPUT

Then open the program into which the program steps are to be inserted

Program editor

and select the program editor.

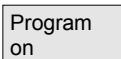
Insert

Using the arrow keys, select the block after which the copied blocks are to be inserted and then press the "Insert" soft key.

7.5 Switching off the program



Function



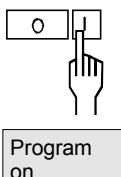
You can use the "Program on" function to interrupt the generation of an EasyStep machining sequence, for example, if you wish to start the parameter assignments for an operating mode as a single cycle first.

Select the soft key labeled "Program on".

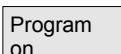


The soft key is no longer highlighted. The machining sequence disappears from the screen. The icon in the status line displays the symbol for the current operating mode.

The soft key "Accept" is no longer available in any of the operating modes.



Press NC Start to start the parameterized cycle.



You can return to the machining sequence by selecting the soft key "Program on" again.

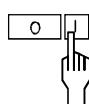
7.6 Start machining sequence



Sequence of operations



Call the current machining sequence via the information key in any mode (except for MANUAL).



Or select the required machining sequence via the parts program management (see Section "Parts program management") in PROGRAM mode.

Start the machining sequence with the "NC Start" key. The program is executed in the specified sequence.

The program step currently being executed is displayed in the status line.

7.7 Single-step mode (single block)



Sequence of operations

In single-step mode (single-block mode) you can execute a machining cycle step by step.

Precondition

The required machining sequence is selected in PROGRAM mode.

Single step



Select the soft key labeled "Single step".

Each time you press the "NC start" key one step of the machining cycle is executed.

7.8 Block search



Function

The block search function allows the program to advance to the desired position in the machining sequence. The search target can be directly determined by positioning the cursor keys.



Sequence of operations

Precondition

The required machining sequence is selected in PROGRAM mode.



Select the step from which you wish to continue execution with the cursor keys.

Block search

Select the soft key labeled "Block search".

The same calculations are performed during block search as in normal program operation. However, the axes do not move!



Press soft key "NC Start" twice.

- The "Block search" function is started when you press it the first time.
- Processing starts at the selected step when you press it again.



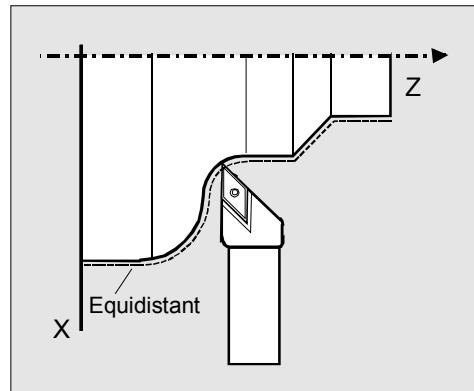
You can cancel the search by activating the Reset key.

7.9 Tool nose radius compensation



Function

When tool nose radius compensation is active, the control automatically calculates the required equidistant tool paths to the programmed contour for the current cutting radius R.



The control requires the following information to calculate the tool paths:

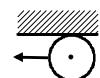
- The tool compensation data (length X, length Z, tool nose radius R, tool point direction). The distance between the tool path and the workpiece contour is calculated from the tool nose radius and the tool point direction.
- The control recognizes the direction in which the tool path is to be moved from the machining direction.



Special functions

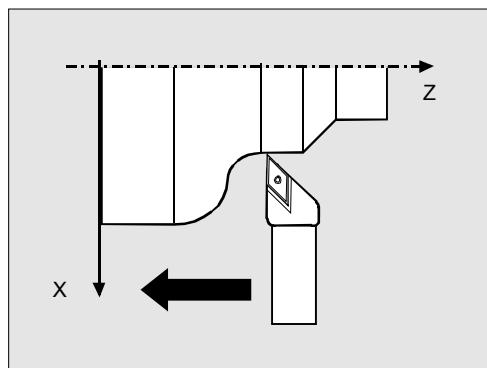
Sequence of operations

The tool nose radius compensation function can be found in the special functions menu in the basic soft key menu in PROGRAM mode.



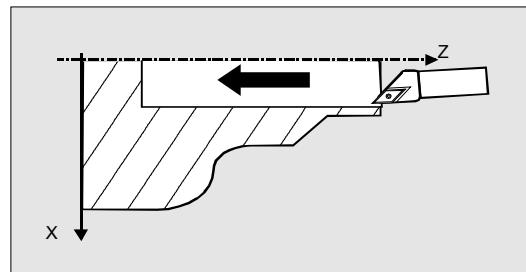
Activate tool nose radius compensation, tool operates in the direction of machining (arrow) **left** of the contour

Example: External machining



Activate tool nose radius compensation, tool operates in the direction of machining (arrow) **right** of the contour

Example: Internal machining



Deactivate tool nose radius compensation

You can deselect tool nose radius compensation both left and right of the contour with "Deactivate tool nose radius compensation". The tool nose radius is then ignored. To ensure collision-free travel deactivate the tool nose radius compensation for travel outside the workpiece contour (e.g. when retracting the tool).

Notes

- The tool nose radius compensation is taken into account in CYCLE (with exception of "Drilling" and "Thread" cycles), STOCK REMOVAL and CONTOUR modes.
- A linear block must always follow a tool nose radius compensation selection or deselection command.

7.10 G code programming

7.10.1 Select program view



Sequence of operations

Precondition

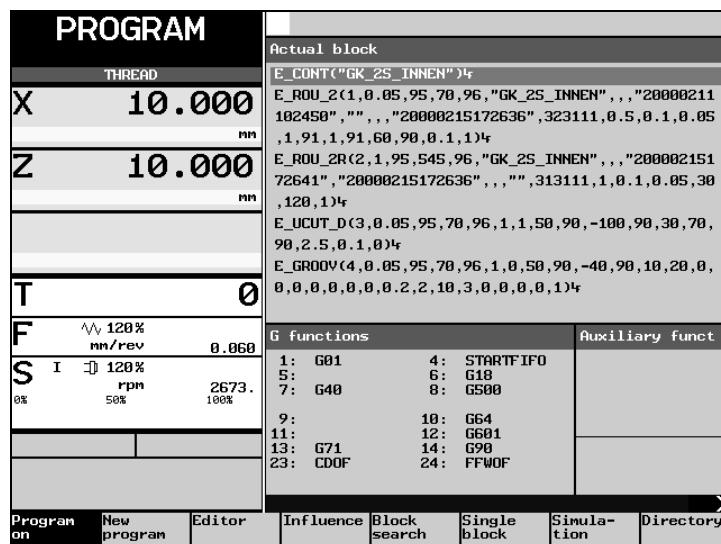


G code

PROGRAM mode is selected.

Activate the soft key "G code" in the extended soft key menu.

The program is displayed in G code view.



Explanation of program view

Current block

The current block is highlighted when the program is being executed.

G functions

The "G functions" window displays a list of all active G functions.

Each G group has its permanent location. The group number (No.) and the active G function of the G groups are only displayed if a G function is active.

Auxiliary functions

The "Auxiliary functions" window displays a list of all active auxiliary functions. Up to 5 M functions and 3 H functions are displayed.



Explanation of soft keys

Program on

Selects/deselects a parts program loaded in the editor.

New program

Creates a new EasyStep chain in the G code editor.

Editor

Calls a selected parts program in the G code editor so that it can be edited.

Block search

See section on "Block search"

Single block

See Section "Single block"

Influence

You can change the machining mode for a parts program during execution by means of the functions "Skip block" and "Program stop M01".

Skip block

Skip block: the block is not taken into account at program run.

M01

Program stop M01: when the function is active program execution is stopped at those blocks in which the M function M01 is programmed. The screen then displays "Stop: M00/M01 active". To continue execution, press the NC Start key. If the function is not activated, the M function M01 (from the parts program) is not taken into account.

Directory

A parts program overview is overlaid.

Program selection for execution is possible in this area (see also the Section "Parts program management").



Press the ETC key to display the functions R parameters and G code.

R parameters

The R parameters are displayed.

G code

Switchover to "G code": soft key is active

G code

Switchover from "G code": soft key is not active

7.10.2 G code editor



Function

The editor offers the following functions:

- Overwrite
- Select/copy/delete/insert block
- Search/replace
- Delete NC block

A parts program or sections or a parts program can only be edited in the G code editor if the respective blocks have not yet been executed. The blocks that can be changed are highlighted. If a parts program is selected and in "Reset" it can be changed completely.

Editor

The editor is opened.



The editor is closed with the "Recall" key, and you return to the program view.



Sequence of operations

Cursor pad



The direction keys are for positioning in the text.



You can page the screen contents forward or back with the "Page" keys.



You can delete the character selected by the cursor with the "Delete" key.



You complete a block by activating the "Input" key. "L_F" ("Line Feed") is generated. Only then is the entered block accepted for execution.



Explanation of soft keys

Editing the text

Overwrite

You change over between insert and overwrite mode by means of this soft key.

Mark block

This soft key marks the beginning of a block.

Position the cursor on the block end. The block is automatically selected.

Copy block

The soft key copies the selected block into the buffer. The block remains stored in the buffer even after the parts program has been changed.

Delete block

This soft key marks the block end and deletes the block in the program. The deleted block is stored in the buffer.

Insert block

This soft key inserts the deleted or copied block from the buffer to the text behind the cursor position.

Delete NC block

An NC block is deleted.

Renumber

Specify the following values in the "Renumber" box:

- Increment for the block numbers (e.g. 1, 5, 10)
- Number of the first block

To remove the step numbering, set the increment or block number to 0.

Searching the text

Search

The soft key opens the "Search term" dialog box.

Enter the term you are searching for. You can set the search direction forwards or backwards with the "Alternat." soft key.

Search Replace

Searches for the next of the term entered under "Search for" and replaces it with the new term.

7.10.3 Create new parts program



Sequence of operations

Precondition

PROGRAM mode is selected.



Activate the soft key "G code" in the extended soft key menu.



A dialog box is overlaid; enter the new file name for the parts program.



The new parts program is created under "Directory" and the G code editor is opened.



Now enter the new parts program blocks.

/PG/, Programming Guide, Fundamentals

/PGA/, Programming Guide, Advanced

7.10.4 Inserting program blocks



Function

Simple contour elements (STRAIGHT, OBLIQUE, or CIRCLE mode) and turning cycles (CYCLE or STOCK REMOVAL mode) can either be inserted into the parts program in the "Current block" window in all modes (except for MANUAL), or in the G code editor in PROGRAM mode.



You cannot create complex contours (CONTOUR mode) within a G code parts program.



Sequence of operations

Precondition

A parts program is loaded in the "Current block" window or in the G code editor.



Position the cursor on the block preceding the location where you would like to insert a new block.



Select the desired mode (here by using the mode switch).

Enter the parameters in the input screenform.



Accept the new block into the parts program.

7.10.5 Editing program blocks



Function

Simple contour elements and turning cycles can either be inserted into the parts program in the "Current block" window in all modes (except MANUAL) or in the G code editor in PROGRAM mode.



Sequence of operations

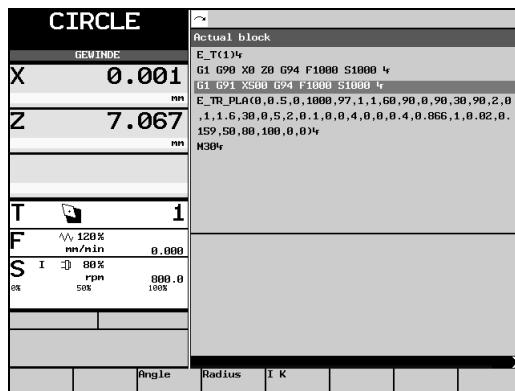
Precondition



Current block

A parts program is loaded in the "Current block" window.

Position the cursor on the block you would like to modify.



Press the "Input" key. The associated ManualTurn input screenform is overlaid.

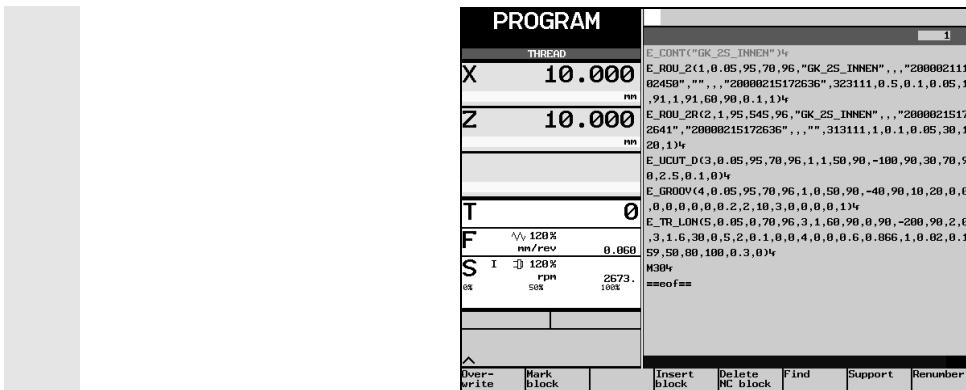
G code editor

Precondition



A parts program is loaded in the "G code editor".

In the parts program, select the block which is to be changed (here a thread undercut to DIN 76).



Support

Press the "Support" soft key. The associated ManualTurn input screenform is overlaid.



Enter the required parameters and validate the changes by pressing the "Accept" soft key.



You can abort the process by pressing the "Recall" key.



Please note that a recompilation into the ManualTurn input screens only takes place if the programming syntax of the ManualTurn cycle language was adhered to.

Contour elements

Examples

The contour elements STRAIGHT, CONICAL and CIRCLE are programmed as G code as follows (valid for CONICAL and CIRCLE without angle programming):

STRAIGHT

```
E_T (1) LF
G1 G91 X100 G95 F1 S300 LF
```

CONICAL

```
E_T (2) LF
G1 G90 X50 Z-50 G95 F1 S300 LF
```

CIRCLE

```
E_T (3) LF
G3 G91 X0.67 Z-2.5 CR=5 G95 F1 S300 LF
```

If no tool change takes place between consecutive NC blocks, the tool that was programmed first acts modally for the following blocks!

Example

```
E_T (1) LF
G1 G91 X100 G95 F1 S300 LF
G1 G90 X50 Z-50 G95 F1 S300 LF
```

Turning cycles

The following cycles are generated from EasyStep and displayed in G code:

Cycle name	Definition
E_ABR_CR	Circular arc: start angle (α), end angle (β), radius
E_AER_CR	Circular arc: start angle (α), aperture angle (E), radius
E_DBR_CR	Circular arc: start angle (D), end angle (β), radius
E_DER_CR	Circular arc: start angle (D), aperture angle (E), radius
E_DRILL	Drilling (deep hole drilling)
E_DR_CIR	Drilling hole circle (face end, peripheral surface)
E_GROOV	Grooving cycle
E_ROUGH	Stock removal cycle: groove
E_ROU_GP	Stock removal cycle: contour (geometry processor)
E_T(n)	Tool change with n = 0, ..., 99
E_TAP	Tapping
E_TA_CIR	Tapping hole circle (face end, peripheral surface)
E_TR_CON	Thread cutting: taper
E_TR_LON	Thread cutting: longitudinal
E_TR_PLA	Thread cutting: face
E_UCUT_D	Thread undercut: to DIN76
E_UCUT_E	Thread undercut: form E DIN509
E_UCUT_F	Thread undercut: form F DIN509
E_UCUT_T	Thread undercut: freely defined
E_XA_CON	Conical: X, angle
E_ZA_CON	Conical: Z, angle

Parts Program Management

8.1	General	8-150
8.2	Select a file.....	8-151
8.3	Delete a file	8-151
8.4	Storing thread undercut and thread cycles	8-152
8.5	Insert a contour in EasyStep machining sequence	8-152
8.6	Rename/copy file	8-153
8.7	Read out a file to an external medium	8-153
8.8	Read in a file	8-154
8.9	Error/transmission log	8-154

8.1 General



Function

A file management function is available to you in the following modes:

- CONTOUR
- PROGRAM
- G code
- CYCLE (for thread and thread undercut cycles only)

Directory

The "Directory" soft key can be found in CONTOUR, PROGRAM and G code modes in the basic soft key menu; with thread cycles it is in the extended soft key menu.

The following are possible for the files:

- selecting,
- copying/renaming,
- deleting and
- exporting and importing via the RS-232 interface.
- for thread and thread undercut cycles: Save input parameters in dialog screenform.
- in CONTOUR mode: Inserting a stored contour into an EasyStep machining sequence

An error log informs you whether any errors occurred while the RS-232 interface was in use.

8.2 Select a file



Sequence of operations

Precondition

You have opened the directory.



Select the file you require in the directory.

The name, size and dates of creation and modification of each file are displayed.

Call file

Precondition

The file is selected.



Press the "Input" key.

This takes you back to the mode from which you called program management (PROGRAM/CONTOUR/dialog form of the thread or thread undercut cycle).

You can now process the file.

8.3 Delete a file



Sequence of operations

Precondition

You have selected the file that you wish to delete.



Select the soft key labeled "Delete".

Answer the prompt with the soft key "Return" (function is abandoned) or with the soft key "Confirm" (function is executed).

8.4 Storing thread undercut and thread cycles**Function**

With the soft key "Store input" in menu "Directory", you can store the parameters you enter in the dialog screenform for the thread undercut and thread cycles in a file.

**Sequence of operations****Precondition**

You have parameterized a thread undercut cycle or a thread cycle.

Directory

Press the soft key "Directory" in the extended soft key menu.

Store
input

Press the soft key "Store input"

and

OK ✓

enter a file name in the window "Insert new worksheet". Store the file name by pressing the "Input" key or "OK" soft key.

Back

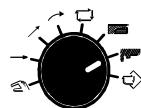
Press soft key "Back" to abandon the function.

8.5 Insert a contour in EasyStep machining sequence**Sequence of operations****Precondition**

You have selected the EasyStep machining sequence you wish to execute.

▲ ▼

Use the cursor keys to select the step after which the contour is to be inserted.



Change to CONTOUR mode.

Directory

Press the soft key "Directory".

▲ ▼

Use the cursor keys to select the desired contour in the directory.

INPUT Accept

Activate the "Input" key and the "Accept" soft key. The contour is inserted at the desired position in the EasyStep machining sequence.

8.6 Rename/copy file



Precondition

Copy

Rename



Sequence of operations

The file is selected.

Select the function you require by choosing the appropriate soft key.

Enter a new file name and accept by pressing the "Input" key or "OK" soft key.

Press soft key "Back" to abandon the function.

8.7 Read out a file to an external medium



Sequence of operations

Position the cursor in the directory on the file you want to export.

Read-out
RS-232-C

Start

Stop



Select the soft key labeled "Read-out RS-232-C".

Start transmission from data device.

Select the soft key "Start".

The selected file is now exported.

The name of the file and the number of transmitted bytes are displayed in the "Read-out RS-232-C" window.

You can interrupt the data transmission by selecting the "Stop" key.

Selecting the "Back" soft key returns you to the higher-level directory overview. The "Read-out RS-232-C" function remains active if a RS-232-C transmission has already been started.

When you press the soft key labeled "All files", all the files stored in the program directory are selected for transfer.

NC start is not possible while the files are being exported.



8.8 Read in a file



Sequence of operations

Precondition

You have selected the interface for your data device correctly.

**Read-in
RS-232-C**

Select the soft key labeled "Read-in RS-232-C".

The "Read-in RS-232-C" window is opened.

Start the transfer from the other side (on the data device).

Start

When you select the soft key "Start", the data are copied in the path you have specified.

The names of the imported files and the number of bytes transmitted are displayed.

Stop

You can interrupt the data import operation by selecting the "Stop" soft key.

Start

Press the soft key labeled "Start" again to restart import of the files, and start the operation again at the opposite end (on the data device).

X
Back

Selecting the "Back" soft key returns you to the higher-level directory overview. The "Read-in RS-232-C" function remains active if a RS-232-C transmission has already been started.



NC start is not possible while the files are being imported.

8.9 Error/transmission log



Function

Select error log

**Error
log**

When you press the soft key "Error log" a log confirming successful file transfer is output.

- For files to be exported
 - the file name including path name and
 - a status acknowledgment.
- For files to be imported
 - the file name and the first line, which generally contains the path, as well as
 - a status acknowledgment.

General Functions

9.1	Simulation and simultaneous recording	9-156
9.1.1	Simulation	9-158
9.1.2	Simultaneous recording	9-159
9.1.3	Dry run.....	9-159
9.2	Teach In	9-161
9.2.1	Selecting the Teach In function.....	9-161
9.2.2	Deselecting "Teach In".....	9-162
9.2.3	Continuing "Teach In".....	9-162
9.2.4	Transferring machining steps to system	9-163
9.2.5	Transferring auxiliary functions to system.....	9-165
9.3	Standard CNC operation.....	9-166

9.1 Simulation and simultaneous recording



Function

You can perform the following actions:

- In fast motion graphically display each axis motion (simulation)
- Record the results of an EasyStep machining sequence on the screen during the machining operation (simultaneous recording)
- Display each axis motion graphically (dry run).

Display elements

The colors used in the graphics display area represent the following:

- | | | |
|----------|---|--|
| • Red | = | Travel path in path feedrate |
| • Green | = | Travel path in rapid traverse |
| • Yellow | = | Cross-hair, tool nose,
Symmetry axis of workpiece |

Cross-hair

You can use the cross-hair to

- select the center of the zoomed area and
- set the measuring point.

Coordinate system

The orientation of the axes (coordinate system) is defined in the machine data.

Please follow the advice of the machine manufacturer.

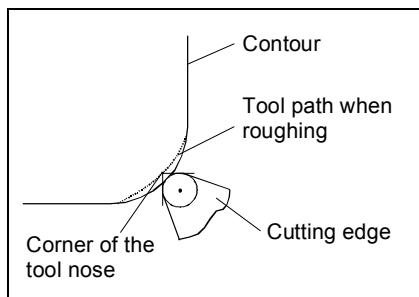
Tool edge

The position of the tool edge corresponds to that defined in the menu "Tool offset" under the soft key labeled "Tool".

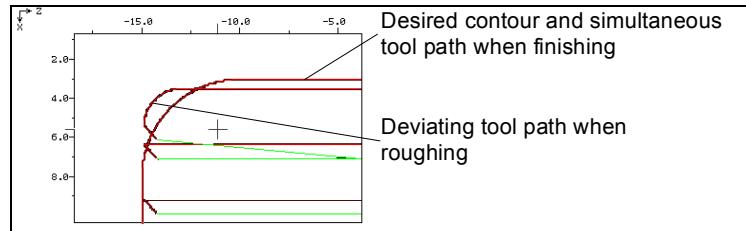
The tool path of the currently edited EasyStep machining sequence is simulated. The start point of the tool nose corresponds to the start point of the machine tool axes.

The graphic display of the tool path may differ from the real tool path on the machine during roughing of inside or outside corners (with or without rounding).

Roughing is always performed without tool nose radius compensation. The broken-line graphics show the tool path during corner machining at the corner of the tool nose.



Finishing is always performed with tool nose radius compensation. The tool path in the broken-line graphics is equivalent to the point of contact between the tool nose and the contour.



Moreover, the graphic display of the tool path may differ from the real tool path on the machine on selection or deselection of tool nose radius compensation.

9.1.1 Simulation**Function**

Each programmed traversing movement can be displayed on the screen in fast motion with the "Simulation" function, without the axes moving.

The total time displayed (in hours/minutes/seconds) is approximately equivalent to the time the programmed traverse movements require when executed on the machine.

**Sequence of operations**

Simula-
tion

The "Simulation" soft key can be selected in each parameterization screenform.

The following soft key functions are available to you:

Return to
original

Select this soft key to return to the original display.

Zoom
Auto

Select this soft key for a window-sized display of all travel paths in the graphic display area.

Zoom
+

Zoom
-

You can increase and decrease the resolution of the screen by pressing the soft keys "Zoom+" and "Zoom-". Navigate the cross-hair to the center of the screen area you wish to display with the cursor keys.

Press the soft key "Cursor fine" to control the increment size of a cursor key movement.

- Soft key is selected:

Cursor moves in "fine" increments

- Soft key is not selected:

Cursor moves in "large" increments

Cursor
fine

Cursor
fine

9.1.2 Simultaneous recording**Function**

The results of an EasyStep machining sequence can be displayed on the screen in real time by means of the "Simultaneous recording" function.

**Sequence of operations**

Simultaneous rec.

You can select the "Simultaneous recording" function in the extended soft key menu of every operating mode.

Press the "Spindle start" and "NC start" keys to start execution.

The same soft key functions are available as under "Simulation" (see Section "Simulation"). The function "Delete screen" is also available.

Delete screen

Select this soft key to delete the current screen contents.

9.1.3 Dry run**Function**

You can simulate a programmed EasyStep machining sequence on the screen by activating the "Dry run" function.

**Explanation of soft keys**

Dry run

All axis movements on the machine tool are disabled.

The "Dry run" soft key only appears if no program is active, that is, if the program is in RESET state (press RESET key).

Dry run feedrate

When you press the soft key "Dry run feedrate", the feed movements are simulated at the dry run speed specified in the machine data.

**Additional notes**

In a dry run the machine and auxiliary functions are output by the NC and can be suppressed by the machine manufacturer, if required. Please follow the advice of the machine manufacturer.



Sequence of operations

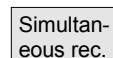
Selecting dry run

Precondition

You have selected the EasyStep machining sequence.



Select the extended soft key menu with the "ETC" key.



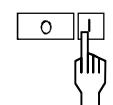
Select the soft key labeled "Simultaneous recording".



Select the soft key labeled "Dry run".



If necessary, activate the dry run speed by pressing the soft key "Dry run feedrate".

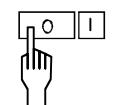


Initiate the program dry run by pressing the "NC start" key.

Deselecting dry run

Precondition

Program is active

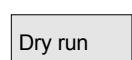


Press the "NC Stop" key twice

and



the soft key "Abort"



Deselect the dry run function by pressing the soft key "Dry run".

The control system returns to the state it was in before Dry Run was selected.



- The dry run function can be selected/deselected provided that no NC start command is active. The spindle is simulated in the actual-value display in the dry run.
- You can terminate graphic simulation of a program during NC operation with the key "Recall".

9.2 Teach In



Function

The "Teach In" function allows you to transfer machining steps that are being performed on the machine tool to the control system. These machining steps can be manually controlled movements (e.g. using the control stick) or programmed steps (e.g. cycles) or complete EasyStep programs.

All steps are transferred to a program, which allows you to automatically create programs that can be reproduced as often as required.

If the machine is equipped with a turret, you can only use the "Teach In" function, if the turret repositioning is also set with the function. This must be set up by the machine manufacturer.

Please follow the advice of the machine manufacturer.

9.2.1 Selecting the Teach In function

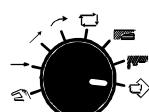


It is recommended that all tools required for a machining operation are measured in advance and their tool nose radii and tool positions entered.

Before you select the function "Teach In" move the slide manually to a suitable start position.



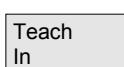
Sequence of operations



An operating mode is selected.



Select the extended soft key menu with the "ETC" key.



Activate the function "Teach In" by pressing soft keys

and



Enter the name of the new Teach In program in the window "Teach new program sequence".

TEACH 2 is now displayed in the status field "Teach In", i.e. the first step in the sequence is always a comment (with the date and time). The second step (TEACH 2) specifies the position at which Teach In was selected.

The workpiece can now be machined step by step and in this way the program created. After each NC start the counter in the status field is incremented by the stored step as TEACH 3, TEACH 4, etc.

9.2.2 Deselecting "Teach In"



Open up the extended soft key menu with the "ETC" key in any operating mode and



deactivate Teach In with the soft key "Teach In".

9.2.3 Continuing "Teach In"



You can append the Teach In steps to an existing EasyStep program.



Press soft key "Continue teach" to call up the directory of PROGRAM mode. Using the cursor keys, select the EasyStep program in which the Teach In steps are to be appended to the machining sequence. The "end of program" step in the selected program is deleted and the first Teach In step starts with a comment block.

The counter (TEACH 1, TEACH 2, etc.) in the Teach In status field only indicates the number of taught steps and not the total number of program steps in the existing EasyStep program.

9.2.4 Transferring machining steps to system

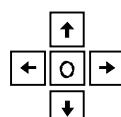
In general, all programmed motions that are executed with "NC start" in modes STRAIGHT, CONICAL, CIRCLE, CONTOUR, CYCLE, STOCK REMOVAL and PROGRAM are automatically stored.

Programmed spindle speed values and feedrates are multiplied by the percentage of the override switch in question at the end of a movement and transferred to the EasyStep program.

In manually performed movements a distinction is made between:

- Fast adjustment of the slide
- Manual movements in the transverse and longitudinal directions
- Slide movements in MANUAL mode using the control stick

Fast adjustment of the slide



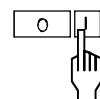
using the control stick and the rapid traverse key.



Additional notes

The program step is automatically stored in PROGRAM mode if the relevant machine data has been set. Please follow the advice of the machine manufacturer.

Manual movements in the transverse and longitudinal directions



When you have performed the manual movements press the "NC Start" key.

Teach rapid traverse

The following three soft keys appear in the soft key menu:

Store movement as rapid traverse.

Teach feedrate

Store movement as feedrate.

The feedrate is stored in a machine data. Please follow the advice of the machine manufacturer.

No Teach

Do not store movement.



Teach RT Teach F

Additional notes

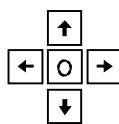
You can accept the movements directly by pressing either the key "Teach RT (rapid traverse)" or the key "Teach F (feedrate)" on the machine control panel.

Please follow the advice of the machine manufacturer.

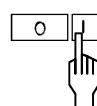
Slide movements in MANUAL mode using the control stick



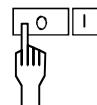
Select MANUAL mode.



Select the direction (straight or conical) with the control stick.



Start the movement by pressing the "NC Start" key.



You can stop the movement by pressing the "NC Stop" key or by returning the control stick to the center position. The movement is stored automatically.

Programmed spindle speed values and feedrates are multiplied by the percentage of the override switch in question at the end of a movement and transferred to the EasyStep program.



You cannot execute the following functions when Teach In is active.

- Zero offset
- Preset
- Enter tool offset data
- Measure tool.

9.2.5 Transferring auxiliary functions to system



Additional notes

In addition to the machining steps, machine functions, such as spindle on/off, coolant on/off, etc., can also be stored if the machine manufacturer has provided these functions.

Please follow the advice of the machine manufacturer.

9.3 Standard CNC operation

Area switchover



Press the key "Area switchover" to switch between the standard CNC operator interface and the ManualTurn interface.



By pressing the "Area switchover" key twice, you can switch between the operating areas that were last selected, e.g. from the "Parameter" operating area to the "Machine" operating area and vice versa.

Note

- The "Area switchover" key will only work if the function has been implemented internally by the machine manufacturer.
- The control must be in RESET status.
Please follow the advice of the machine manufacturer.



Please note the following points when switching between ManualTurn and standard CNC mode:

- NC start is not active
- The spindle is stationary
- The zero offsets ZO1...ZO4 in ManualTurn are not identical to the G commands G54...G57 in CNC mode.
- Preset in ManualTurn is not identical to Preset in standard CNC mode.
- EasyStep programs created in ManualTurn can also be operated in standard CNC mode. You will find the EasyStep programs in the standard CNC user interface under "Programs" in the EasyStep directory.
- EasyStep programs that have been changed in standard CNC mode cannot be executed in ManualTurn.

Please read the following documentation before switching from ManualTurn to standard CNC mode:

- References:**
- /BA/, Operator's Guide
 - /PG/, Programming Guide Fundamentals
 - /PGA/, Programming Guide Advanced
 - /PGZ/, Programming Guide Cycles

Intervention in the Machining Process

10.1	Aborting a machining operation	10-168
10.2	Repositioning.....	10-169
10.3	Saving manual offset with "Store offset" function.....	10-170

10.1 Aborting a machining operation

You can cancel a machining operation that has already started in:

MANUAL – Straight mode

with the "NC stop" key

or

MANUAL – Conical mode

with the control stick in the OFF position (center position).

Operating modes with automatic motion switch-off (not MANUAL mode)

Interrupt movement with the "NC stop" key

The start and stop lamps light up



When you press the "NC stop" key again the soft key menu containing the soft key "Abort" is displayed (for a limited time).



Start is reset when you press the "Abort" soft key.

The start and stop lamps go out.

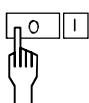
10.2 Repositioning



Sequence of operations



Select operating mode with automatic switch-off (not MANUAL mode).



Interrupt movement with the "NC stop" key



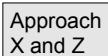
Retract tool from workpiece with Setup



The position display switches from "Distance to go" to "Retract" and the distance to the point of interruption is displayed.



Press the "NC start" key.



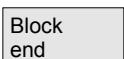
Approach in the X and Z direction



Approach parallel to the axis



Approach contour at the beginning of the block



Approach contour at the end of the block



When you press the Start key again the tool first approaches the point of interruption and then continues the interrupted movement.



When retract "0.000" is reached the distance to go is again displayed.

The automatic approach to the point of interruption is performed at the feedrate of the interrupted movement which can be overridden the JOG keys.

10.3 Saving manual offset with "Store offset" function**Function**

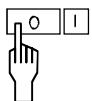
With the function "Store offset" you can save the path of retraction as a manual offset during an interruption. This offset is then included in all following traverse movements.

**Sequence of operations**

Select operating mode with automatic switch-off (not MANUAL mode)



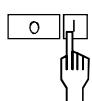
Interrupt movement with the "NC stop" key



Set manual offset with Setup



The position display switches from "Distance to go" to "Retract" and the distance to the point of interruption is displayed.



Press the "NC start" key.



Select soft key labeled "Store offset"



When you press Start the interrupted movement is continued offset in relation to the contour.



"Manual offs." appears in the status field to inform the operator that a manual offset is active. The manual offset can be reset in the "Offset" menu (see Section "Preparatory Functions for Machining").

Alarms and Messages

11.1	Alarms and messages in ManualTurn cycles	11-172
11.1.1	Error handling in cycles	11-172
11.1.2	Overview of cycle alarms	11-172
11.1.3	Messages in cycles	11-173
11.2	Alarms with ManualTurn	11-174
11.2.1	Overview of alarms	11-174
11.2.2	Selecting the alarm/message overview	11-174
11.2.3	Description of alarms	11-175

11.1 Alarms and messages in ManualTurn cycles

11.1.1 Error handling in cycles

If an illegal status is recognized in the cycles an alarm is generated and machining aborted.

Alarms 61000 to 62999 are generated in the cycles. The deletion criterion for this range is NC_RESET. This aborts block preparation in the NC. The error text that is displayed together with the alarm number gives more detailed information about the cause of the fault.

11.1.2 Overview of cycle alarms

The error numbers are classified as follows:

6	–	X	–	–
---	---	---	---	---

- X=0 General cycle alarms
- X=1 Deep hole drilling cycle alarms
- X=6 Turning cycle alarms

The table below lists all the errors that occur in the cycles, where they occur and how they can be remedied.

Alarm number	Alarm text	Cycle	Explanation, Remedy
61101	"Reference plane incorrectly defined"	Deep hole drilling	Either different values must be entered for the reference plane and return plane for incremental definition of the depth or an absolute value must be entered for the depth.
61107	"First drilling depth incorrectly defined"	Deep hole drilling	First drilling depth is not compatible with total drilling depth.
61244	"Thread pitch change causes undefined thread"	Thread cutting	Check thread geometry.
61602	"Tool width incorrectly defined"	Grooving in STOCK REMOVAL	Grooving tool is larger than programmed groove width
61604	"Active tool violates programmed contour"	Cutting in STOCK REMOVAL	Contour violation in relief cut elements caused by the clearance angle of the tool used, i.e. use a different tool or check the programmed contour
61605	"Contour incorrectly programmed"	Cutting in STOCK REMOVAL	Illegal relief cut element detected

Alarm number	Alarm text	Cycle	Explanation, Remedy
61606	"Error during contour preparation"	Cutting in STOCK REMOVAL	During preparation of the contour an error was detected, this alarm is always output with NCK alarm 10930 ... 10934, 15800 or 15810 (see Diagnostics Guide)
62200	"Start spindle"	Thread	The spindle must be running before the thread can be machined.

11.1.3 Messages in cycles

The cycles output messages in the dialog line of the control. These messages do not interrupt processing.

Messages provide you with information about cycle behavior and processing progress. They are usually retained over a processing section or until the end of the cycle, e.g. messages about the current thread during machining as a longitudinal or face thread.

11.2 Alarms with ManualTurn

11.2.1 Overview of alarms

If an illegal status is recognized in ManualTurn an alarm is generated and machining aborted.

The error text that is displayed together with the alarm number gives more detailed information about the cause of the fault.

Overview of alarms

100000–100999	Basic system	
101000–101999	Diagnosis	
102000–102999	Services	
103000–103999	Machine	
104000–104999	Parameters	
105000–105999	Programming	
106000–106999	Reserved	
107000–107999	OEM	
110000–110999		Reserved
111000–112999	ManualTurn	
120000–120999		Reserved



Danger

Carefully check the installation using the description of the alarm that has been triggered. Eliminate the cause of the alarm and acknowledge the alarm in the way described. Non-observance could result in danger to the machine, workpiece, stored settings and, in certain circumstances, to your health.

If you are in CNC mode, refer to the following alarm descriptions:

Reference: /DA/ Diagnostics Guide

11.2.2 Selecting the alarm/message overview



Function

You can display and acknowledge alarms and messages.



Sequence of operations



The alarm/message overview displays all queued alarms and messages, together with the numbers, date/time and a description.

You acknowledge the alarm/message with the "NC Reset" soft key.

11.2.3 Description of alarms

111 001

Explanation

Non-interpretable step in line %1

%1 = Line number

The step is not an element of ManualTurn.

Reaction

Alarm display

EasyStep sequence is not loaded

Remedy

Delete program step or edit program in operating area PROGRAMS of SINUMERIK 840D or 810D (CNC mode).

111 002

Explanation

Not enough memory

Abort in line %1

%1 = Line number

EasyStep sequence contains too many steps

Reaction

Alarm display

EasyStep sequence is not loaded

Remedy

Edit the program in the PROGRAMS operating area of the SINUMERIK 840D or 810D (CNC mode).

111 003

Explanation

ManualTurn: %1

%1 = Error number

Internal system message via the user interface of ManualTurn

Reaction

Alarm display

Remedy

Acknowledge error and inform Siemens.

111 004

Explanation

File does not exist or is incorrect: %1

%1 = Name of the file/contour

EasyStep sequence cannot interpret a step with contour programming.

Contour does not exist in directory.

Reaction

Alarm display

NC start disable

Remedy

Load contour in directory.

111 005

Explanation

Error on interpreting the contour %1

%1 = Name of the contour

Contour is incorrect

Reaction

Alarm display

NC start disable

Remedy

Check machining sequence of contour

111 006

Explanation

Maximum number of contour elements exceeded %1

%1 = Name of the contour

Max. permissible number of 50 contour elements exceeded during interpretation of machining sequence of a contour.

Reaction

Alarm display

Remedy

Check machining sequence of a contour and correct, if necessary.

111 007

Explanation	Error in line %1 %2
	%1 = Line number
	%2 = Description of error
Reaction	Alarm display NC start disable
Remedy	Remove the error.

111 008

Explanation	Spindle not synchronized
Reaction	Alarm display
Remedy	Allow spindle to turn for at least one revolution (M3, M4).

111 009

Explanation	Insert new tool: T%1
	T%1 = Tool number
	Tool change program requests a new tool
Reaction	Alarm display NC Stop
Remedy	Insert new tool

111 010

Explanation	Teach In Cancellation: Log overflow
Reaction	Teach in procedure canceled. Alarm display
Remedy	Teach in file closed. The refresh rate must be increased by 100 to 200 ms in MD 9606: \$MM_CTM_SIMULATION_TIME_NEW_POS.

111 100

Explanation	Wrong position programmed for spindle
	A position outside the range 0 - 359.999 has been programmed for a modulo axis.
Reaction	Alarm display Interface signals are set Interpreter stop NC start disable
Remedy	Program position in the range 0 - 359.999. Reset alarm with RESET key. Restart program.

111 105

Explanation	SPCON, SPOS or SPOSA has been programmed. These functions require at least one measuring system. According to MD: NUM_ENCS the machine axis/spindle has no measuring system.
Reaction	Alarm display Interface signals are set Interpreter stop NC start disable
Remedy	Add a measuring system. Reset alarm with RESET key. Restart program.

111 106

Explanation

Spindle does not stop on block change

The displayed spindle has been programmed as a spindle or an axis although a positioning operation from a previous block is still running (with SPOSA ... spindle positioning beyond end of block).

Example

N100 SPOSA [2] = 100

:

N125 S2 = 1000 M2 = 04 ;Error if spindle S2 from
;block N100 is still turning!

Reaction

NC start disable

NC stop on alarm

Alarm display

Interface signals are set

Remedy

Before reprogramming the spindle/axis after a SPOSA instruction, trigger a wait for the programmed spindle position with the WAITS command.

Example

N100 SPOSA [2] = 100

:

N125 WAITS (2)

N126 S2 = 1000 M2 = 04

Reset alarm with RESET key. Restart program.

111 107

Explanation

Reference mark not found

On referencing, the spindle rotated further than specified in axis-specific MD 34 060 REFP_MAX_MARKER_DIST without receiving a reference mark signal. This is checked on spindle positioning with SPOS or SPOSA if the spindle has not yet turned with speed control (S=...).

Reaction

NC start disable

NC stop on alarm

Alarm display

Interface signals are set

Remedy

Check and correct MD 34 060 REFP_MAX_MARKER_DIST. The value entered specifies the path in [mm] or [degrees] between 2 zero marks.

Reset alarm with RESET key. Restart program.

111 108

Explanation

No transition from speed control to position control

- A spindle stop (SPOS/SPOSA) has been programmed or the position control of the spindle has been activated with SPCON but no spindle encoder is defined.
- When the position control is activated the spindle speed is greater than the limit speed of the measuring system.

11.2 Alarms with ManualTurn

Reaction	NC start disable NC stop on alarm Alarm display Interface signals are set
Remedy	Spindle without mounted encoder: NC language elements that require encoder signals must not be used. Spindle with mounted encoder: Enter number of spindle encoders used via MD NUM_ENCS . Reset alarm with RESET key. Restart program.
111 109	Configured positioning speed too fast
Note	See alarm description for 111 107
111 110	Velocity/speed is negative
Note	See alarm description for 111200
111 111	Set speed is zero
Explanation	The programmed spindle speed setpoint is zero.
Reaction	Alarm display
Remedy	Set permissible spindle speed setpoint.
111 112	Illegal gear step
Explanation	Illegal gear step requested by PLC.
Reaction	Alarm display
Remedy	Check PLC program and axis-specific NC machine data.
111 115	Programmed position not reached
Note	See alarm description for 111200
111 126	Absolute minus value impossible
Note	See alarm description for 111200
111 127	Absolute plus value impossible
Note	See alarm description for 111200
111 200	Positioning error spindle
Explanation	Alarms 111110, 111115, 111126, 111127 and 111200 can occur on spindle start/stop.
Reaction	Alarm display
Remedy	Inform the service department. Please contact your regional Siemens office.
111 300	NC start key defective
Explanation	Checkback signal to PLC user program that NC start key is defective, i.e. N.C. and N.O. signal = 1

Reaction	Alarm display NC start disable
Remedy	Replace key
111 301	NC stop key defective
Explanation	Checkback signal to PLC user program that NC stop key is defective, i.e. N.C. and N.O. signal = 1
Reaction	Alarm display NC start disable
Remedy	Replace key
111 302	Spindle start key defective
Explanation	Checkback signal to PLC user program that spindle start key is defective, i.e. N.C. and N.O. signal = 1
Reaction	Alarm display NC start disable
Remedy	Replace key
111 303	Spindle stop key defective
Explanation	Checkback signal to PLC user program that spindle stop key is defective, i.e. N.C. and N.O. signal = 1
Reaction	Alarm display NC start disable
Remedy	Replace key
111 304	Connection to PLC broken
Explanation	Checkback signal to PLC user program that connection to MANUALTURN is broken.
Reaction	Alarm display MANUALTURN-PLC terminated
Remedy	Check PLC user program.
111 305	Asynchronous subroutine not processed
Note	In the asynchronous subroutine, internal settings in the NC are triggered by the user interface. If any of the alarms 111 306 to 111 310 is triggered, these settings cannot be executed.
Reaction	Alarm display
Remedy	Press NC reset.
111 306	Error on selection or deselection of constant cutting speed
111 307	Error on resetting handwheel offset
111 308	Error on setting spindle speed upper limit
111 309	Error on selecting tool
111 310	Error on selecting zero offset

111 400**Unknown PLC error**

Explanation

An error signal unknown to the user interface has been output by the PLC.

Reaction

Alarm display

NC start disable

Remedy

Activate POWER ON, inform Siemens.

111 410**Tool %1 was created**

Explanation

As ManualTurn starts, it check that all standard tools are present. If any are not, the missing tools are generated automatically. If more than one tool is created, the tools are output in a combined message.

Meaning:

Example:

%1 Number of tool which was created,

5

%1 First and last tool which were created,

5...16

Reaction

None

Remedy

None

111 411**%1 tool(s) could not be created**

Explanation

As ManualTurn starts, it check that all standard tools are present. If any are not, the missing tools are generated automatically.

The specified number of tools could not be created in this case.

Reaction

Alarm display

NC start disable

Remedy

Increase machine data 18082 \$MM_NUM_TOOL by the specified amount.

111 420**Error at inch/metric switchover!****Check all data!**

Explanation

Data switchover was not fully terminated for inch/metric switchover.

Reaction

Alarm display

NC start disable

Remedy

Check the following data:

- Display machine data:

MD9004: \$MM_DISPLAY_RESOLUTION

MD9600: \$MM_CTM_SIMULATION_DEF_X

MD9601: \$MM_CTM_SIMULATION_DEF_Y

MD9602: \$MM_CTM_SIMULATION_DEF_VIS_AREA

MD9603: \$MM_CTM_SIMULATION_MAX_X

MD9604: \$MM_CTM_SIMULATION_MAX_Z

MD9605: \$MM_CTM_SIMULATION_MAX_VIS_AREA

MD9616: \$MM_CTM_TEACH_HANDW_FEED_P_MIN

		MD9617: \$MM_CTM_TEACH_HANDW_FEED_P_REV MD9620: \$MM_CTM_CYCLE_SAFETY_CLEARANCE MD9633: \$MM_CTM_INC_DEC_FEED_PER_MIN MD9634: \$MM_CTM_INC_DEC_FEED_PER_ROT MD9637: \$MM_CTM_MAX_INP_FEED_P_MIN MD9638: \$MM_CTM_MAX_INP_FEED_P_ROT MD9639: \$MM_CTM_MAX_TOOL_WEAR MD9648: \$MM_CTM_ROUGH_O_RELEASE_DIST MD9649: \$MM_CTM_ROUGH_I_RELEASE_DIST MD10240: \$MN_SCALING_SYSTEM_IS_METRIC MD20150 [12]: \$MC_GCODE_RESET_VALUES
		<ul style="list-style-type: none"> • Tool data: Length X, length Z, radius wear lengths X and Z, vconst • Zero offsets: Position in X, Z
Note		This alarm can only occur with hardware faults.
111 430		Program not loaded. Error when converting old cycles into G code. No NC memory.
Explanation		In previous ManualTurn versions, STRAIGHT, CONICAL and CIRCULAR steps were saved as cycles. They are now saved as G-code (CONICAL, CIRCLE without angle programming). When a sequence is loaded, a check is performed to see if any old cycles were used. If any old cycles are found, the chain is converted and saved again in the NC. In the event of an error (memory full), this alarm is output.
Reaction		Sequence is not loaded.
Remedy		To create a new sequence, there must be sufficient free memory to store the original sequence and the backup copy.
111 900		It is only possible to start in the basic display
Explanation		A G-code program can only be started from the basic display for a mode (except for MANUAL).
Reaction		Alarm display
Remedy		Change to the basic display for a mode (except for MANUAL). Start single-step with NC start.
111 901		Contour is contained in the current program Processing not enabled
Explanation		There is a contour in the current EasyStep sequence that must not be altered.
Reaction		Alarm display
Remedy		Stop execution. Reload EasyStep sequence and edit as required.

11.2 Alarms with ManualTurn

111 902

Explanation

Reaction

Remedy

Start with valid reference point only

The axes have no valid reference point.

Alarm display

Reference all axes

111 904

Explanation

Reaction

Remedy

4th axis not configured, that is, no powered tool possible

MD9643 CTM_ENABLE_DRIVEN_TOOL was activated, without a 4th axis being known to the system.

Alarm display

Create a 4th axis. To do this, change the following machine data:

- Channel-specific
20070 \$MC_AXCONF_MACHAX_USED [3]=4
- Axis-specific for the 4th axis
30300 \$MA_IS_ROT_AX=1
30310 \$MA_ROT_IS_MODULO=1
30320 \$MA_DISPLAY_IS_MODULO=1
30350 \$MA_SIMU_AX_VDI_OUTPUT=1
35000 \$MA_SPIND_ASSIGN_TO_MACHAX=2

112 999

Faulty graphics data

Exit graphic and start again

Explanation More data have been generated than the operator interface can read.

Reaction Alarm display

Stop graphic

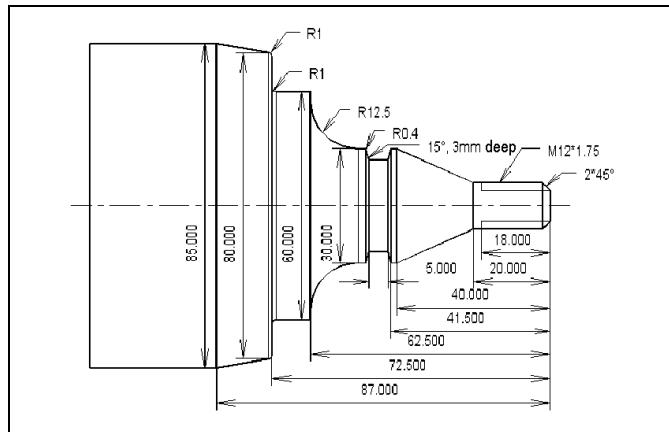
Remedy Deselect graphic and select again.

Examples

- | | | |
|------|---|--------|
| 12.1 | Example 1: External machining with groove and thread | 12-184 |
| 12.2 | Example 2: External machining with sphere | 12-189 |
| 12.3 | Example 3: External machining with thread undercuts and grooves | 12-194 |
| 12.4 | Example 4: External machining with thread undercut and groove | 12-200 |

12.1 Example 1: External machining with groove and thread

Workpiece drawing



Blank

Dimensions: Ø 85x120mm

Material: Aluminum

Tools

T1: 80° Roughing tool R 0.8

T2: 35° Finishing insert R 0.4

T3: Plunge-cutter 3mm

T4: Threading tool 1.75

T5: 35° Roughing insert R 0.8

Program

1. Create new program

Call PROGRAM mode and press the soft key. Enter program name (here: Part_1) and confirm.

2. Stock removal cycle for surface turning

- Call STOCK REMOVAL mode and press the soft key.

• Set parameters:

T1 F: 0.2 mm/rev **S:** 180 m/min

Position: Set cutting position

Parallel to X axis

Reference point **X0:** 87 **Z0:** 1

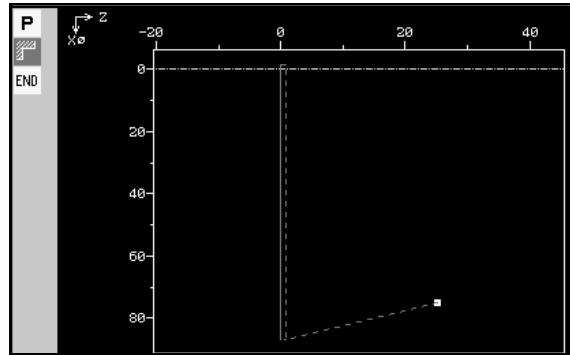
End point **X1:** -1.6 **Z1:** 0

Infeed depth **D:** 2 inc

Finishing allowance **Ux:** 0 **Uz:** 0.1

- Press soft key

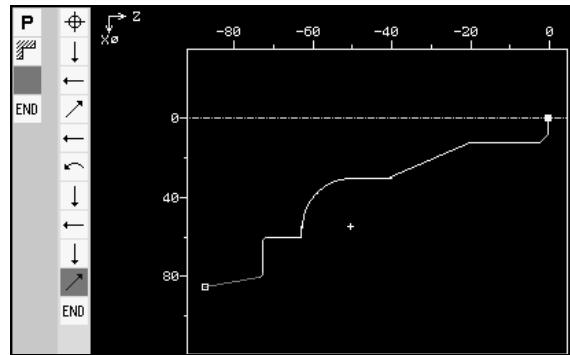
Result

3. Contour input
with contour calculator

- Call CONTOUR mode
- Press soft key New contour
- Enter contour name (here: Cont_11) and confirm
- Define contour start point **X0: 0 Z0: 0**
FRCM: -
- Press soft key Accept
- Enter the following contour elements and confirm each time by
pressing soft key Accept:

1.	X: 12 FS: 2
2.	Z: -20
3.	X: 30 Z: -40 R: 0.5
4.	Z: -50
5.	R: 12.5 Z: -62.5 Select
6.	X: 60 Select
7.	Z: -72.5 R: 1
8.	X: 80 R: 1
9.	X: 85 Z: -87 FS: 0

Result



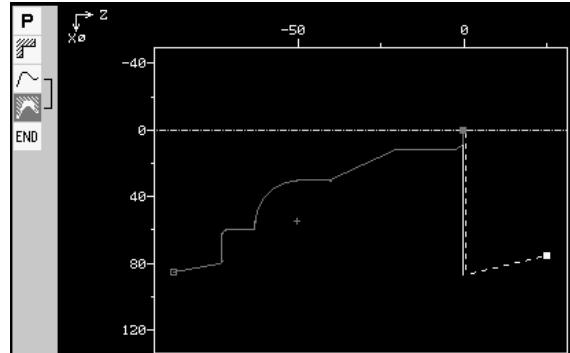
- Press soft key Accept

4. Stock removal (roughing)

- and enter stock removal parameters for roughing:
 T_1 F : 0.2 mm/rev FR : 0.1 mm/rev S : 150 m/min
 Longitudinal
 External
 Infeed depth D : 3 inc
 Finishing allowance U_x : 0.4 inc U_z : 0.1 inc
 Blank description: Cylinder
 Allowance X_D : 0 inc Z_D : 2 inc
 Machine relief cuts: No
 Machine tool angle: Yes
 Insert angle α : 80°
 Main cutting edge angle β : 93°

- Press the soft key

Result



5. Stock removal (finishing)

- and enter stock removal parameters for finishing:
 T_2 F : 0.14 mm/rev S : 220 m/min
 Longitudinal
 External
 Machine relief cuts: No
 Machine tool angle: Yes

Insert angle $\alpha: 35^\circ$
 Main cutting edge angle $\beta: 93^\circ$

- Press soft key  Accept

6. Groove

- Call  STOCK REMOVAL mode and press the  soft key

- Set parameters:

T3 F: 0.12 mm/rev S: 180 m/min

Set position and reference point 

Reference point X0: 30 Z0: -48

Groove width B1: 5 inc

Groove depth T1: 3 inc

Flank angle $\alpha_1: 15^\circ$ $\alpha_2: 15^\circ$

Radius/chamfer R1: 0.4 R2: 0 R3: 0 FS4: 0.4

Infeed depth 1st cut D: 0

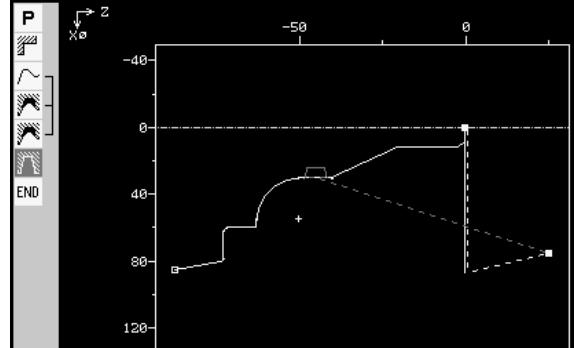
Finishing allowance U: 0

Tool edge width BS: 3

Number of grooves N: 1

- Press soft key  Accept

Result



7. Longitudinal thread M12x1.75

- Call  CYCLE mode and press the  Thread  Longitud. thread soft keys

- Set parameters:

T4 P: 1.75 mm/rev G: 0 S: 600 rev/min

External  Linear

Reference point X0: 12 Z0: 0

Longitudinal thread Z1: -18

Thread run-in W: 6

Thread run-out R: 0

Thread depth K: 1.1

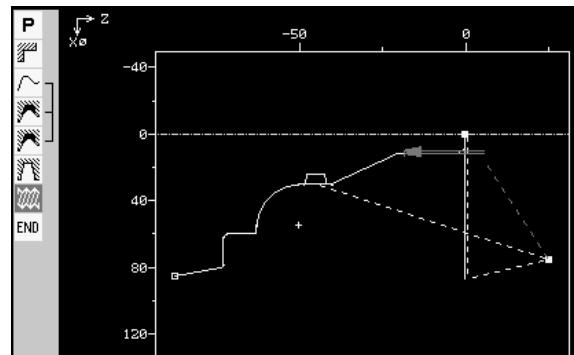
Helix angle $\alpha: 29^\circ$

12.1 Example 1: External machining with groove and thread

Number of cuts **AS: 14**
Insertion depth **E: 0.24**
Finishing allowance **U: 0**
Lift in X **V: 2**
Starting angle offset **Q: 0**

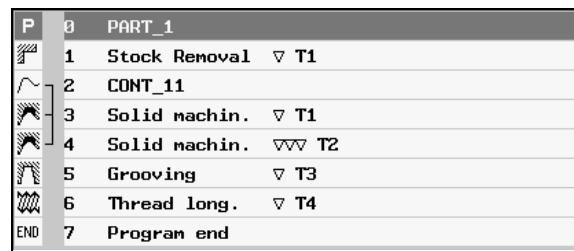
- Press soft key  **Accept**

Result



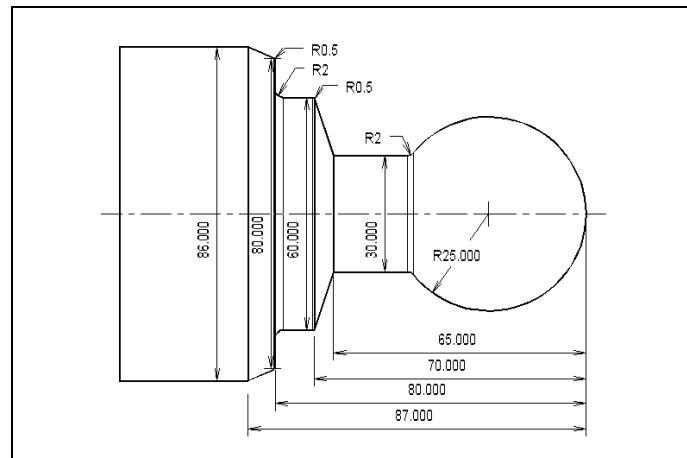
8. Completed EasyStep
program

Switch from the programming graphics to the flow chart by pressing
 the **HELP** key:



12.2 Example 2: External machining with sphere

Workpiece drawing



Blank Dimensions: Ø 85x120mm

Material: Aluminum

Tools T1: 80° Roughing tool R 0.8

T2: 35° Finishing insert R 0.4

T3: Plunge-cutter 3mm

T4: Threading tool 1.75

T5: 35° Roughing insert R 0.8

Program

1. Create new program

Call PROGRAM mode and press the **New program** soft key
Enter program name (here: Part_2) and confirm

2. Stock removal cycle for surface turning

- Call STOCK REMOVAL mode and press the soft key

• Set parameters:

T5 F: 0.3 mm/rev S: 160 m/min

Position: Set cutting position



Parallel to X axis

Reference point X0: 87 Z0: 1

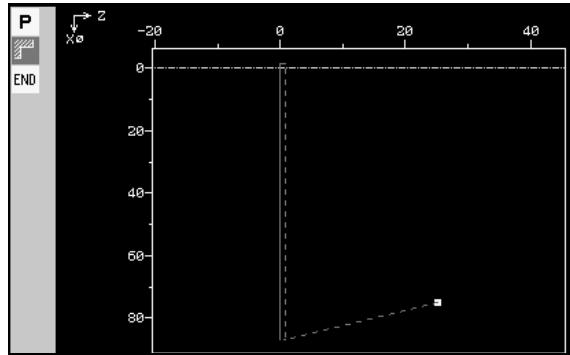
End point X1: -1.6 Z1: 0

Infeed depth D: 2 inc

Finishing allowance Ux: 0 Uz: 0.1

- Press soft key

Result

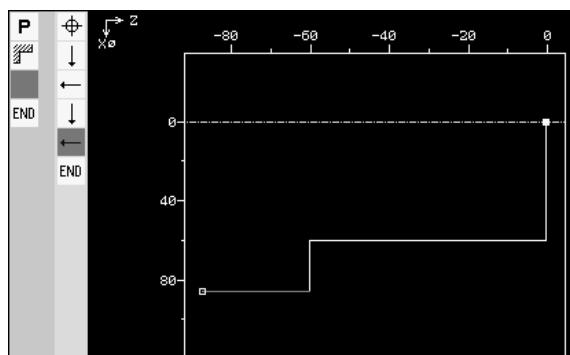


3. Blank contour input
with contour calculator

- Call CONTOUR mode
- Press soft key New contour
- Input blank contour name (here: Cont_121) and confirm
- Blank contour start point **X0: 0 Z0: 0**
FRCM: -
- Press soft key Accept
- Enter the following contour elements and confirm each time by
pressing soft key Accept :

1.	X: 60
2.	Z: -60
3.	X: 86
4.	Z: -87
5.	X: 0
6.	Z: 0

Result



4. Finished-part contour input
with contour calculator

- Press soft key  Accept

- Press soft key  New contour

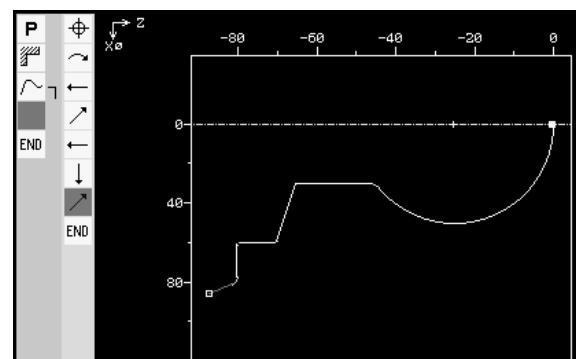
- Enter finished-part contour name (here: Cont_122) and confirm
- Finished-part contour start point **X0: 0 Z0: 0 FRCM: -**

- Press soft key  Accept

- Enter the following contour elements and confirm each time by pressing soft key  Accept :

1.			R: 25	X: 30	K: -25
		 Select dialog	 Accept dialog		R: 2
2.			Z: -65		
3.		X: 60	Z: -70	R: 0.5	
4.		Z: -80	R: 0.5		
5.		X: 80	R: 2		
6.		X: 86	Z: -87		

Result



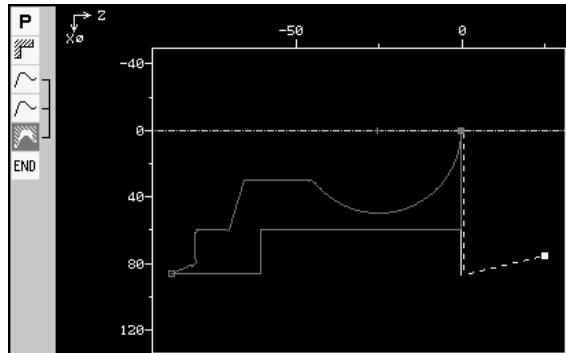
- Press soft key  Accept

12.2 Example 2: External machining with sphere

5. Stock removal (roughing)

- and enter stock removal parameters for roughing:
T1 **F**: 0.3 mm/rev **FR**: 0.15 mm/rev **S**: 160 m/min
 Longitudinal
 External
 Infeed depth **D**: 2 inc
 Machining allowance **Ux**: 0.2 **Uz**: 0.1
 Blank description: Contour
 Machine relief cuts: No
 Machine tool angle: Yes
 Insert angle α : 80°
 Main cutting edge angle β : 93°

Result



6. Remove residual material

- and enter parameters for roughing:

T5 **F**: 0.3 mm/rev **FR**: 0.15 mm/rev **S**: 160 m/min
 Longitudinal
 External
 Infeed depth **D**: 2 inc
 Machining allowance **Ux**: 0 **Uz**: 0
 Machine relief cuts: Yes
 Machine tool angle: Yes
 Insert angle α : 35°
 Main cutting edge angle β : 93°

7. Stock removal (finishing)

- and enter stock removal parameters for finishing:

T2 **F**: 0.12 mm/rev **S**: 200 m/min

Longitudinal
 External
 Machine relief cuts: Yes

Machine tool angle: Yes
Insert angle $\alpha: 35^\circ$
Main cutting edge angle $\beta: 93^\circ$

- Press soft key  Accept

8. Completed EasyStep program

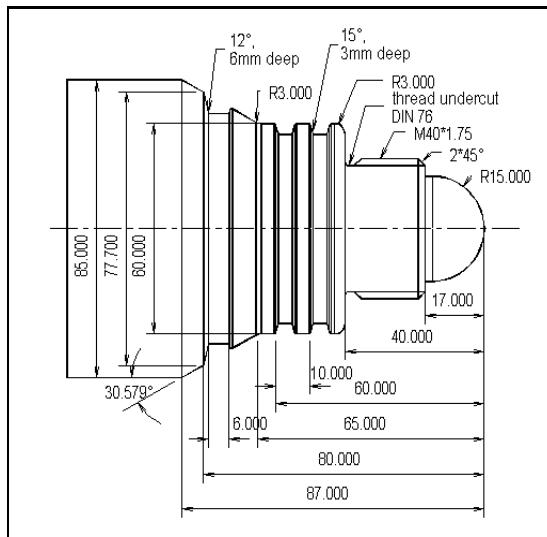
Switch from the programming graphics to the flow chart by pressing

the  HELP key:

P	0 PART_2
1	Stock Removal ▽ T5
2	Blank: CONT_121
3	Finished part: CONT_122
4	Solid machin. ▽ T1
5	Residual mat. ▽ T5
6	Solid machin. ▽▽▽ T2
7	Program end

12.3 Example 3: External machining with thread undercuts and grooves

Workpiece drawing



Blank

Dimensions: Ø 85x120mm

Material: Aluminum

Tools

T1: 80° Roughing tool R 0.8

T2: 35° Finishing insert R 0.4

T3: Plunge-cutter 3mm

T4: Threading tool 1.75

T5: 35° Roughing insert R 0.8

Program

1. Create new program

Call PROGRAM mode and press the soft key
Enter program name (here: Part_3) and confirm

2. Stock removal cycle for surface turning

- Call STOCK REMOVAL mode and press the soft key

• Set parameters:

T1 **F**: 0.2 mm/rev **S**: 150 m/min

Position: Set cutting position

 Parallel to X axis

Reference point **X0**: 87 **Z0**: 1

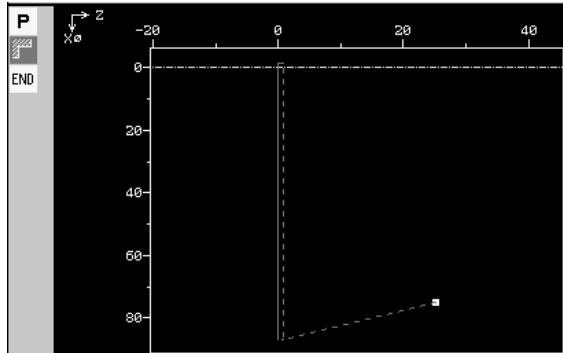
End point **X1**: -1.6 **Z1**: 0

Infeed depth **D**: 2 inc

Finishing allowance **Ux**: 0 **Uz**: 0.1

- Press soft key

Result:

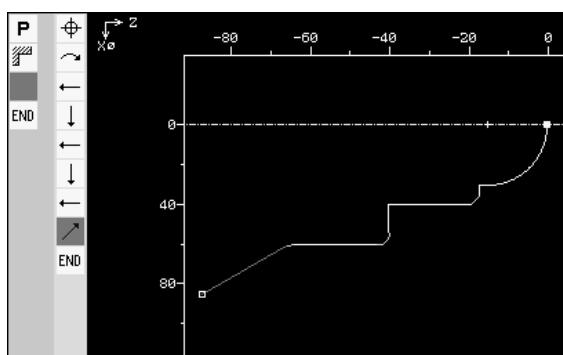


3. Contour input
with contour calculator

- Call CONTOUR mode
- Press soft key New contour
- Enter contour name (here: Cont_13) and confirm
- Contour start point **X0: 0 Z0: 0**
FRCM: -
- Press soft key Accept
- Enter the following contour elements and confirm each time by
pressing soft key Accept:

- | | | |
|----|--|--|
| 1. | R: 15
<input type="button" value="Select dialog"/> | X: 30 Z: -15
<input type="button" value="Accept dialog"/> |
| 2. | Z: -17 | |
| 3. | X: 40 FS: 2 | |
| 4. | Z: -40 | |
| 5. | X: 60 R: 3 | |
| 6. | Z: -65 R: 3 | |
| 7. | X: 85 Z: -87 | |

Result:



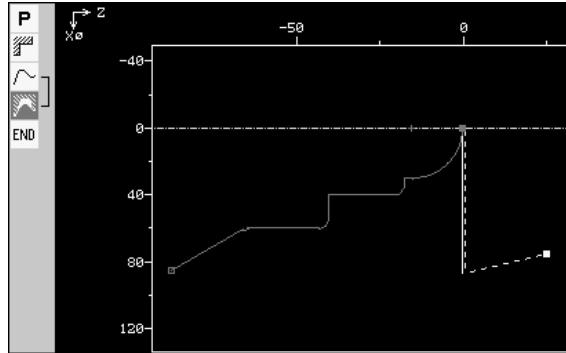
- Press soft key  Accept

4. Stock removal (roughing)

-  and enter stock removal parameters for roughing:
- T1 F: 0.2 mm/rev FR: 0.1 mm/rev S: 160 m/min
- Longitudinal 
External 
- Infeed depth D: 3 inc
Finishing allowance Ux: 0.2 Uz: 0.1
Blank description: Cylinder
Allowance X_D: 0 Z_D: 2
Machine relief cuts: No
Machine tool angle: Yes
Insert angle α: 80°
Main cutting edge angle β: 93°

- Press soft key  Accept

Result:



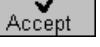
5. Stock removal (finishing)

-  and enter stock removal parameters for finishing:

T2 F: 0.1 mm/rev S: 220 m/min

Longitudinal 
External 

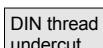
Machine relief cuts: No
Machine tool angle: Yes
Insert angle α: 35°
Main cutting edge angle β: 93°

- Press soft key  Accept

6. Undercut: DIN thread

M40*1.75

(Roughing and finishing)

- Call  CYCLE mode and press the   soft keys
- Set parameters:
T2 F: 0.1 mm/rev S: 180 rev/min

Position: Set cutting position

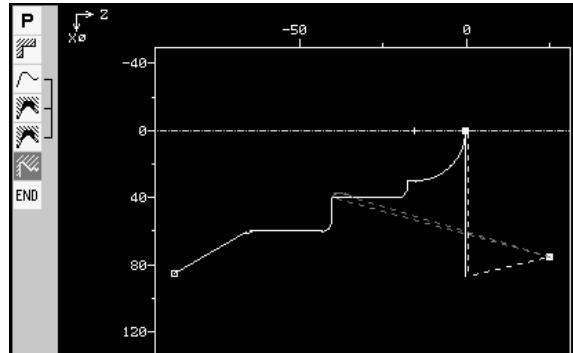


External

Thread lead **P**: 1.75 mm/rev
 Reference point **X0**: 40 **Z0**: -40
 Plunge angle α : 30
 Cross-feed in X **V**: 40
 Infeed **D**: 1 inc
 Finishing allowance **U**: 0.1 inc

- Press soft key **Accept**

Result:



7. Groove with tapers/radii (roughing)

- Call STOCK REMOVAL mode and press the soft key
- Set parameters:

T3 **F**: 0.12 mm/rev **S**: 180 m/min

Set position and reference point

Reference point **X0**: 60 **Z0**: -60

Groove width **B1**: 4 inc

Groove depth **T1**: 3 inc

Flank angle $\alpha 1$: 15 $\alpha 2$: 15

Radius **R1**: 0.5 **R2**: 0 **R3**: 0 **R4**: 0.5

Infeed depth 1st cut **D**: 0

Finishing allowance **U**: 0.1

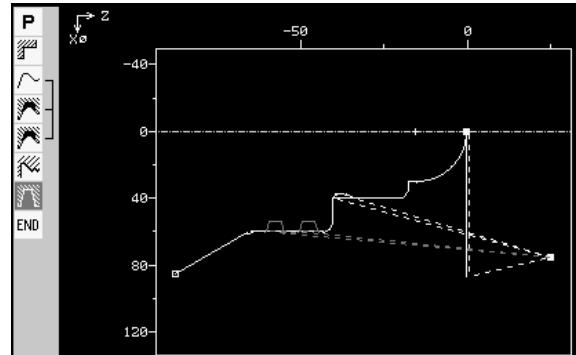
Tool edge width **BS**: 3

Number of grooves **N**: 2

Groove offset **P**: 10 inc

- Press soft key **Accept**

Result:

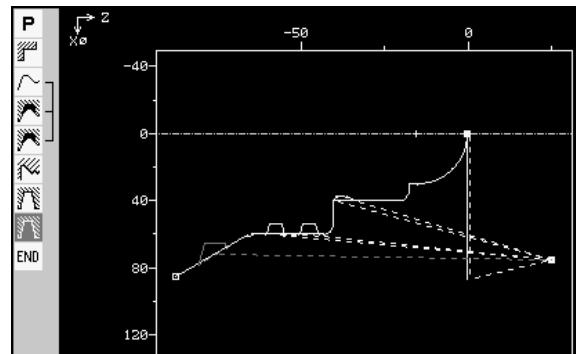


8. Groove with tapers/radii in a taper (roughing)

- Call STOCK REMOVAL mode and press the soft key
- Set parameters:

T3	F: 0.12 mm/rev	S: 180 m/min		
Set position and reference point				
Reference point	X0: 77.7	Z0: -80		
Groove width	B1: 6 inc			
Groove depth	T1: 6 inc			
Taper angle	$\alpha 0$: -30.579			
Flank angle	$\alpha 1$: 16	$\alpha 2$: 16		
Radius	R1: 0.5	R2: 0	R3: 0	R4: 0.5
Infeed depth 1st cut	D: 0			
Finishing allowance	U: 0.0			
Tool edge width	BS: 3			
Number of grooves	N: 1			
- Press soft key

Result:



9. Longitudinal thread M12x1.75 (roughing)

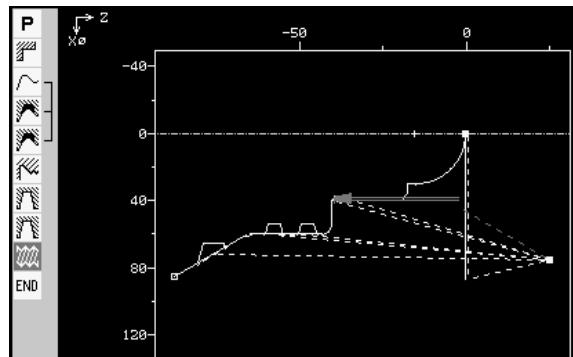
- Call CYCLE mode and press the soft keys
- Set parameters:

T4	P: 1.75 mm/rev	G:0	S: 400 rev/min
External		Linear	

Reference point	X0: 40	Z0: -17
Longitudinal thread	Z1: -39	
Thread run-in (2*F)	W: 15	
Thread run-out	R: 0	
Thread depth	K: 1.1	
Helix angle	α : 29	
Number of cuts	AS: 8	
Insertion depth	E: 0.24	
Finishing allowance	U: 0	
Return distance	V: 2	
Starting angle offset	Q: 0	

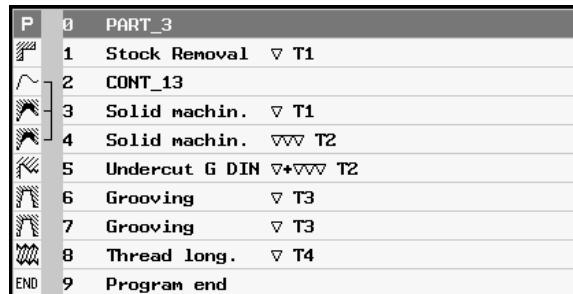
- Press soft key  Accept

Result:



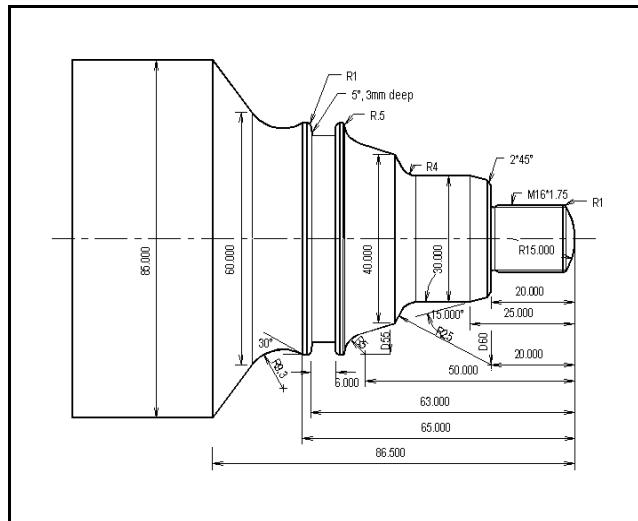
10. Completed EasyStep program

Switch from the programming graphics to the flow chart by pressing the  key:



12.4 Example 4: External machining with thread undercut and groove

Workpiece drawing



Blank

Dimensions: Ø 85x120mm

Material: Aluminum

Tools

T1: 80° Roughing tool R 0.8

T2: 35° Finishing insert R 0.4

T3: Plunge-cutter 3mm

T4: Threading tool 1.75

T5: 35° Roughing insert R 0.8

Program

1. Create new program

Call PROGRAM mode and press the soft key

Enter program name (here: Part_4) and confirm

2. Stock removal cycle for surface turning

- Call STOCK REMOVAL mode and press the soft key

• Set parameters:

T1 F: 0.3 mm/rev **S**: 160 m/min

Position: Set cutting position

Parallel to X axis

Reference point **X0**: 87 **Z0**: 1

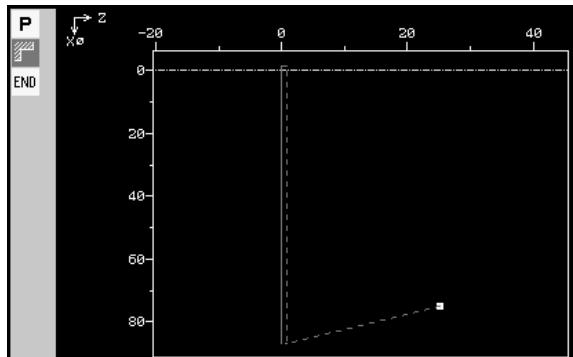
End point **X1**: -1.6 **Z1**: 0

Infeed depth **D**: 2 inc

Finishing allowance **Ux**: 0 **Uz**: 0.1

- Press soft key

Result



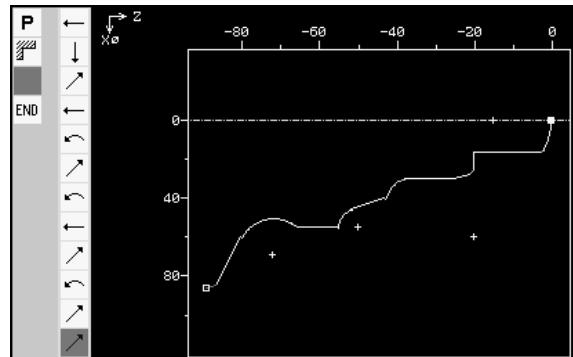
3. Contour input
with contour calculator

- Call CONTOUR mode
- Press soft key New contour
- Enter contour name (here: Cont_14) and confirm
- Contour start point **X0: 0 Z0: 0**
FRCM: -
- Press soft key Accept
- Enter the following contour elements and confirm each time by
pressing soft key Accept :

1.	R: 15	X: 16	K: -15	
	Select dialog	Accept dialog	R: 1	
2.	Z: -20	FS: 0		
3.	FS: 1			
4.	X: 30	Z: -25	$\alpha 1: 165$	FS: 0
5.	$\alpha 1: 180$	R: 4		
6.	R: 25	X: 40	I: 60	
	Select dialog	Accept dialog	Select dialog	Accept dialog
	K: -20	FS: 0		
7.	FS: 0			
8.	R: 5	X: 55	I: 55	K: -50
	Select dialog	Accept dialog		
	Tangent to prec.			
9.	Z: -65	FS: 0		

10. $\alpha_1: -150$ **FS:** 0
11. **R:** 9.3 **X:** 60 **Z:** -80 **FS:** 0
 Tangent to prec. Select dialog Accept dialog
12. **Z:** -86.5 **FS:** 0
 Tangent to prec.
13. **X:** 86 **Z:** -89 **FS:** 0

Result



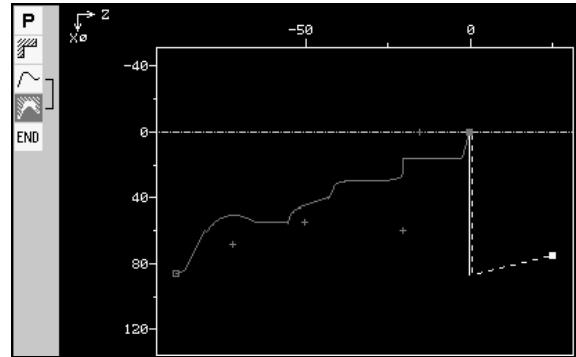
- Press soft key **Accept**

4. Stock removal (roughing)

- Stock removal** and enter stock removal parameters for roughing:
T5 **F:** 0.3 mm/rev **FR:** 0.15 mm/rev **S:** 180 m/min
 Longitudinal
 External
 Infeed depth **D:** 2 inc
 Machining allowance **Ux:** 0.15 **Uz:** 0.1
 Blank description: Cylinder
 Allowance **X_D:** 0 **Z_D:** 2
 Machine relief cuts: Yes
 Machine tool angle: Yes
 Insert angle **α:** 35°
 Main cutting edge angle **β:** 93°

- Press soft key **Accept**

Result



5. Stock removal (finishing)

- Stock removal and enter stock removal parameters for finishing:

T2 **F:** 0.1 mm/rev **S:** 200 m/min

Longitudinal 

External 

Machine relief cuts: Yes

Machine tool angle: Yes

Insert angle $\alpha: 35^\circ$

Main cutting edge angle $\beta: 93^\circ$

- Press soft key 

6. Undercut: DIN thread

M40*1.75

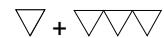
(Roughing and finishing)

- Call  CYCLE mode and press the Undercut DIN thread undercut soft keys

- Set parameters:

T5 **F:** 0.1 mm/rev **S:** 500 rev/min

Position: Set cutting position

 +  External

Thread lead **P:** 1.75 mm/rev

Reference point **X0:** 16 **Z0:** -20

Plunge angle $\alpha: 30$

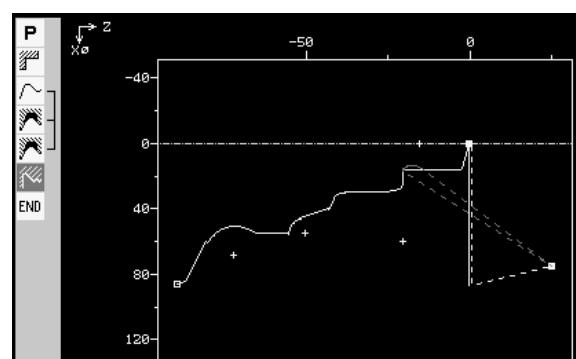
Cross-feed in X **V:** 17

Infeed **D:** 1 inc

Finishing allowance **U:** 0 inc

- Press soft key 

Result

7. Groove with tapers/radii
(roughing)

- Call  STOCK REMOVAL mode and press the  soft key

- Set parameters:

T3 **F:** 0.12 mm/rev **S:** 180 m/min

Set position and reference point 

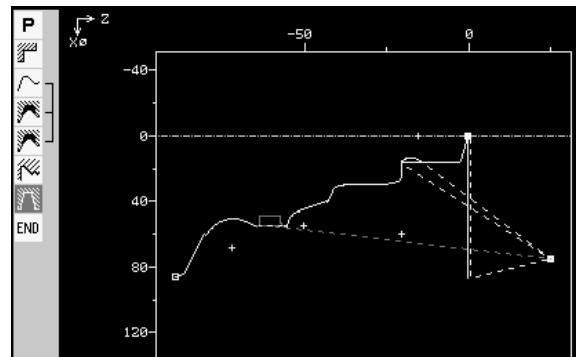
Reference point **X0:** 55

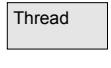
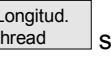
Z0: -63.5

Groove width	B1: 6 inc
Groove depth	T1: 3 inc
Flank angle	α1: 5 α2: 5
Radius	R1: 1 R2: 0 R3: 0 R4: 1
Infeed depth 1st cut	D: 0
Finishing allowance	U: 0.0
Tool edge width	BS: 3
Number of grooves	N: 1

- Press soft key  Accept

Result

8. Longitudinal thread
M12x1.75 (roughing)

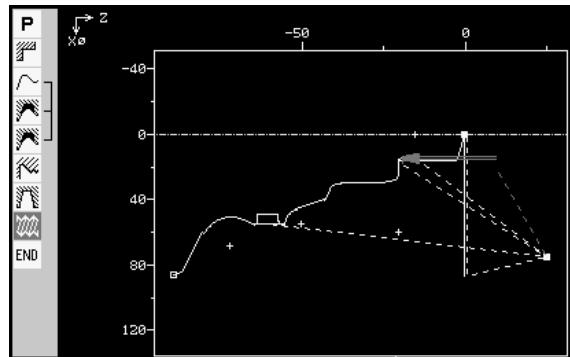
- Call  CYCLE mode and press the  Thread  Longitud. thread soft keys
- Set parameters:

T4 **P:** 1.75 mm/rev **G:0** **S:** 500 rev/min
 External  Linear

Reference point	X0: 16	Z0: 0
Longitudinal thread	Z1: -19	
Thread run-in (2*F)	W: 10	
Thread run-out	R: 0	
Thread depth	K: 1.1	
Helix angle	α: 29	
Number of cuts	AS: 14	
Insertion depth	E: 0.24	
Finishing allowance	U: 0	
Return distance	V: 2	
Starting angle offset	Q: 0	

- Press soft key  Accept

Result



9. Completed EasyStep program

Switch from the programming graphics to the flow chart by pressing the key:

Result

P	0 PART_4
1	Stock Removal ▽ T1
2	CONT_14
3	Solid machin. ▽ TS
4	Solid machin. ▽▽▽ T2
5	Undercut G DIN ▽+▽▽▽ TS
6	Grooving ▽ T3
7	Thread long. ▽ T4
END	8 Program end

Notes

Appendix

A	Abbreviations.....	A-208
B	Terms	A-211
C	References.....	A-214
D	Index	A-227

A Abbreviations

ABS	Absolute measure
CNC	Computerized Numerical Control
COM	Communication Component of the NC control performing and coordinating communication
D	Cutting edge
DIN	Deutsche Industrie Norm (Germany Industry Standard)
DRF	Differential Resolver Function The function generates an incremental zero offset in automatic mode with an electronic handwheel.
DRY	Dry Run
F	Feed
GRC	Grinding Wheel Radius Compensation
GUD	Global User Data
HW	Hardware
INC	Increment: Incremental dimension
INI	Initializing Data
LED	Light Emitting Diode
M01	M function: Programmed stop
M17	M function: End of subroutine
MCS	Machine Coordinate System (Machine)
MD	Machine Data
MDA	Manual Data Automatic
MDI	Manual Data Input
MLFB	Machine-Readable Product Code
MPF	Main Program File

NC	Numerical Control The NC control comprises the components NCK, PLC, PCU and COM.
NCK	Numerical Control Kernel Component of the NC control executing programs and coordinating the movements of the machine tool.
OP	Operator Panel
PC	Personal Computer
PCU	Personal Computer Unit Component of the NC control that permits communication between the operator and the machine.
PLC	Programmable Logic Control Component of the NC control processing the control logic of the machine tool
PRT	Program Test
REF	Reference point approach
REPOS	Repositioning
ROV	Rapid Override
RS-232-C (V.24)	Serial interface
S	Spindle speed
SBL	Single Block
SI	Safety Integrated
SK	Soft key
SKP	Skip block
SPF	Subprogram File
SW	Software
T	Tool
TMZ	Tool Magazine Zero
V	Cutting rate

WCS	Workpiece Coordinate System (Work)
WPD	WorkPiece Directory
ZO	Zero Offset (work offset)

B Terms

Important terms are listed below in alphabetical order, accompanied by explanations. Cross-references to other entries in this glossary are indicated by the symbol ->.

A

Absolute measure

A destination for an axis movement is defined by a measure referring to the origin of the currently active coordinate system.

See also -> incremental dimension.

Alarm

All -> messages and alarms are displayed on the control panel in plain-text form and accompanied by date and time and the symbol for the appropriate deletion criterion.

Archiving

Exporting files and/or directories to an **external** storage device.

Auxiliary functions

Auxiliary functions can be used to pass -> parameters to the -> PLC in the -> parts program, triggering reactions there which are defined by the machine manufacturer.

Axes

The CNC axes are categorized by their functional scope as follows:

- Axes: interpolative path axes
- Auxiliary axes: non-interpolative infeed and positioning axes with axis-specific feedrates. Auxiliary axes do not participate in workpiece machining as such and include tool feeders, tool magazines, etc.

Axis identifier

In compliance with DIN 66217, axes are identified as X, Y, Z for a right-handed rectangular Cartesian -> coordinate system rotating in the clockwise direction. Rotary axes rotating around X, Y, Z are assigned the identifiers A, B, C. Additional axes, which are parallel to those specified, can be identified with other letters.

B

Blank

The unmachined workpiece.

C

C axis

Axis about which the tool spindle describes a controlled rotational and positioning movement.

CNC

-> NC

Contour

Outline of a -> workpiece

Coordinate system

See -> Machine coordinate system -> Workpiece coordinate system (work).

Cycle

Protected subroutine for execution of a recurring machining process on the -> workpiece.

E

Editor

The editor allows programs/text/program blocks to be created, modified, appended, moved in blocks and inserted.

Electronic handwheel

Electronic handwheels are used to traverse the selected axes simultaneously under manual control. The handwheel clicks are analyzed by the increment analyzer.

F**Feedrate override**

The current feedrate setting entered via the control panel or by the PLC is overlaid on the programmed feedrate (0–200 %). The feedrate can also be corrected by a programmable percentage factor (1–200 %) in the machining program.

I**Inch system of measurement**

System of measurement in which distances are measured in inches and fractions of inches.

Incremental dimension

A destination for axis traverse is defined by a distance to be covered and a direction referenced to a point already reached. See also -> absolute dimension.

Length of the traversing path given by the number of increments. The number of increments can be stored as -> setting data or selected using the corresponding 1, 10, 100.

M**Machine axes**

Axes which exist physically on the machine tool.

Machine control panel

An operator panel on a -> machine tool with operating elements such as keys, rotary switches, etc. and simple indicators such as LEDs. It is used for direct control of the machine tool via the PLC.

Machine coordinate system

System of coordinates (Machine) based on the axes of the machine tool.

Machine zero

A fixed point on the machine tool which can be referenced by all (derived) measuring systems.

Messages

All messages programmed in the parts program and -> alarms detected by the system are displayed on the control panel in plain-text form with date and time and the appropriate symbol for the deletion criterion. Alarms and messages are displayed separately.

Metric system

Standardized system of units: the units of measure for length for example, are mm (millimeter) and m (meter).

N**NC**

Numerical Control

O**Operating area**

The basic functions of the control are subdivided into individual operating modes.

Oriented spindle stop

Stops the workpiece spindle with a specified orientation angle, e.g. to perform an additional machining operation at a specific position.

Override

Manual or programmable control feature which enables the operator to overlay programmed feedrates or speeds to adapt them to a specific workpiece or material.

P**PLC**

Programmable logic controller: A component of the -> NC control: A control which can be programmed to control the logic on a machine tool.

	Preset	The preset function is a means of redefining the control zero in the machine coordinate system. Preset does not trigger movement of the axes: instead, a new position value is entered for the current axis position.
	R	
	Rapid traverse	Highest speed of an axis, used, for example, to bring the tool from an idle position to the -> workpiece contour or move it back from the workpiece contour.
	Reference point	A point on the machine tool to which the measurement system of the -> machine refers.
	Reference point approach	If the system of position measuring employed is not based on absolute-value encoders, the control must perform a reference point approach in order to ensure that the measured values supplied by the measuring system are in harmony with the machine coordinate values.
	Rotary axis	Rotary axes cause the tool or workpiece to rotate to a specified angle position.
	S	
	Setting data	Data which provide the NC with information on the properties of the machine tool in a way defined by the software.
	Soft key	A key whose name appears on an area of the screen. The choice of soft keys displayed is adapted dynamically to the operating situation. The freely assignable function keys (soft keys) are assigned to functions defined in the software.
	T	
	Teach In	Teach In is a means of creating or correcting parts programs. The individual program blocks are keyed in via the keyboard and executed immediately. Positions entered with the direction keys or handwheels can also be stored. Additional specifications such as G functions, feedrates and M functions can be entered in the same block.
	Tool	A part which in a machine tool is employed to shape the workpiece, for example a turning tool, milling cutter, drill, laser beam, etc.
	Tool nose radius compensation	When a contour is programmed, it is assumed that the tool is pointed. Since this is not always the case in practice, the curvature radius of the tool used is specified so that the control can make allowance for it. The curvature center point is guided along an equal distance to the contour at an offset corresponding to the curvature radius.
	W	
	Workpiece	<ol style="list-style-type: none">1. Part to be produced/machined by the machine tool.2. A workpiece is a directory in which programs and other data are stored. Workpieces are also stored in a directory.

C References**General Documentation**

/BU/ SINUMERIK 840D/840Di/810D/802S, C, D
Ordering Information
Catalog NC 60
Order No: E86060-K4460-A101-A9-7600

/ST7/ **SIMATIC**
SIMATIC S7 Programmable Logic Controllers
Catalog ST 70
Order No: E86060-K4670-A111-A3

/Z/ SINUMERIK, SIROTEC, SIMODRIVE
Accessories and Equipment for Special-Purpose Machines
Catalog NC Z
Order No: E86060-K4490-A001-A8-7600

Electronic Documentation

/CD1/ The SINUMERIK System
DOC ON CD (11.02 Edition)
(includes all SINUMERIK 840D/840Di/810D/802 and
SIMODRIVE publications)
Order No.: 6FC5 298-6CA00-0BG3

User Documentation

/AUK/	SINUMERIK 840D/810D AutoTurn Short Operating Guide Order No.: 6FC5 298-4AA30-0BP3	(10.02 Edition)
/AUP/	SINUMERIK 840D/810D AutoTurn Graphic Programming System Operator's Guide Programming/Setup Order No: 6FC5 298-4AA40-0BP3	(02.02 Edition)
/BA/	SINUMERIK 840D/810D Operator's Guide MMC Order No: 6FC5 298-6AA00-0BP0	(10.00 Edition)
/BAD/	SINUMERIK 840D/840Di/810D Operator's Guide HMI Advanced Order No.: 6FC5 298-6AF00-0BP2	(11.02 Edition)
/BEM/	SINUMERIK 840D/810D Operator's Guide HMI Embedded Order No.: 6FC5 298-6AC00-0BP2	(11.02 Edition)
/BAH/	SINUMERIK 840D/840Di/810D Operator's Guide HT 6 Order No.: 6FC5 298-0AD60-0BP2	(06.02 Edition)
/BAK/	SINUMERIK 840D/840Di/810D Short Operating Guide Order No: 6FC5 298-6AA10-0BP0	(02.01 Edition)
/BAM/	SINUMERIK 810D/840D Operator's Guide ManualTurn Order No: 6FC5 298-6AD00-0BP0	(08.02 Edition)
/BAS/	SINUMERIK 840D/810D Operator's Guide ShopMill Order No: 6FC5 298-6AD10-0BP1	(11.02 Edition)
/BAT/	SINUMERIK 840D/810D Operator's Guide ShopTurn ShopTurn Order No.: 6FC5 298-6AD50-0BP2	(03.03 Edition)
/BNM/	SINUMERIK 840D/840Di/810D User's Guide Measuring Cycles Order No: 6FC5 298-6AA70-0BP2	(11.02 Edition)

/CAD/	SINUMERIK 840D/840Di/810D Operator's Guide CAD-Reader Order No: (included in online Help)	(03.02 Edition)
/DA/	SINUMERIK 840D/840Di/810D Diagnostics Guide Order No: 6FC5 298-6AA20-0BP3	(11.02 Edition)
/KAM/	SINUMERIK 840D/810D Short Guide ManualTurn Order No: 6FC5 298-5AD40-0BP0	(04.01 Edition)
/KAS/	SINUMERIK 840D/810D Short Guide ShopMill Order No: 6FC5 298-5AD30-0BP0	(04.01 Edition)
/KAT/	SINUMERIK 840D/810D Short Guide ShopTurn Order No: 6FC5 298-6AF20-0BP0	(07.01 Edition)
/PG/	SINUMERIK 840D/840Di/810D Programming Guide Fundamentals Order No: 6FC5 298-6AB00-0BP2	(11.02 Edition)
/PGA/	SINUMERIK 840D/840Di/810D Programming Guide Advanced Order No.: 6FC5 298-6AB10-0BP2	(11.02 Edition)
/PGK/	SINUMERIK 840D/840Di/810D Short Guide Programming Order No.: 6FC5 298-6AB30-0BP1	(02.01 Edition)
/PGM/	SINUMERIK 840D/840Di/810D Programming Guide ISO Milling Order No: 6FC5 298-6AC20-0BP2	(11.02 Edition)
/PGT/	SINUMERIK 840D/840Di/810D Programming Guide ISO Turning Order No: 6FC5 298-6AC10-0BP2	(11.02 Edition)
/PGZ/	SINUMERIK 840D/840Di/810D Programming Guide Cycles Order No.: 6FC5 298-6AB40-0BP2	(11.02 Edition)

/PI/**PCIN 4.4**

Software for Data Transfer to/from MMC Module
Order No.: 6FX2 060-4AA00-4XB0 (Eng., Fr., Ger.)
Order from: WK Fürth

/SYI/

SINUMERIK 840Di

System Overview

(02.01 Edition)

Order No.: 6FC5 298-6AE40-0BP0

Manufacturer/Service Documentation

a) Lists

/LIS/ SINUMERIK 840D/840Di/810D
SIMODRIVE 611D
Lists (11.02 Edition)
Order No.: 6FC5 297-6AB70-0BP3

b) Hardware

/BH/ SINUMERIK 840D/840Di/810D
Operator Components Manual (Hardware) (11.02 Edition)
Order No: 6FC5 297-6AA50-0BP2

/BHA/ SIMODRIVE Sensor
Absolute Position Sensor with Profibus-DP
User's Guide (Hardware) (02.99 Edition)
Order No.: 6SN1 197-0AB10-0YP1

/EMV/ SINUMERIK, SIROTEC, SIMODRIVE
EMC Installation Guide (06.99 Edition)
Planning Guide (Hardware)
Order No.: 6FC5 297-0AD30-0BP1

/PHC/ SINUMERIK 810D
Configuring Manual (Hardware) (03.02 Edition)
Order No.: 6FC5 297-6AD10-0BP0

/PHD/ SINUMERIK 840D
Configuring Manual NCU 561.2-573.2 (Hardware) (10.02 Edition)
Order No.: 6FC5 297-6AC10-0BP2

/PMH/ SIMODRIVE Sensor
Measuring System for Main Spindle Drives
Configuring/Installation Guide, SIMAG-H (Hardware) (05.99 Edition)
Order No.: 6SN1197-0AB30-0BP0

c) Software

/FB1/ SINUMERIK 840D/840Di/810D/FM-NC
Description of Functions, Basic Machine (Part 1) (11.02 Edition)
(the various sections are listed below)
Order No.: 6FC5 297-6AC20-0BP2

- A2 Various Interface Signals
- A3 Axis Monitoring, Protection Zones
- B1 Continuous Path Mode, Exact Stop and Look Ahead
- B2 Acceleration

- D1 Diagnostic Tools
- D2 Interactive Programming
- F1 Travel to Fixed Stop
- G2 Velocities, Setpoint/Actual-Value Systems, Closed-Loop Control
- H2 Output of Auxiliary Functions to PLC
- K1 Mode Group, Channels, Program Operation Mode
- K2 Axes, Coordinate Systems, Frames
 - Actual-Value System for Workpiece, External Zero Offset
- K4 Communication
- N2 EMERGENCY STOP
- P1 Transverse Axes
- P3 Basic PLC Program
- R1 Reference Point Approach
- S1 Spindles
- V1 Feeds
- W1 Tool Compensation

/FB2/ SINUMERIK 840D/840Di/810D(CCU2)

Description of Functions, Extended Functions (11.02 Edition)
(Part 2)

including FM-NC: Turning, Stepper Motor

(the various sections are listed below)

Order No.: 6FC5 297-6AC30-0BP2

- A4 Digital and Analog NCK I/Os
- B3 Several Operator Panels and NCUs
- B4 Operation via PC/PG
- F3 Remote Diagnostics
- H1 JOG with/without Handwheel
- K3 Compensations
- K5 Mode Groups, Channels, Axis Replacement
- L1 FM-NC Local Bus
- M1 Kinematic Transformation
- M5 Measurements
- N3 Software Cams, Position Switching Signals
- N4 Punching and Nibbling
- P2 Positioning Axes
- P5 Oscillation
- R2 Rotary Axes
- S3 Synchronous Spindles
- S5 Synchronized Actions (up to and including SW 3, then /FBSY/)
- S6 Stepper Motor Control
- S7 Memory Configuration
- T1 Indexing Axes
- W3 Tool Change
- W4 Grinding

/FB3/

SINUMERIK 840D/840Di/810D(CCU2)

Description of Functions, Special Functions (Part 3)(the various sections are listed below) (11.02 Edition)
Order No.: 6FC5 297-6AC80-0BP2

- F2 3-Axis to 5-Axis Transformation
- G1 Gantry Axes
- G3 Cycle Times
- K6 Contour Tunnel Monitoring
- M3 Coupled Motion and Leading Value Coupling
- S8 Constant Workpiece Speed for Centerless Grinding
- T3 Tangential Control
- TE1 Clearance Control
- TE2 Analog Axis
- TE3 Master-Slave for drives
- TE4 Transformation Package Handling
- TE5 Setpoint Exchange
- TE6 MCS Coupling
- TE7 Retrace Support
- TE8 Path-Synchronous Switch Signal
- V2 Preprocessing
- W5 3D Tool Radius Compensation

/FBA/

SIMODRIVE 611D/SINUMERIK 840D/810D

Description of Functions, Drive Functions

(11.02 Edition)

(the various sections are listed below)
Order No.: 6SN1 197-0AA80-0BP9

- DB1 Operational Messages/Alarm Reactions
- DD1 Diagnostic Functions
- DD2 Speed Control Loop
- DE1 Extended Drive Functions
- DF1 Enable Commands
- DG1 Encoder Parameterization
- DL1 Linear Motor MD
- DM1 Calculation of Motor/Power Section Parameters and Controller Data
- DS1 Current Control Loop
- DÜ1 Monitors/Limitations

/FBAN/

SINUMERIK 840D/SIMODRIVE 611 DIGITAL

Description of Functions **ANA MODULE**

(02.00 Edition)

Order No.: 6SN1 197-0AB80-0BP0

/FBD/

SINUMERIK 840D

Description of Functions **Digitizing**

(07.99 Edition)

Order No.: 6FC5 297-4AC50-0BP0

- DI1 Start-up
- DI2 Scanning with Tactile Sensors (scancad scan)

	DI3	Scanning with Lasers (scancad laser)
	DI4	Milling Program Generation (scancad mill)
/FBDN/	SINUMERIK 840D/810D	
	IT Solutions	
	System for NC Data Management and Data Distribution	
	(DNC NT-2000)	
	Description of Functions	(01.02 Edition)
	Order No.: 6FC5 297-5AE50-0BP2	
/FBDT/	SINUMERIK 840D/810D	
	IT Solutions	
	NC Data Transfer (SinDNC)	
	Description of Functions	(09.01 Edition)
	Order No.: 6FC5 297-1AE70-0BP1	
/FBFA/	SINUMERIK 840D/840Di/810D	
	ISO Dialects for SINUMERIK	(11.02 Edition)
	Description of Functions	
	Order No.: 6FC5 297-6AE10-0BP2	
/FBFE/	SINUMERIK 840D/810D	
	Description of Functions Remote Diagnosis	(11.02 Edition)
	Order No.: 6FC5 297-0AF00-0BP2	
/FBH/	SINUMERIK 840D/840Di/810D	
	HMI Configuring Package	(11.02 Edition)
	Order No.: (supplied with the software)	
	Part 1 User's Guide	
	Part 2 Description of Functions	
/FBHLA/	SINUMERIK 840D/SIMODRIVE 611 digital	
	Description of Functions HLA Module	(04.00 Edition)
	Order No.: 6SN1 197-0AB60-0BP2	
/FBMA/	SINUMERIK 840D/810D	
	Description of Functions ManualTurn	(08.02 Edition)
	Order No.: 6FC5 297-5AD50-0BP2	
/FBO/	SINUMERIK 840D/810D	
	Configuring OP 030 Operator Interface	(09.01 Edition)
	Description of Functions	
	(the individual sections are listed below)	
	Order No.: 6FC5 297-6AC40-0BP0	
	BA Operator's Guide	
	EU Development Environment (Configuring Package)	

	PSE	Introduction to Configuring of Operator Interface	
	IK	Screen Kit: Software Update and Configuration	
/FBP/	SINUMERIK 840D		
	Description of Functions C-PLC Programming	(03.96 Edition)	
	Order No: 6FC5 297-3AB60-0BP0		
/FBR/	SINUMERIK 840D/810D		
	IT Solutions		
	SINCOM Computer Link (SinCOM)	(09.01 Edition)	
	Description of Functions		
	Order No.: 6FC5 297-6AD60-0BP0		
	NFL Host Computer Interface		
	NPL PLC/NCK Interface		
/FBSI/	SINUMERIK 840D/SIMODRIVE		
	Description of Functions		
	SINUMERIK Safety Integrated	(09.02 Edition)	
	Order No: 6FC5 297-6AB80-0BP1		
/FBSP/	SINUMERIK 840D/810D		
	Description of Functions ShopMill	(11.02 Edition)	
	Order No: 6FC5 297-6AD80-0BP1		
/FBST/	SIMATIC		
	FM STEPDRIVE/SIMOSTEP	(01.01 Edition)	
	Description of Functions		
	Order No.: 6SN1 197-0AA70-0YP4		
/FBSY/	SINUMERIK 840D/810D		
	Description of Functions Synchronized Actions	(10.02 Edition)	
	Order No.: 6FC5 297-6AD40-0BP2		
/FBT/	SINUMERIK 840D/810D		
	Description of Functions ShopTurn	(03.03 Edition)	
	Order No.: 6FC5 297-6AD70-0BP2		
/FBTC/	SINUMERIK 840D/810D		
	IT Solutions		
	SINUMERIK Tool Data Communication SinTDC	(01.02 Edition)	
	Description of Functions		
	Order No.: 6FC5 297-5AF30-0BP0		
/FBTD/	SINUMERIK 840D/810D		
	IT Solutions	(03.01 Edition)	
	Tool Information System (SinTDI) with Online Help		
	Description of Functions		
	Order No.: 6FC5 297-6AE00-0BP0		

/FBU/	SIMODRIVE 611 universal/universal E Closed-Loop Control Component for Speed Control and Positioning Description of Functions Order No.: 6SN1 197-0AB20-0BP5	(02.02 Edition)
/FBW/	SINUMERIK 840D/810D Description of Functions Tool Management Order No.: 6FC5 297-6AC60-0BP1	(10.02 Edition)
/FBWI/	SINUMERIK 840D/810D Description of Functions WinTPM Order No.: The document is an integral part of the software	(02.02 Edition)
/HBA/	SINUMERIK 840D/840Di/810D Manual @Event Order No.: 6AU1900-0CL20-0AA0	(03.02 Edition)
/HBI/	SINUMERIK 840Di Manual Order No.: 6FC5 297-6AE50-0BP1	(09.02 Edition)
/INC/	SINUMERIK 840D/840Di/810D Commissioning Tool SINUMERIK SinuCOM NC System Description Order No.: (an integral part of the Online Help for the start-up tool)	(02.02 Edition)
/PFK/	SIMODRIVE Planning Guide 1FT5/1FT6/1FK6 Motors AC Servo Motors for Feed and Main Spindle Drives Order No: 6SN1 197-0AC20-0BP0	(12.01 Edition)
/PJE/	SINUMERIK 840D/810D Configuring Package HMI Embedded Description of Functions: Software Update, Configuration Installation Order No.: 6FC5 297-6EA10-0BP0 (the document PS Configuring Syntax is supplied with the software and available as a pdf file)	(08.01 Edition)

/PJFE/**SIMODRIVE**

Planning Guide

(09.01 Edition)

Built-In Synchronous Motors 1FE1

AC Motors for Main Spindle Drives

Order No.: 6SN1 197-0AC00-0BP1

/PJLM/**SIMODRIVE**Planning Guide **Linear Motors 1FN1, 1FN3**

(11.01 Edition)

ALL General Information about Linear Motors

1FN1 1FN1 AC Linear Motor

1FN3 1FN3 AC Linear Motor

CON Connections

Order No.: 6SN1 197-0AB70-0BP2

/PJM/**SIMODRIVE**Planning Guide **Motors**

AC Motors for Feed and Main Spindle Drives

(11.00 Edition)

Order No.: 6SN1 197-0AA20-0BP5

/PJU/**SIMODRIVE 611**Planning Guide **Inverters**

(08.02 Edition)

Order No.: 6SN1 197-0AA00-0BP6

PMS**SIMODRIVE**Planning Guide **ECO Motor Spindle**

(04.02 Edition)

for Main Spindle Drives

Order No.: 6SN1 197-0AD04-0BP0

/POS1/**SIMODRIVE POSMO A**

User's Guide

(08.02 Edition)

Distributed Positioning Motor on PROFIBUS DP

Order No.: 6SN2 197-0AA00-0BP3

/POS2/**SIMODRIVE POSMO A**

Installation Instructions (enclosed with POSMO A)

/POS3/**SIMODRIVE POSMO SI/CD/CA**

Distributed Servo Drive Systems, User's Guide

(08.02 Edition)

Order No.: 6SN2 197-0AA20-0BP3

/PPH/	SIMODRIVE Planning Guide 1PH2/1PH4/1PH7 Motors AC Asynchronous Induction for Main Spindle Drives Order No.: 6SN1 197-0AC60-0BP0	(12.01 Edition)
/PPM/	SIMODRIVE Planning Guide Hollow-Shaft Motors for Main Spindle Drives 1PM4 and 1PM6 Order No.: 6SN1 197-0AD03-0BP0	(10.01 Edition)
/S7H/	SIMATIC S7-300 - Manual: Assembly, CPU Data (Hardware) - Reference Manual: Module Data Order No.: 6ES7 398-8AA03-8AA0	(10.98 Edition)
/S7HT/	SIMATIC S7-300 Manual STEP7, Fundamentals, V.3.1 Order No.: 6ES7 810-4CA02-8AA0	(03.97 Edition)
/S7HR/	SIMATIC S7-300 Manual STEP7, Reference Manuals, V.3.1 Order No.: 6ES7 810-4CA02-8AR0	(03.97 Edition)
/S7S/	SIMATIC S7-300 FM 353 Positioning Module for Stepper Drive Order together with configuring package	(04.97 Edition)
/S7L/	SIMATIC S7-300 FM 354 Positioning Module for Servo Drive Order together with configuring package	(04.97 Edition)
/S7M/	SIMATIC S7-300 FM 357.2 Multimodule for Servo and Stepper Drives Order together with configuring package	(01.01 Edition)
/SP/	SIMODRIVE 611-A/611-D SimoPro 3.1 Program for Configuring Machine-Tool Drives Order No.: 6SC6 111-6PC00-0AA□ Order from: WK Fürth	

*Installation
and Start-up*

/IAA/

SIMODRIVE 611A**Installation and Start-Up Guide**

(10.00 Edition)

Order No.: 6SN 1197-0AA60-0BP6

/IAC/

SINUMERIK 810D

Installation and Start-Up Guide

(03.02 Edition)

(incl. description of SIMODRIVE 611D start-up software)

Order No.: 6FC5 297-6AD20-0BP0

/IAD/

SINUMERIK 840D/SIMODRIVE 611D

Installation and Start-Up Guide

(11.02 Edition)

(incl. description of SIMODRIVE 611D start-up software)

Order No.: 6FC5 297-6AB10-0BP2

/IAM/

SINUMERIK 840D/840Di/810D

HMI/MMC Installation and Start-Up Guide

(11.02 Edition)

Order No.: 6FC5 297-6AE20-0BP2

AE1 Updates/Supplements

BE1 Expanding the Operator Interface

HE1 Online Help

IM2 Starting up HMI Embedded

IM4 Starting up HMI Advanced

TX1 Creating Foreign Language Texts

D Index

A

Aborting a machining process 10-164

Absolute measure 2-37

Additive zero offset 3-47

Alarms 11-167

 Description 11-171

 Overview 11-170

Alphanumeric pad 2-20, 2-21

Angle offset 5-88

Angle reference system 2-38

Approach reference points 3-42

Area switchover key 2-20, 2-21

Auxiliary commands 7-131

Auxiliary functions 7-139

B

Basic functions 1-16

Blank 6-123

Block search 7-136

Broken-line graphics 6-115

C

C axis mode 3-54

Changing program steps 7-133

CIRCLE 2-31

Contour

 Graphic representation 6-115

 Symbolic representation 6-114

CONTOUR 2-31

Contour elements

 Arc 6-117

 Creating 6-116

 Deleting 6-120

 Inserting 6-120

 Modifying 6-119

 Straight line 6-117

Contour handwheel 2-23, 4-75

Control stick 2-22

Correction keys 2-20

Create G code program 7-143

Creating a new contour 6-121

Cross-hair 9-154

Current block 7-139

Cursor keys 2-20

Cursor pad 2-21

Cut segmentation 5-83

Cutting cross-section 5-83

Cutting depth 5-83

CYCLE 2-31

Cycles 5-79

 Alarms 11-168

 Error handling 11-168

 Messages 11-169

D

Deep hole drilling 5-93

Directory 8-148

Dry run 9-156

E

EasyStep flow chart 7-130

EasyStep program step 7-131

EasyStep programming 2-30, 7-128

Edit key 2-33

Editing contour elements 6-119

Emergency stop key 2-25

Error log 8-152

ETC key 2-33

Examples 12-179

Extended soft key menu 2-35

F

Face turning 4-67

Facing 4-64

Feed 2-24, 2-39

Feed transitions 6-113

Feedrate override switch 2-23

File

 Call 8-149

 Copy 8-151

 Delete 8-149

 Read in 8-152

 Read out to an external medium 8-151

 Rename 8-151

 Select 8-149

Function keys 2-33

G

G code editor 7-141

G code programming 7-128, 7-139

 Contour elements 7-145

 Insert contour elements 7-143

 Insert turning cycles 7-143

 Modify contour elements 7-144

 Modify turning cycles 7-144

 Turning cycles 7-146

- G functions 7-139
Graphics display 9-154
Graphics interface 2-28
Graphics monitor 2-20, 2-21
Groove position 5-107
Grooving cycles 5-105
- H**
Hole circle drilling 5-97
Home key 2-33
- I**
Inch 3-61
Incremental dimension 2-37
Incremental feed mode 3-45
Information key 2-33
Insert a contour in EasyStep machining sequence 8-150
Insert G code step 7-133
Inserting program step 7-132
Intervention in the Machining Process 10-163
- M**
Machine control panel 2-22
 Sample configuration 2-22
Machine sequence programming 2-30
Machining residual material 6-125
Machining sequence 2-30, 7-128
 Create 7-129
 Display 7-130
 Start 7-135
MANUAL 2-31
Manual machining 4-75
Manual mode 4-64
Manual offset 3-47, 3-49
 Delete 3-50
ManualTurn 1-16
Measure tool 3-58
Measuring system 3-61
Message overview key 2-33
Messages 11-167
Metric 3-61
Mini handheld unit 2-26
Minus key 4-76
Modifying a contour element 6-126
- N**
NC Start key 2-24
NC Stop key 2-24
New contour 6-113
Notes 1-17
Numeric pad 2-20, 2-21
- O**
Offset 10-166
Offsets 3-47
OP 031 slimline operator panel 2-20
OP 032S slimline operator panel 2-21
Operating area switchover key 2-33
Operating mode switch 2-22
Operating modes 2-31
Operating system 2-30
Operator panel 2-20
 Control elements 2-20
Oriented spindle stop 3-53
- P**
Parts program management 8-147
Path dimension input 4-66
Plus key 4-76
Pocket calculator 2-35
Positioning 4-65
Positioning linear axis 4-70
Positioning rotary axis 4-68, 4-70
Preset 3-48
PROGRAM 2-32
Program editor 7-134
Program step 7-128
 Copying 7-134
 Delete 7-134
 Inserting 7-134
Programmed stop 7-140
- R**
R parameters 7-140
Radii machining 4-71
Recall key 2-33
Reference axes 3-42
Relief cut elements 6-124
Repositioning 10-165
RESET 2-25
Re-working a thread 5-88
- S**
Saving manual offset with "Store offset" function 10-166
Scratching 3-58
Selecting the alarm overview 11-170
Selecting the message overview 11-170
Setup 3-44
Simulation 9-155
Simultaneous recording 9-156
Single-cycle machining 2-30, 6-126
Single-step mode 2-25, 7-136

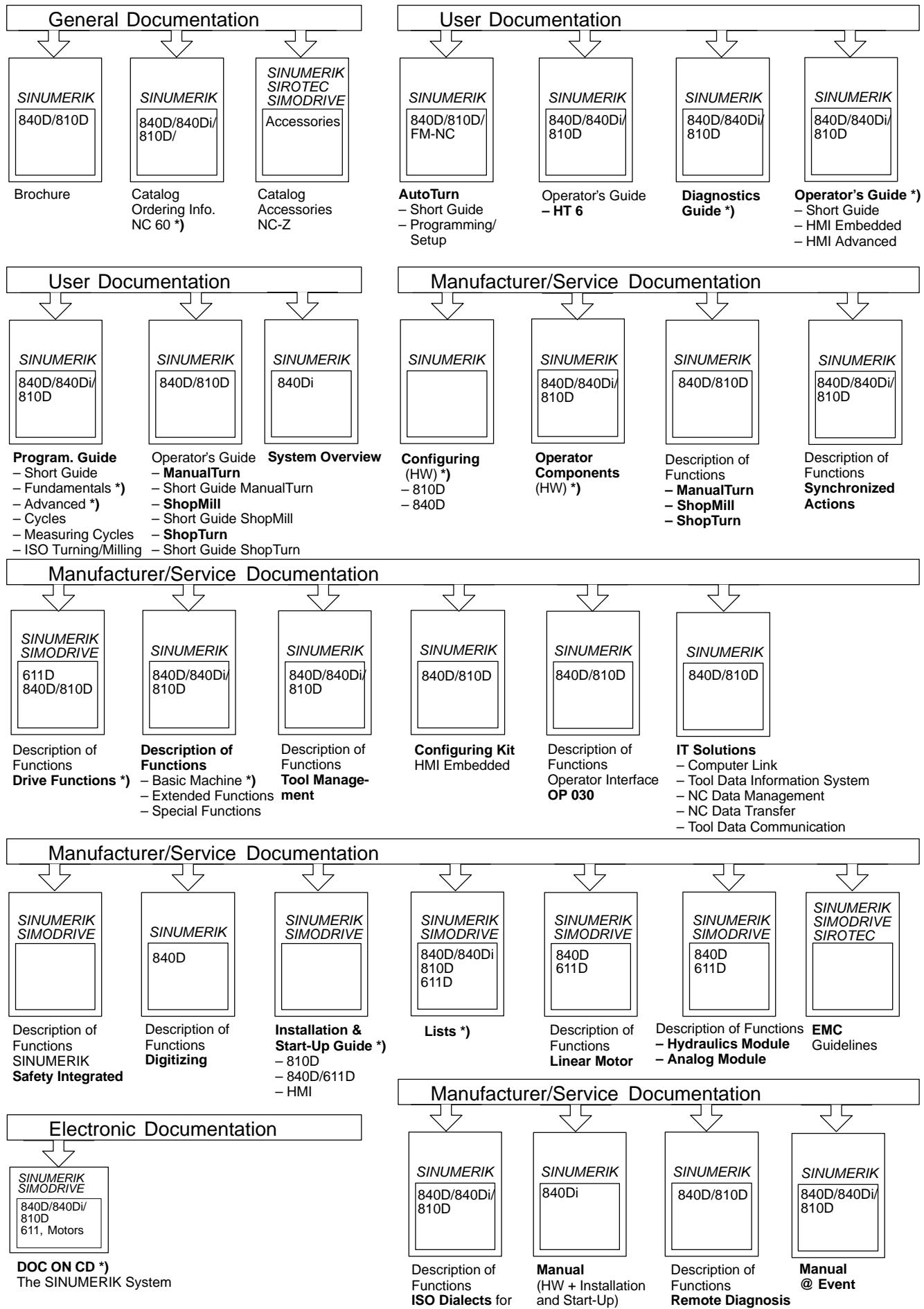
- Skip block 7-140
Soft keys 2-34
Special functions 7-131
Spindle control 2-23
Spindle speed 2-40
Spindle speed limitation 3-52
Spindle stop 3-53
Standard CNC operation 9-162
STOCK REMOVAL 2-31
Stock removal along contour 6-122
Stock removal cycles 5-100
Storing thread cycles 8-150
Storing thread undercut 8-150
STRAIGHT 2-31
Switching off 1-18
Switching off the program 7-135
Switching on 1-18
- T**
- TAPER 2-31
Taper turning 4-65, 4-69
Tapping 5-95
TEACH 2-25
Teach In 9-158
Continuing 9-159
Deselecting 9-159
Select 9-158
Thread 5-81
Multiple-start 5-86
Thread cutting 5-81
Thread cutting operations 5-87
- Thread forward travel 5-84
Thread pitch 2-39
Thread undercuts 5-91
Toggle key 2-33
Tool 2-39, 3-55
Tool approach 7-132
Tool nose radius compensation 7-137
Tool offset 3-47
Deselect 3-57
Select 3-57
Tool offset data 3-55
Tool point direction 3-56
Tool wear compensation 3-60
Transferral of auxiliary functions 9-161
Transferring machining steps to system 9-160
Transferring manual movements 9-160
Transmission log 8-152
Traversing direction 2-22
- Turning cycles
Calling 5-80
- U**
- Undercut
Form E 5-89
Form F 5-89
- Z**
- Zero offset 3-47, 3-51
Deselect 3-51
Input 3-51
Reselect 3-51

Notes

To SIEMENS AG A&D MC BMS P.O. Box 3180 D-91050 Erlangen, Germany Phone: ++49-(0)180-5050-222 [Hotline] Fax: ++49-(0)9131-98-2176 [Documentation] Email: motioncontrol.docu@erlf.siemens.de	Suggestions Corrections For Publication/Manual: SINUMERIK 840D/810D ManualTurn User Documentation
From: Name <hr/> Company/Dept. Address: <hr/> Zip Code: Town: <hr/> Phone: / <hr/> Fax: /	Operation/Programming Order No.: 6FC5298-6AD00-0BP0 Edition: 08.02 Should you come across any printing errors when reading this publication, please notify us on this sheet. Suggestions for improvement are also welcome.

Suggestions and/or corrections

Overview of SINUMERIK 840D/840Di/810D Documentation (08.2002)



*) These documents are a minimum requirement

Siemens AG
Automation & Drives
Motion Control Systems
P.O. Box 3180, D-91050 Erlangen
Germany

www.ad.siemens.de

© Siemens AG, 2002
Subject to change without prior notice
Order No: 6FC5298-6AD00-0BP0

Printed in Germany