# SINUMERIK 840D/840Di/810D

## AutoTurn

**Graphic Programming System**

Programming/Setup

---

**Applies to**

<table>
<thead>
<tr>
<th>Control</th>
<th>Software Version</th>
</tr>
</thead>
<tbody>
<tr>
<td>840D</td>
<td>4, 5, 6</td>
</tr>
<tr>
<td>840DE (export version)</td>
<td>4, 5, 6</td>
</tr>
<tr>
<td>840D Powerline</td>
<td>6</td>
</tr>
<tr>
<td>840DE Powerline (export version)</td>
<td>6</td>
</tr>
<tr>
<td>840Di</td>
<td>1, 2</td>
</tr>
<tr>
<td>840Di (export version)</td>
<td>1, 2</td>
</tr>
<tr>
<td>810D Powerline</td>
<td>6</td>
</tr>
<tr>
<td>810D Powerline (export version)</td>
<td>6</td>
</tr>
</tbody>
</table>

**02.02 Edition**

---

<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Introduction</td>
<td>1</td>
</tr>
<tr>
<td>Tools</td>
<td>2</td>
</tr>
<tr>
<td>Workholders</td>
<td>3</td>
</tr>
<tr>
<td>Cutting Values</td>
<td>4</td>
</tr>
<tr>
<td>Functions</td>
<td>5</td>
</tr>
<tr>
<td>Toolholder Preassignment</td>
<td>6</td>
</tr>
<tr>
<td>Examples of Program Sequences</td>
<td>7</td>
</tr>
<tr>
<td>Program Generation and Setup</td>
<td>8</td>
</tr>
<tr>
<td>Interactive Programming</td>
<td>9</td>
</tr>
<tr>
<td>Undefined Contour Elements</td>
<td>10</td>
</tr>
<tr>
<td>Two-Sided Machining/Counterspindle</td>
<td>11</td>
</tr>
<tr>
<td>Link to CAD</td>
<td>12</td>
</tr>
<tr>
<td>Sample Program: Sprocket Shaft</td>
<td>13</td>
</tr>
<tr>
<td>Adaptation to Machine</td>
<td>14</td>
</tr>
<tr>
<td>NC Words</td>
<td>15</td>
</tr>
<tr>
<td>NC Blocks</td>
<td>16</td>
</tr>
<tr>
<td>Error Messages</td>
<td>17</td>
</tr>
<tr>
<td>Appendix</td>
<td>A</td>
</tr>
</tbody>
</table>
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 reprint with new Order No.
C .... Revised edition with new status.

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

<table>
<thead>
<tr>
<th>Edition</th>
<th>Order No.</th>
<th>Remarks</th>
</tr>
</thead>
<tbody>
<tr>
<td>09.95</td>
<td>6FC5298-3AA40-0BP0</td>
<td>A</td>
</tr>
<tr>
<td>08.97</td>
<td>6FC5298-4AA40-0BP0</td>
<td>C</td>
</tr>
<tr>
<td>02.98</td>
<td>6FC5298-4AA40-0BP1</td>
<td>C</td>
</tr>
<tr>
<td>09.99</td>
<td>6FC5298-4AA40-0BP2</td>
<td>C</td>
</tr>
<tr>
<td>02.02</td>
<td>6FC5298-4AA40-0BP3</td>
<td>C</td>
</tr>
</tbody>
</table>

This book forms part of the documentation available on CD-ROM (DOCONCD)

<table>
<thead>
<tr>
<th>Edition</th>
<th>Order No.</th>
<th>Remarks</th>
</tr>
</thead>
<tbody>
<tr>
<td>09.02</td>
<td>6FC5 298-6CA00-0BG3</td>
<td>C</td>
</tr>
</tbody>
</table>

Registered Trademarks

SIMATIC®, SIMATIC HMI®, SIMATIC NET®, SIROTEC®, SINUMERIK® and SIMODRIVE® are registered trademarks of Siemens AG. Other names used in this publications may be trademarks of their respective owners and their use by third parties may constitute a violation of copyright.

Further information can be found under the following Internet address:
http://www.ad.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.

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.

Subject to change without prior notice.
Table of Contents

Introduction 1-17

1.1 Use of AutoTurn ................................................................. 1-18
1.2 Operating elements ............................................................ 1-19
  1.2.1 Operator keyboard ......................................................... 1-19
  1.2.2 AutoTurn PC, AutoTurn Plus PC ....................................... 1-20
1.3 Typical programming sequence ............................................ 1-22
1.4 Screen layout ........................................................................ 1-23
  1.4.1 Programming menu layout .............................................. 1-23
  1.4.2 Menu layout Setup ......................................................... 1-24
1.5 Start-up ................................................................................ 1-25
  1.5.1 Start-up SINUMERIK control ......................................... 1-25
  1.5.2 AutoTurn installation ..................................................... 1-26
1.6 Tips and tricks ..................................................................... 1-28
  1.6.1 PC settings .................................................................... 1-28
  1.6.2 Operating instructions .................................................. 1-28
  1.6.3 Miscellaneous .............................................................. 1-29

Tools 2-31

2.1 Tool groups ......................................................................... 2-32
  2.1.1 Overview ....................................................................... 2-32
  2.1.2 Operation ....................................................................... 2-33
  2.1.3 Application ..................................................................... 2-34
2.2 Tool master data ................................................................. 2-35
  2.2.1 Overview ....................................................................... 2-35
  2.2.2 Cutting edge types/fits/general data ................................. 2-36
  2.2.3 External turning tools .................................................... 2-38
  2.2.4 Boring rods .................................................................. 2-39
  2.2.5 Grooving tools .............................................................. 2-40
  2.2.6 Thread chasers .............................................................. 2-41
  2.2.7 Drilling tools .................................................................. 2-42
  2.2.8 Milling tools ................................................................. 2-44
  2.2.9 Knurling tool ................................................................. 2-45
  2.2.10 Thread cutting tools ...................................................... 2-46
2.3 Tool data dialog ................................................................. 2-47
  2.3.1 Changing/copying tool data .......................................... 2-47
  2.3.2 Creating a new tool ...................................................... 2-49
  2.3.3 Deleting a tool .............................................................. 2-50
  2.3.4 Find tool ........................................................................ 2-51
Contents

Workholders
3-53
3.1 Workholder groups .................................................................3-54
  3.1.1 Overview ......................................................................... 3-54
  3.1.2 Operation ........................................................................ 3-55

3.2 Workholder master data ..........................................................3-56
  3.2.1 General data .....................................................................3-56
  3.2.2 Outside and inside jaws .....................................................3-56
  3.2.3 Collets/Face drivers ........................................................3-56
  3.2.4 Tailstock points ............................................................... 3-57
  3.2.5 Workholder dialog ......................................................... .3-58

Cutting Values
4-59
4.1 Machining method ...............................................................4-60
  4.1.1 Overview .........................................................................4-60
  4.1.2 Operation .........................................................................4-61
  4.1.3 Description .......................................................................4-62

4.2 Cutting values dialog ..............................................................4-64
  4.2.1 Changing material data ................................................... 4-64
  4.2.2 Copy material data .......................................................... 4-66
  4.2.3 Delete material ................................................................. 4-68

Functions
5-69
5.1 Overview ..................................................................................5-70
  5.1.1 Setup functions ............................................................... 5-70
  5.1.2 Programming functions ...................................................5-70

5.2 Programming ...........................................................................5-71
  5.2.1 Description .......................................................................5-71
  5.2.2 Operation .........................................................................5-71

5.3 Program management ..............................................................5-73
  5.3.1 Description .......................................................................5-73
  5.3.2 Operation .........................................................................5-74

5.4 Blank .........................................................................................5-78
  5.4.1 Blank stocks and dimensions ..........................................5-78
  5.4.2 Free blank stocks .............................................................5-79
  5.4.3 Material selection ............................................................ 5-79

5.5 Finished part .............................................................................5-80
  5.5.1 Dimensions / Positioning ..................................................5-80
  5.5.2 Display .............................................................................5-83
  5.5.3 Outside and inside contour ...............................................5-85
  5.5.4 End face and peripheral surface ................................. 5-86
5.5.5 Cylinder/Taper/Circle................................................................. 5-91
5.5.6 Contour definition................................................................. 5-94
5.5.7 Form element........................................................................... 5-97
5.5.8 Chamfer/Rounding/Roughness height/Fit class......................... 5-98
5.5.9 Thread...................................................................................... 5-100
5.5.10 Correction............................................................................. 5-103

5.6 Mirror workpiece/machining position............................................ 5-108
5.7 Cut-off part ................................................................................ 5-108

5.8 Workholders................................................................................ 5-110
5.8.1 Machining side ....................................................................... 5-110
5.8.2 Select workholder................................................................. 5-111
5.8.3 Tailstock................................................................................ 5-114

5.9 Defining machining limits............................................................. 5-115

Toolholder Preassignment 6-117
6.1 Description .............................................................................. 6-118
6.2 Operation................................................................................. 6-119
6.3 Toolholder without counterspindle............................................... 6-121
6.4 Counterspindle machining with 1 or 2 toolholders....................... 6-122

Examples of Program Sequences 7-127
7.1 Programming example: Basic functions...................................... 7-128
7.1.1 General ............................................................................... 7-128
7.1.2 Program selection............................................................... 7-130
7.1.3 Defining the blank.............................................................. 7-131
7.1.4 Entering the finished part geometry.................................... 7-133
7.1.5 Selecting the workholder for the workpiece......................... 7-143
7.2 Programming example: Any blank forms.................................... 7-146
7.3 Programming example: End face machining................................. 7-150
7.4 Programming example: Machining on the peripheral surface........ 7-159

Program Generation and Setup 8-161
8.1 Program generation/Machining plan........................................... 8-162
8.1.1 Program generation.............................................................. 8-162
8.1.2 Machining plan..................................................................... 8-164
8.2 Simulation................................................................................ 8-166
8.3 Setup dialog............................................................................. 8-167
8.4 Program handling.................................................................... 8-168
Interactive Programming 9-169

9.1 Single-step machining ................................................................. 9-170
  9.1.1 Introduction ................................................................. 9-170
  9.1.2 Operation ................................................................. 9-175

9.2 Macros .................................................................................. 9-178
  9.2.1 General notes ................................................................. 9-178
  9.2.2 Milling ................................................................. 9-183
  9.2.3 Macro programming example ........................................ 9-185

Undefined Contour Elements 10-191

10.1 Introduction ........................................................................ 10-192

10.2 Programming example ........................................................ 10-193

Two-Sided Machining/Counterspindle 11-199

11.1 Introduction ........................................................................ 11-200

11.2 Programming example: Machining on the main spindle (two-sided machining) ...... 11-201

11.3 Programming example: Machining on the counterspindle ........................................ 11-208
  11.3.1 Special features of working with 2 toolholders ........................................... 11-214
  11.3.2 Cut off and pull forward with counterspindle ............................................ 11-215

11.4 Progr. ex.: Machining on the main spindle and counterspindle ................................ 11-217

Link to CAD 12-221

12.1 Link to CAD systems (SW 4 and higher) ............................................. 12-222

12.2 DXF boundary conditions (SW 4 and higher) .......................................... 12-222

Sample Program: Sprocket Shaft 13-223

13.1 Creating a program .................................................................. 13-224

13.2 Definition blank/machined part .................................................. 13-227

13.3 Creating the outside contour ...................................................... 13-228

13.4 Machining the end face ............................................................ 13-234
  13.4.1 Drilled holes ........................................................................ 13-234
  13.4.2 Pocket ................................................................................ 13-236

13.5 Workholder selection .................................................................. 13-240

13.6 Setting the machining limits ....................................................... 13-241

13.7 Single machining operations ..................................................... 13-242
  13.7.1 Rough face turning ............................................................ 13-243
  13.7.2 Rough turning longitudinal ................................................ 13-245
### Contents

#### 13.7 Miscellaneous Functions
- 13.7.3 Rough grooving recess ................................................................. 13-247
- 13.7.4 Finish grooving .............................................................................. 13-249
- 13.7.5 Drilling ............................................................................................. 13-250
- 13.7.6 Milling .............................................................................................. 13-251

#### 13.8 Machining plan .................................................................................. 13-252

#### Adaptation to Machine

<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>14.1 Overview</td>
<td>14-254</td>
</tr>
<tr>
<td>14.1.1 Description</td>
<td>14-254</td>
</tr>
<tr>
<td>14.1.2 Operation</td>
<td>14-255</td>
</tr>
<tr>
<td>14.2 Settings</td>
<td>14-256</td>
</tr>
<tr>
<td>14.2.1 Description</td>
<td>14-256</td>
</tr>
<tr>
<td>14.2.2 Operation</td>
<td>14-256</td>
</tr>
<tr>
<td>14.3 NC words</td>
<td>14-260</td>
</tr>
<tr>
<td>14.3.1 Description</td>
<td>14-260</td>
</tr>
<tr>
<td>14.3.2 Operation</td>
<td>14-263</td>
</tr>
<tr>
<td>14.4 NC blocks</td>
<td>14-265</td>
</tr>
<tr>
<td>14.4.1 Description</td>
<td>14-265</td>
</tr>
<tr>
<td>14.4.2 Operation</td>
<td>14-267</td>
</tr>
<tr>
<td>14.5 Time calculation</td>
<td>14-269</td>
</tr>
<tr>
<td>14.5.1 Description</td>
<td>14-269</td>
</tr>
<tr>
<td>14.5.2 Operation</td>
<td>14-270</td>
</tr>
<tr>
<td>14.6 System</td>
<td>14-271</td>
</tr>
<tr>
<td>14.6.1 Description</td>
<td>14-271</td>
</tr>
<tr>
<td>14.6.2 Operation</td>
<td>14-272</td>
</tr>
</tbody>
</table>

#### NC Words

<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>15.1 Formal word description</td>
<td>15-274</td>
</tr>
<tr>
<td>15.1.1 Information about formal words</td>
<td>15-274</td>
</tr>
<tr>
<td>15.1.2 Coordinate computation</td>
<td>15-275</td>
</tr>
<tr>
<td>15.2 Functions for turning</td>
<td>15-276</td>
</tr>
<tr>
<td>15.2.1 General functions</td>
<td>15-276</td>
</tr>
<tr>
<td>15.2.2 Tool change</td>
<td>15-277</td>
</tr>
<tr>
<td>15.2.3 Spindle and feedrate</td>
<td>15-278</td>
</tr>
<tr>
<td>15.2.4 Thread</td>
<td>15-280</td>
</tr>
<tr>
<td>15.2.5 Insertion texts</td>
<td>15-280</td>
</tr>
<tr>
<td>15.3 Special functions for C-axis machining</td>
<td>15-281</td>
</tr>
<tr>
<td>15.4 Special functions with counterspindle (SW 4 and higher)</td>
<td>15-282</td>
</tr>
</tbody>
</table>
NC Blocks 16-283

16.1 Formal block description ................................................................. 16-284
16.2 NC blocks with examples ................................................................. 16-285
16.2.1 General NC blocks ........................................................................ 16-285
16.2.2 Tool change...................................................................................... 16-286
16.2.3 Normal turning .............................................................................. 16-289
16.2.4 C axis .......................................................................................... 16-292
16.2.5 Counterspindle ............................................................................. 16-295
16.3 Assignment between machining operations and blocks called .......... 16-301

Error Messages 17-303

17.1 Introduction....................................................................................... 17-304
17.2 Alphabetical listing of error messages ............................................ 17-305

Appendix A-319

A References .......................................................................................... A-320
B Index .................................................................................................. A-335
Structure of the documentation

The SINUMERIK documentation comprises three parts:

- General Documentation
- User Documentation
- Manufacturer/Service Documentation

Reader group

This manual is intended for users of machine tools. This publication provides detailed information that the user requires for operating the SINUMERIK 840D, 840Di and 810D controls with the AutoTurn Graphic Programming System.

Hotline

If you have any queries, please contact the hotline:

A&D Technical Support: Tel. +49 (180) 5050 – 222
Fax: +49 (180) 5050 – 223
email: techsupport@ad.siemens.de

Please send any queries about the documentation (suggestion, correction) per fax or email to the following address:

Fax: +49 (9131) 98 – 2176
email: motioncontrol.docu@erlf.siemens.de

Fax form: see return form on the last page of the publication

Internet address

http://www.ad.siemens.de/sinumerik

SINUMERIK 840D powerline

As of 09.2001,
- SINUMERIK 840D powerline and
- SINUMERIK 840DE powerline are available with improved performance. A list of available powerline modules is provided in the hardware description:

References: /PHD/, Configuring Manual SINUMERIK 840D

SINUMERIK 810D powerline

As of 12.2001,
- SINUMERIK 810D powerline and
- SINUMERIK 810DE powerline are available with improved performance. A list of available powerline modules is provided in the hardware description:

References: /PHC/, Configuring Manual SINUMERIK 810D
Preface

Proper use

Scope of the documentation

This Operator's Guide describes the standard functionality. A description of add-on features or modifications made by the machine manufacturer is provided in a separate documentation by the machine manufacturer.

This Operator's Guide describes only the functions and operations for the option "AutoTurn Graphic Programming System".

Please read the general Operator's Guides for the SINUMERIK 840D/840Di/810D controls before you use the AutoTurn Graphic Programming System. They have priority and should be consulted in conjunction with the present Operator's Guide!

- The general operation of the SINUMERIK 840D and 810D controls is explained in the User Documentation "Operator's Guide SINUMERIK 840D/810D Order No. 6FC5 298-6AA00-0BP0".
- The general operation of the SINUMERIK 840Di control is explained in the Manufacturer Documentation "SINUMERIK 840Di Manual Order No. 6FC5 297-6AE60-0BP0".

Programming/Setup

This documentation satisfies the following requirements:

- It gives the machine operator detailed information about part programming using programming examples and guides him through all the menus.
- It describes all operator actions that the machine setter requires for startup, installation, adaptation to the machine and parameterization.

Short Guide

In addition to this detailed Operator's Guide, there is also a "Short Guide AutoTurn Operation, order no. 6FC5 298-4AA30-0BP2" in handy A5 format which is available to the machine operator as a ready reference for operating and programming.

For information about other publications relating to the SINUMERIK 840D, 840Di and 810D controls and on documentation concerning all SINUMERIK controls, please contact your local Siemens office.
Validity

This Operator's Guide applies to the following controls:
- SINUMERIK 840D, 840DE SW 4, 5, 6
- SINUMERIK 840Di SW 1, 2
- SINUMERIK 810D powerline SW 6 and higher with MMC 103 (HW: MMC 103), HMI Advanced (HW: PCU 50)

Explanation of symbols

Sequence of operations

The section below provides an overview of the keystroke sequence required. If inputs are necessary in the individual steps, or if further information is required, a description is given next to the key symbols.

Explanation

This symbol is used for important explanations.

Function

Following this symbol, you may obtain vital information for a better understanding of the operating functions.

Parameters

Under this symbol you will find detailed descriptions of data and parameters.

Programming example

Here, the programming described is explained using an example.

Programming

This symbol is used for important programming notes.
Further notes

For safety reasons, some functions are locked against unauthorized access. The machine manufacturer can influence or modify the function behavior described. Please observe the information given by the machine manufacturer!

The following notes with special significance are used in the documentation:

This symbol appears in this documentation wherever more detailed information is given or where attention must be paid to important information.

In this documentation you will find this symbol with information about an Ordering Data Option. The function described can only be executed if the control contains this option.

The following 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.

References
This symbol appears wherever specific information can be found in more detailed reference literature.
The Appendix contains a complete list of references.

Basis
Your SIEMENS 840D/840Di/810D is state of the art and complies with recognized safety rules, standards and regulations.

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 setup, 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).
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 Operator's Guide.
- 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 may result in unforeseen dangers for:
- The health and life of people
- The control, machine and other property of the operating company and user.
Introduction

1.1 Use of AutoTurn ............................................................................................................. 1-18
1.2 Operating elements ........................................................................................................ 1-19
  1.2.1 Operator keyboard ................................................................................................... 1-19
  1.2.2 AutoTurn PC, AutoTurn Plus PC ............................................................................. 1-20
1.3 Typical programming sequence ...................................................................................... 1-22
1.4 Screen layout .................................................................................................................. 1-23
  1.4.1 Programming menu layout ....................................................................................... 1-23
  1.4.2 Menu layout Setup ................................................................................................... 1-24
1.5 Start-up .................................................................................................................... ........ 1-25
  1.5.1 Start-up SINUMERIK control ................................................................................... 1-25
  1.5.2 AutoTurn installation .............................................................................................. 1-26
1.6 Tips and tricks ............................................................................................................. .... 1-28
  1.6.1 PC settings............................................................................................................... 1-28
  1.6.2 Operating instructions .............................................................................................. 1-28
  1.6.3 Miscellaneous .......................................................................................................... 1-29
1.1 Use of AutoTurn

AutoTurn and AutoTurn Plus are workshop-oriented graphic programming systems. They are used in the workshop directly on the CNC lathe and can be used with SINUMERIK systems 840D/840Di and 810D.

The software has been designed in such a way as to allow skilled operators to incorporate CNC programs in a way that is familiar to them.

It's just a short step from the workpiece drawing to the workpiece, even for complex parts. This is supported by a comprehensive technology database, automatic machining plan generation (one touch) and the programming of blanks and finished parts:

- Consistent user guidance runs like a thread through the entire system.
- AutoTurn increases productivity for unit and small batch production and for medium-sized batches considerably.

Please also note that AutoTurn PC and AutoTurn Plus PC are only delivered in conjunction with SinuTrain Plus.

- The PC products run under Win 95/98 and Win NT,
- and are intended for production planning and for training purposes.

Programs generated with them can be transferred directly to the turning machine in the workshop where they can then be executed. Programs are transferred via the RS232C (V24) interface, diskette or Ethernet network.

AutoTurn supports the programming and simulation of the main spindle behind the turning center. Notwithstanding, the generated parts program can also be used before the turning center.
1.2 Operating elements

1.2.1 Operator keyboard

All elements of the operator keyboard required to operate the AutoTurn Graphic Programming System are depicted and explained below:

**Softkeys**
These 16 keys (8 in the horizontal softkey bar and 8 in the vertical softkey bar) are assigned different functions depending on the menu which is currently selected.

**Recall key**
You return to the higher-level menu without storing the values entered in the window which is currently open.

**Menu extension key, etc.**
This key extends the horizontal softkey bar within the same menu.

**Shift key**
When the "Shift" key is pressed, the upper key function is activated for keys with a double assignment (like on the OP 031 operator panel).

**Cursor keys**
With these keys you can move the cursor up, down, left and right.

**Next input block (End of line) key**
This key is particularly important for adaptation to the machine. You use it to activate entry boxes.

**Page down, Page up**
With these keys you can change the portion of a display which is visible on the screen, i.e. page the screen up and down. The scroll bars indicate the position of the visible portion.
1.2 Operating elements

Delete key (backspace)
Every time you press this key, you delete one character to the left of the cursor.

Blank
You can write blanks with this key.

Numeric input

Alphabetic input
This Operator's Guide shows the alphanumeric keyboard with dual-assigned keys on the slimline OP 031 operator panel.
You can access the alphabetic keys by pressing the "Shift" key.
If your machine is equipped with a full CNC keyboard with QWERTY layout (on the OP 032), you do not have to press "Shift" for the alphabetic keys (single assignment).

Input key
This key is used to accept an edited value or to activate/deactivate a field.

1.2.2 AutoTurn PC, AutoTurn Plus PC

The following table shows how you emulate the functionality of the softkeys and special keys for the AutoTurn Graphic Programming System on the PC keyboard.

Further notes
You can connect a standard PC keyboard (MF-II), order no. 6FC5 203-0AC01-0AA0, to the MMC 103 and to the PCU 50 submodules.

Caution
The standard PC keyboard does not meet the same standards (e.g. EMC standards) as a SINUMERIK control. It should, therefore, only be used for installation and service tasks.
### 1.2 Operating elements

<table>
<thead>
<tr>
<th>PC keyboard</th>
<th>AutoTurn function</th>
</tr>
</thead>
<tbody>
<tr>
<td>F1 to F8</td>
<td>Horizontal softkeys 1 to 8</td>
</tr>
<tr>
<td>Shift + F1 to F8</td>
<td>Vertical softkeys 1 to 8</td>
</tr>
<tr>
<td>F9</td>
<td>Recall</td>
</tr>
<tr>
<td>Shift + F9</td>
<td>Menu extension, etc.</td>
</tr>
<tr>
<td>Page up</td>
<td>Page up</td>
</tr>
<tr>
<td>Page down</td>
<td>Page down</td>
</tr>
<tr>
<td>Tab</td>
<td>Next input block</td>
</tr>
<tr>
<td>Enter</td>
<td>Input</td>
</tr>
<tr>
<td>F10</td>
<td>Operating area switchover</td>
</tr>
</tbody>
</table>

The other operations such as "shift" or cursor control are identical on the control operator keyboard and the standard PC keyboard.
This chapter illustrates the typical program sequence for AutoTurn. Starting at programming through to machine setup. Not all the individual steps listed are necessary for machining a workpiece.

<table>
<thead>
<tr>
<th>Step</th>
<th>described in Section</th>
</tr>
</thead>
<tbody>
<tr>
<td>Programs</td>
<td>Create new program 5.2</td>
</tr>
<tr>
<td>Define a blank</td>
<td>Select a form for a blank 5.4.1, Free blank form (as of SW 4.x) 5.4.2, Select material 5.4.3</td>
</tr>
<tr>
<td>Define finished part</td>
<td>Basic dimensions and positioning of finished part 5.5.1, Described outside and inside contour 5.5.3, Machine end face and peripheral surfaces (as of SW 4.x) 5.5.4, Cut off workpiece 5.7, Single machining operations 9.1, Work with macros 9.2, Two-sided machining 11</td>
</tr>
<tr>
<td>Chuck</td>
<td>Define workpiece position 5.6 and 5.8.1, Select chuck 5.8.2, Select tailstock 5.8.3</td>
</tr>
<tr>
<td>Machining limits</td>
<td>Set the machining limits for outside and inside contour 5.9</td>
</tr>
<tr>
<td>Program creation</td>
<td>Create program with automatic function 8.1.1, Define workplan 8.1.2, Simulate machining process 8.2</td>
</tr>
<tr>
<td>Setup dialog</td>
<td>Define tools in toolholder 8.3</td>
</tr>
</tbody>
</table>
1.4 Screen layout

1.4.1 Programming menu layout

Display fields and operator control elements:

1. Vertical softkey bar with 8 softkeys:
   - Inputs: Modification of diagrams

2. Menu bar:
   - Displays left: Active menu
   - Displays right: Active part

3. Menu contents

4. Horizontal softkey bar with 8 softkeys:
   - Inputs for programming
1.4.2 Menu layout Setup

An overview of the menus described in Chapters 2/3/4/14/15/16 is given below:

In the "System" menu, the machine manufacturer defines the access authorization for softkeys 2 to 5 of machine adaptation.
1.5 Start-up

1.5.1 Start-up SINUMERIK control

Note for the machine manufacturer:

The SINUMERIK 840D, 840Di or 810D controls must be started up as specified in the relevant Installation & Start-up Guides:

- The "SINUMERIK 840D/SIMODRIVE 611D Installation & Start-up Guide, Order No. 6FC5 297-6AB10-0BP1" is binding for the SINUMERIK 840D control.
- The "SINUMERIK 840Di Manual, Order No. 6FC5 297-5AE60-0BP0" is binding for the SINUMERIK 840Di control.
- The "SINUMERIK 810D Installation & Start-up Guide, Order No. 6FC5 297-4AD20-0BP1" is binding for the SINUMERIK 810D control.

First start-up
Series machine start-up

The operator actions specified in the above guides for first and series machine startup must be performed and successfully completed before the AutoTurn Graphic Programming System is installed! The SINUMERIK control must be in a fully functional state!
1.5.2 AutoTurn installation

Installation from diskette on MMC 103/PCU 50

For the most recent installation guide, please read the readme file (German or English) on the diskette.

You will also find current information on
- Installation on MMC 103/PCU 50
- Installation on MMC 103/PCU 50 via RS232C in this readme file.

Please observe the connection conditions of the MMC 103 module and the PCU 50 in: SINUMERIK 840D/840Di/810D Operator Components Manual, Manufacturer Documentation Order No. 6FC5 297-6AA50-0BP1.

Installation on PC

Your PC must have the following specification:
- Pentium 333 MHz or higher,
- at least 32 MB RAM (64 MB recommended),
- at least 340 MB hard-disk storage,
- Windows 95/98/NT operating system (country setting: United States)
- CD-ROM drive.

For the most recent installation guide, please read the readme file (German or English) on the SinuTrain Plus-CD.
Three ordering variants are available for the AutoTurn Graphic Programming System with the following order numbers:

For the machine (not on PC)

1. **AutoTurn** 6FC5253-0GX00-0AG0
   with one toolholder (slide) and with C axis

2. **AutoTurn Plus** 6FC5253-0GX01-0AG0
   with two toolholders, counterspindle and C axes

For production planning (on PC)
Software on CD-ROM including documentation

3. **SinuTrain Plus** 6FC5270-0AX71-0AG0
   Includes AutoTurn and AutoTurn Plus for PC. The particular version can be selected during installation.

**Further notes**

The explanatory text in Chapters 5 and 7 always refers to one spindle (main spindle) and one toolholder. Once you have learned the programming steps for one workpiece on one spindle, please refer to Chapter 11 for a summarized description of the programming functionality provided for machines with a main spindle and a counterspindle.
1.6 Tips and tricks

1.6.1 PC settings

**Small fonts**
The font size for the display must be set to "small fonts"; otherwise, the applications will not be displayed properly.
Please correct, if necessary
Start > Settings > Control Panel > Display ("Settings" tab).

**Country settings**
Please correct the following settings, if necessary:
- Separator for decimal places must be a dot ".",
- the number of decimal places must be 3,
- The symbol for digital grouping must be ",," via
  Start > Settings > Control Panel > Country Settings
  ("Numbers" tab).

1.6.2 Operating instructions

**Printing NC blocks/words**
Proceed as follows:
1. After you have pressed the "Start-Up" softkey, enter a password.
2. Press the "MMC" softkey to open the editor.
3. Call the WA.ini file in the ADD-ON directory.
4. Add the following entry to the last line:
   
   [PRINT]
   PRINT=1

The "PRINT" softkey now appears on the vertical softkey panel for NC blocks (as of V05.02.10 also NC words) in the machine adaptation.
- When you press the "PRINT" softkey, the file SENTENCE.PRT is copied to the clipboard CLP.
- It can now be copied from the clipboard to a diskette.
- You can use an editor to read and print this file.
Please do so prior to upgrading!
**Sign for K parameter**

Editable sign of K parameter for G331 and G332 (rigid tapping):

In the "wp.ini" file (SW V05.02.10 only) in the directory "ADD_ON", it is possible to assign the value "1" or "-1" to the "Factor" parameter:

Excerpt from "wp.ini" file:

```ini
... 
[G331K]
Factor=1
... 
```

This enables you to edit the K pitch, since the direction of rotation may differ for powered workpieces and powered tools.

If the "Factor" is assigned the value "-1", the direction of rotation of the powered tool is changed.

**Saving data blocks**

In the machine adaptation, it is now possible to create several data blocks (e.g. for different machine types). A subdirectory is created in `AUTOTURN.DIR` to enable you to do this. You can choose any directory in the "Save/Load" input mask.

**Mouse operation**

As of SW 5.2, mouse support is offered not only for the softkey panel but also for all of the input masks. Lists (e.g. work plan, cutting materials, etc.) can be scrolled using the cursor keys.

### 1.6.3 Miscellaneous

You can obtain up-to-date information on the Internet at the following site:

- [www.siemens.de/sinutrain](http://www.siemens.de/sinutrain)
- [www.siemens.de/jobshop](http://www.siemens.de/jobshop)

The topic **AutoTurn** contains latest news about the AutoTurn Graphic Programming System.
1.6 Tips and tricks

MMC 103  PCU 50
Tools

2.1 Tool groups ................................................................................................................. 2-32
  2.1.1 Overview .............................................................................................................. 2-32
  2.1.2 Operation ............................................................................................................ 2-33
  2.1.3 Application ......................................................................................................... 2-34

2.2 Tool master data ........................................................................................................ 2-35
  2.2.1 Overview .............................................................................................................. 2-35
  2.2.2 Cutting edge types/fits/general data ................................................................... 2-36
  2.2.3 External turning tools .......................................................................................... 2-38
  2.2.4 Boring rods .......................................................................................................... 2-39
  2.2.5 Grooving tools .................................................................................................... 2-40
  2.2.6 Thread chaser ...................................................................................................... 2-41
  2.2.7 Drilling tools ......................................................................................................... 2-42
  2.2.8 Milling tools ......................................................................................................... 2-44
  2.2.9 Knurling tool ....................................................................................................... 2-45
  2.2.10 Thread cutting tools .......................................................................................... 2-46

2.3 Tool data dialog ........................................................................................................ 2-47
  2.3.1 Changing/copying tool data ................................................................................ 2-47
  2.3.2 Creating a new tool ............................................................................................. 2-49
  2.3.3 Deleting a tool ..................................................................................................... 2-50
  2.3.4 Find tool .............................................................................................................. 2-51
2.1 Tool groups

2.1.1 Overview

The tools are divided into different groups:

<table>
<thead>
<tr>
<th>Number</th>
<th>Tool group</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Face+Longitudinal turning tools</td>
</tr>
<tr>
<td>2</td>
<td>Facing tools</td>
</tr>
<tr>
<td>3</td>
<td>Longitudinal turning tools</td>
</tr>
<tr>
<td>4</td>
<td>Copying tools</td>
</tr>
<tr>
<td>5</td>
<td>Face+Longitudinal boring bars</td>
</tr>
<tr>
<td>6</td>
<td>Facing boring bars</td>
</tr>
<tr>
<td>7</td>
<td>Longitudinal boring bars</td>
</tr>
<tr>
<td>8</td>
<td>Copying boring bars</td>
</tr>
<tr>
<td>9</td>
<td>External grooving tools</td>
</tr>
<tr>
<td>A</td>
<td>Internal grooving tools</td>
</tr>
<tr>
<td>B</td>
<td>Face grooving tools</td>
</tr>
<tr>
<td>C</td>
<td>Cut-off tools</td>
</tr>
<tr>
<td>D</td>
<td>External thread chasers</td>
</tr>
<tr>
<td>E</td>
<td>Internal thread chasers</td>
</tr>
<tr>
<td>F</td>
<td>Spot drills</td>
</tr>
<tr>
<td>G</td>
<td>Center drills</td>
</tr>
<tr>
<td>H</td>
<td>Drills</td>
</tr>
<tr>
<td>I</td>
<td>Step drills</td>
</tr>
<tr>
<td>J</td>
<td>Insert drills</td>
</tr>
<tr>
<td>K</td>
<td>Countersinks</td>
</tr>
<tr>
<td>M</td>
<td>Reamers</td>
</tr>
<tr>
<td>N</td>
<td>Threading taps</td>
</tr>
<tr>
<td>O</td>
<td>Stock stops</td>
</tr>
<tr>
<td>P</td>
<td>Knurling tools</td>
</tr>
<tr>
<td>Q</td>
<td>End mills</td>
</tr>
<tr>
<td>R</td>
<td>Fluted end mills</td>
</tr>
<tr>
<td>Y</td>
<td>Cutting dies</td>
</tr>
</tbody>
</table>
AutoTurn assigns tools for different machining operations to the above groups: Note that tools are not pre-defined in each group; currently, approx. 50 tools are pre-defined in the graphic programming system.

You can define your own tools and assign them to the groups!

### 2.1.2 Operation

**Sequence of operations**

You are in the AutoTurn main menu, press the "Tools" softkey and the list of tool groups appears on the screen.

Use the cursor keys to page through the list or select a tool group directly by entering a number or letter.

Press the "List groups" softkey to display the total number of defined tools and the number of tools per tool group.

You return to the tool main menu.
2.1.3 Application

Unfortunately, it is not possible in AutoTurn to mark tools according to their application. The table below is therefore provided as a means of orientation:

<table>
<thead>
<tr>
<th>Technology</th>
<th>Tool group</th>
</tr>
</thead>
<tbody>
<tr>
<td>External machining</td>
<td>Face+Longitudinal turning tools, Facing tools</td>
</tr>
<tr>
<td></td>
<td>Longitudinal turning tools, Copying tools</td>
</tr>
<tr>
<td>Internal machining</td>
<td>Face+Longit. boring bars, Facing boring bars</td>
</tr>
<tr>
<td></td>
<td>Longitudinal boring bars, Copying boring bars</td>
</tr>
<tr>
<td>External roughing</td>
<td>Face+Longitudinal turning tools, Longitudinal turning tools:</td>
</tr>
<tr>
<td>long</td>
<td>- Quadrant 3</td>
</tr>
<tr>
<td></td>
<td>- Setting angle &gt;= 90° &lt;= 100°</td>
</tr>
<tr>
<td></td>
<td>- Largest insert angle</td>
</tr>
<tr>
<td></td>
<td>Copying tool:</td>
</tr>
<tr>
<td></td>
<td>- Quadrant 3</td>
</tr>
<tr>
<td></td>
<td>- Setting angle &gt;= 90° &lt; 95°</td>
</tr>
<tr>
<td></td>
<td>- Insert angle &gt;= 55°</td>
</tr>
<tr>
<td>External face</td>
<td>Face+Longitudinal turning tools:</td>
</tr>
<tr>
<td>roughing</td>
<td>- Quadrant 3</td>
</tr>
<tr>
<td></td>
<td>- Setting angle &gt;= 90° &lt;= 100°</td>
</tr>
<tr>
<td></td>
<td>- Largest insert angle</td>
</tr>
<tr>
<td></td>
<td>Facing tool:</td>
</tr>
<tr>
<td></td>
<td>- Quadrant 3</td>
</tr>
<tr>
<td></td>
<td>- Setting angle insert &gt;= 90°</td>
</tr>
<tr>
<td></td>
<td>- Largest insert angle</td>
</tr>
<tr>
<td>External finishing</td>
<td>Copying tool:</td>
</tr>
<tr>
<td></td>
<td>- Quadrant 3, smallest board angle</td>
</tr>
<tr>
<td></td>
<td>- Setting angle &gt; 105°, clearance angle &gt;= 17°</td>
</tr>
<tr>
<td></td>
<td>- Setting angle &gt; 90°, clearance angle &gt;= 30°</td>
</tr>
<tr>
<td>Internal roughing</td>
<td>Face+Longitudinal boring bars, Longitudinal boring bars:</td>
</tr>
<tr>
<td>longitudinal</td>
<td>- Quadrant 2</td>
</tr>
<tr>
<td></td>
<td>- Setting angle &gt;= 90°</td>
</tr>
<tr>
<td></td>
<td>- Largest possible Dmin</td>
</tr>
<tr>
<td>Internal face</td>
<td>Face+Longitudinal boring bars:</td>
</tr>
<tr>
<td>roughing</td>
<td>- Quadrant 2</td>
</tr>
<tr>
<td></td>
<td>- Setting angle &gt;= 90°</td>
</tr>
<tr>
<td></td>
<td>- Largest possible Dmin</td>
</tr>
<tr>
<td></td>
<td>Facing boring bars:</td>
</tr>
<tr>
<td></td>
<td>- Quadrant 2</td>
</tr>
<tr>
<td></td>
<td>- Setting angle &gt;= 90°</td>
</tr>
<tr>
<td></td>
<td>- Largest possible Dmin</td>
</tr>
<tr>
<td></td>
<td>Copying boring bars:</td>
</tr>
<tr>
<td></td>
<td>- Quadrant 2</td>
</tr>
<tr>
<td></td>
<td>- Setting angle &gt; 105°, clearance angle &gt;= 17°</td>
</tr>
<tr>
<td></td>
<td>- Largest possible Dmin</td>
</tr>
<tr>
<td>Internal finishing</td>
<td>Facing boring bars:</td>
</tr>
<tr>
<td></td>
<td>- Quadrant 2</td>
</tr>
<tr>
<td></td>
<td>- Setting angle &gt;= 90°</td>
</tr>
<tr>
<td></td>
<td>- Largest possible Dmin</td>
</tr>
</tbody>
</table>
2.2 Tool master data

2.2.1 Overview

Using the implemented tools as an example, the section below provides a detailed description of all the tool data used.

The handling of these tool data is explained in Section 2.3 "Tool data dialog"!

AutoTurn only supports the entry of tools for machining behind the turning center.

Sequence of operations

Call up the "Tools" menu from the main menu of AutoTurn with

and

go to the window "Tool/Change-Copy".

By pressing the "Page +/–" softkey, you can open the second page ("Cutting edge", if it exists!) and page back to the first page.

The "Neutral view" softkey presents the tool with neutral dimensions (L/Q dimensions),

and the "Machine view" softkey presents the tool in the machine coordinate system in which it is clamped (X/Z dimensions).
2.2 Tool master data

2.2.2 Cutting edge types/fits/general data

With turning tools you have the support of the following insert types under the tool data “Cutting edge type”:

<table>
<thead>
<tr>
<th>Type</th>
<th>Insert</th>
</tr>
</thead>
<tbody>
<tr>
<td>V</td>
<td>35° insert angle rhombic</td>
</tr>
<tr>
<td>D</td>
<td>55° insert angle rhombic</td>
</tr>
<tr>
<td>K</td>
<td>55° insert angle rhombic</td>
</tr>
<tr>
<td>E</td>
<td>75° insert angle rhombic</td>
</tr>
<tr>
<td>C</td>
<td>80° insert angle rhombic</td>
</tr>
<tr>
<td>B</td>
<td>82° insert angle rhombic</td>
</tr>
<tr>
<td>A</td>
<td>85° insert angle rhombic</td>
</tr>
<tr>
<td>M</td>
<td>86° insert angle rhombic</td>
</tr>
<tr>
<td>W</td>
<td>80° insert angle triangular</td>
</tr>
<tr>
<td>T</td>
<td>60° insert angle triangular</td>
</tr>
<tr>
<td>S</td>
<td>90° insert angle square</td>
</tr>
<tr>
<td>L</td>
<td>90° insert angle rectangular</td>
</tr>
<tr>
<td>R</td>
<td>Circular insert</td>
</tr>
</tbody>
</table>

Sequence of operations

You position the cursor on the appropriate field and press the "Help" softkey and the cutting edge type is shown on the screen with all its dimensions.

For reamers and countersinks, the fit class/type is entered under "Fit" (e.g.: H7, H10).
The following general tool data are valid for all tools and are not repeated in the specific data description:

- ID number: up to 20 characters (input under "Create new")
- Name: up to 30 characters
- Comment: up to 60 characters (e.g. SPACETOOL)
- Cutter material: up to 5 characters (up to 8 cutter materials per machining process)
- Length dimension L: (in "Neutral view")
- Setting dimension X: (in "Machine view")
- Cross dimension Q: (in "Neutral view")
- Setting dimension Z: (in "Machine view")
- Holder diameter DH:

The ID number refers to the tool. You can also find this ID number in the setup dialog, in the machining process plan and in the simulation.
The dimensions are shown in the tool diagrams which follow.
2.2.3 External turning tools

For the turning tools of the tool groups

FACE+LONGITUDINAL
TURNING TOOLS
FACING TOOLS
LONGITUDINAL TURNING
TOOLS
COPYING TOOLS

please enter the following data:

- Tool type: R/L/N (right/left/neutral)
- Cutting edge visible/invisible (the spindle rotation is output depending on this parameter, (M3/M4): S/U
- Collision dimension KE:
- Collision dimension KK:
- Collision dimension KF:
- Max. plunge angle EW:
- Clear. angle FW:
- Cutting edge length SLA:
- (Cutting edge) radius:
- Quadrant:
- Max. cutting depth longitudinal:
- Max. cutting depth face:
2.2.4 Boring rods

For the tools of the groups

FACE+LONGITUDINAL
BORING BARS
FACING BORING BARS
LONGIT. BORING BARS
COPY-TURN BORING BARS

please enter the following data:

- Tool type R/L/N (right/left/neutral)
- Cutting edge visible/invisible: S/U
- Collision dimension KE:
- Collision dimension KK:
- Collision dimension KF:
- Max. plunge angle EW:
- Minimum turning diameter DMIN:
- Mounting/clamping angle: 180° for main spindle (left), 360° for counterspindle (right)
- Clearance angle FW:
- Cutting edge length SLA:
- (Cutting edge) radius:
- Quadrant:
- Max. cutting depth longitudinal:
- Max. cutting depth face:
2.2.5 Grooving tools

For the tools of the groups

EXTERNAL GROOVING TOOLS
CUT-OFF TOOLS

the following data are available:

- Tool type: R/L/N (right/left/neutral)
- Cutting edge visible/invisible: S/U
- Collision dimension KE:
- Collision dimension KK:
- Mounting/clamping angle: 270°
- Grooving tool width ST:

And for the group

INTERNAL GROOVING TOOLS

you also require:

- Mounting/clamping angle 180°
- Min. turning diameter DMIN:

And for the group

FACE GROOVING TOOLS

you also require:

- Mounting/clamping angle 180°
- Min. turning diameter DMIN:
- Max. turning diameter DMAX:
2.2.6 Thread chasers

Thread chasers of the tool group

EXTERNAL THREAD CHASERS are defined by the following data:
- Tool type: R/L/N (right/left/neutral)
- Cutting edge visible/invisible: S/U
- Collision dimension KE:
- Collision dimension KK:
- Mounting/clamping angle: 270°
- Cutting edge length SLA:
- (Thread) lead STE:
- Measuring quadrant: (not relevant)

And for the group

INTERNAL THREAD CHASERS you also require:
- Mounting/clamping angle 180°
- Min. turning diameter DMIN:
2.2.7 Drilling tools

For drills, countersinks and reamers of the following groups, the data below are relevant:

**SPOT DRILLS**
- Drills
- Countersinks

- Direction of rotation: 4/3 (equivalent to M04/M03)
- Collision dimension KK:
- Tool diameter D1:
- Tool angle W1 (tip angle):
- Max. drilling depth T1:
- Transmission ratio: = 0: rigid tool ≠ 0: powered tool
- Mounting/clamping angle: 180° for main spindle 360° for counterspindle
- Measuring quadrant: (not relevant)

**CENTER DRILLS**
Here some additional data are required for the spot drill:
- (Small) tool diameter D1:
- (Large) tool diameter D2:
- Tool angle W1 (tip angle):
- Tool angle W2 (to D2):
- Max. drilling depth T1 (to D1):
- Max. drilling depth T2 (to D2):

A value ≠ 0 must be entered for the transmission ratio for powered tools (C axis machining).
For the following tools these data must be entered in addition to those for the spotdrills:

**STEP DRILLS**
- (Small) tool diameter D1:
- (Large) tool diameter D2:
- Tool angle W1 (tip angle):
- Tool angle W2 (to D2):
- Max. drilling depth T1 (to D1):
- Max. drilling depth T2 (to D2):

**INSERT DRILLS**
- Tool type: R/L/N (right/left/neutral)
- Cutting edge visible/invisible: S/U
- Cutting edge length SLA:
- Quadrant 1:
- Max. clamping depth longitudinal T2:
- Mounting/clamping angle

**REAMERS**
- Collision dimension KF:
- Fitting:

**THREAD TAPS**
- Collision dimension KF:
- (Thread) lead ST:

**LIMIT STOP**
- Diameter D:
2.2.8 Milling tools

For the milling cutters of tool groups

END MILLS
FLUTED END MILLS

you must not only enter the general data but also the following specific data:

• Direction of rotation: 4/3 (equivalent to M03/M04)
• Collision dimension KK:
• Milling cutter length KE:
• Milling cutter diameter D1:
• Shank diameter D2 (e.g.: for side mills):
• Max. milling depth T:
• Number of teeth (number of cutting edges):
• Transmission ratio: ≠ 0 for all milling tools
• Mounting/clamping angle: 180°, 270°, 360° (see figure)
• Measuring quadrant: (not relevant)
2.2.9 Knurling tool

Knurling tools are defined by the following data:

- Direction of rotation:
- Setting dimensions X, Z:
- Tool diameter D1:
- Knurling width T1:
- Division:
- Mounting/clamping angle: 180°
- Measuring quadrant:
2.2.10 Thread cutting tools

Thread cutting tools are defined by the following data:

- Direction of rotation:
- Setting dimensions X, Z:
- Holder diameter DH:
- Collision dimension KK:
- Max. drilling depth T1:
- Cutting edge type:
- Lead in mm/rev.:
- Thread diameter D1:
- External diameter:
- Transmission ratio:
- Mounting/clamping angle: 180°
- Measuring quadrant:
2.3 Tool data dialog

2.3.1 Changing/copying tool data

Sequence of operations

Press the "Tools" softkey in the AutoTurn main menu to obtain a list of the tool groups.

Use the cursor keys

select a tool group directly by entering a number or letter.

Press the "Change/copy" softkey,

Select the ID number displayed for the tool using the cursor keys and press "OK" to confirm.

The one or two-page input window for your tool data appears on the screen (see Section 2.2 "Tool master data").
Sequence of operations

With the "Page +/-" softkey, you can page up and down in the two-page input window.

If several tools are already defined in the tool group, you can switch between the tools using these softkeys.

If you press the "Neutral view" softkey the tool is presented in neutral dimensions (L/Q dimensions).

With the "Machine view" softkey, the tool is shown in its real clamped state (machine coordinate system) (X/Z dimensions).

Place the cursor on the required input field, write the new values into the input fields, and confirm with "Input".

If you want to accept the changed values for the current tool, press the "OK" softkey and answer the query "Save altered values?" by pressing the appropriate softkey.
2.3 Tool data dialog

Sequence of operations

To copy the changed values into a new tool, press the "Copy" softkey.

Enter ID number of your new tool and press "Input". You can follow the ID number in the toolholder default, in the process plan and in the simulation.

When you confirm with "OK", the new tool appears in the selected tool group under the new ID number.

Press "End" to quit.

2.3.2 Creating a new tool

Sequence of operations

Choose a tool group (see Subsection 2.3.1 "Changing tool data")
Press the "Create new" softkey,

Enter the ID number of the new tool and press "Input".

You can now accept or overwrite the values suggested in the input window.

The maximum tool length you can enter is 999 mm.
Each value you enter must be confirmed separately with "Input".
It is not possible to confirm several words at once!

Press the "OK" sofkey and

press "Yes" in response to the prompt to
save the data for your new tool,
which immediately appears in the tool group.

Press the "End" sofkey and
quit the "Create new" menu.

### 2.3.3 Deleting a tool

#### Sequence of operations

After you have selected the tool group from which the tool is to be deleted press the "Delete" sofkey

and select the tool to be deleted with the cursor keys.

Now confirm with the "OK" sofkey
and the following query appears "Tool ... to be deleted?".

If you now press the "Yes" sofkey, **this tool is irrevocably deleted from the memory!**

If you press the "End" sofkey, you return to the "Tool groups" menu.
### 2.3.4 Find tool

#### Sequence of operations

You have pressed the "Tools" softkey in the initial menu of AutoTurn and are now in the "Tool groups" menu.

Pressing the "Find ID" softkey initiates a search for a certain tool.

Enter the known part of the ID number (Example: 1234)

and use an "asterisk" in front or behind (e.g.: *1234, 1234* or *1234*).

Confirm with the "Input key" and all the tools are listed that contain the known part of the ID number.

#### Note

If you enter the "asterisk" character on its own you obtain a list of all the tools that are stored!

With the softkeys “Change/copy” that are now accessible or "Delete"

you can further process the tool found (see Subsections 2.3.1/2.3.3).

Selecting the "Back" softkey

followed by "End"

brings you back to the initial "Tool groups" menu.
2.3 Tool data dialog

MMC 103  PCU 50
## Workholders

3.1 Workholder groups .......................................................................................................................... 3-54
3.1.1 Overview ..................................................................................................................................... 3-54
3.1.2 Operation ................................................................................................................................... 3-55

3.2 Workholder master data .................................................................................................................... 3-56
3.2.1 General data ............................................................................................................................. 3-56
3.2.2 Outside and inside jaws .............................................................................................................. 3-56
3.2.3 Collets/Face drivers .................................................................................................................. 3-56
3.2.4 Tailstock points ........................................................................................................................... 3-57
3.2.5 Workholder dialog ..................................................................................................................... 3-58
3.1 Workholder groups

3.1.1 Overview

The workholders used are split up into several groups:

<table>
<thead>
<tr>
<th>Number</th>
<th>Workholder group</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Outside jaws soft without fixed stop</td>
</tr>
<tr>
<td>2</td>
<td>Outside jaws soft with fixed stop</td>
</tr>
<tr>
<td>3</td>
<td>Outside jaws hard without fixed stop</td>
</tr>
<tr>
<td>4</td>
<td>Outside jaws hard with fixed stop</td>
</tr>
<tr>
<td>5</td>
<td>Inside jaws soft</td>
</tr>
<tr>
<td>6</td>
<td>Inside jaws hard</td>
</tr>
<tr>
<td>7</td>
<td>Collets</td>
</tr>
<tr>
<td>8</td>
<td>Face drivers</td>
</tr>
<tr>
<td>9</td>
<td>Tailstock point type A</td>
</tr>
<tr>
<td>A</td>
<td>Tailstock point type B</td>
</tr>
<tr>
<td>B</td>
<td>Tailstock point type C</td>
</tr>
<tr>
<td>C</td>
<td>Tailstock point type D</td>
</tr>
</tbody>
</table>

AutoTurn assigns the workholders to the groups stated above. You may define your own workholders and assign these to the toolholder groups as described in Subsection 3.2.5.
3.1.2 Operation

**Sequence of operations**

Press the "Workholder" softkey in the main menu of AutoTurn.

The list of workholder groups appears.

Press the "Change/copy" softkey,

select the ID number

and confirm with "OK".

Open the workholder window.
3.2 Workholder master data

3.2.1 General data

The following general workholder data apply to all workholders and are not repeated in the specific data descriptions:

- ID number: max. 20 characters (enter for "Create new")
- Designation: max. 30 characters
- Comment: Comment

The ID number refers to the workholder. You will need this ID number in the workholder dialog in order to select your workholder.

3.2.2 Outside and inside jaws

For outside and inside jaw chucks in workholder groups

Outside jaws soft
  with/without fixed stop
Outside jaws hard
  with/without fixed stop
Inside jaws soft/hard

please enter these values:
- Length dimension L:
- Height dimension Q:
- Depth dimension T  (for "with fixed stop" only):

3.2.3 Collets/Face drivers

You cannot define values for collets, since collets that match the blank diameter are used. You are prompted to enter the dimensions in the "Programming" menu. Face driver dimensions differ only with respect to the diameter D.
3.2.4 Tailstock points

Specify the following dimensions for your tailstock point types A, B, C or D:

Tailstock point type A
- (Small) diameter D1:
- (Large) diameter D2:
- Length dimension L1:
- Angle W1:

Tailstock point type B
- (Small) diameter D1:
- (Large) diameter D2:
- Length dimension L1:

Tailstock point type C
- Diameter D1:

Tailstock point type D
- Diameter D1:
- Diameter D2:
- Angle W1:
3.2.5 Workholder dialog

The operator input sequence for the workholder dialog is identical with that of the tool data dialog!

Please observe the operator input sequence as described in the Section 2.3, replacing the word "tool" with the word "workholder". The following functions are available:

- Create new
- Change and copy
- Delete.
Cutting Values

4.1 Machining method
   4.1.1 Overview
   4.1.2 Operation
   4.1.3 Description

4.2 Cutting values dialog
   4.2.1 Changing material data
   4.2.2 Copy material data
   4.2.3 Delete material
4.1 Machining method

4.1.1 Overview

Materials form the basis of all cutting values. Each material used can be machined by 12 different methods:

<table>
<thead>
<tr>
<th>Number</th>
<th>Machining method</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Turning - ROUGHING</td>
</tr>
<tr>
<td>2</td>
<td>Turning - FINISHING</td>
</tr>
<tr>
<td>3</td>
<td>Grooving - ROUGHING</td>
</tr>
<tr>
<td>4</td>
<td>Grooving - FINISHING</td>
</tr>
<tr>
<td>5</td>
<td>Threads</td>
</tr>
<tr>
<td>6</td>
<td>Drilling with twist drills</td>
</tr>
<tr>
<td>7</td>
<td>Drilling with insert drills</td>
</tr>
<tr>
<td>8</td>
<td>Reaming</td>
</tr>
<tr>
<td>9</td>
<td>Sinking</td>
</tr>
<tr>
<td>A</td>
<td>Tapping</td>
</tr>
<tr>
<td>B</td>
<td>End milling - ROUGHING</td>
</tr>
<tr>
<td>C</td>
<td>End milling - FINISHING</td>
</tr>
</tbody>
</table>

You can assign up to 8 different cutter materials and the optimum cutting speeds to each of these machining methods. Cutting values therefore depend on:

- Material,
- Machining method and
- Cutter material.

This gives you a finely structured method of optimizing the cutting values for your application.
4.1.2 Operation

Sequence of operations

Press the "Cutting values" softkey in the initial menu of AutoTurn and a material selection is displayed on the screen.

Use the cursor keys to select the material and then press the "Change" softkey.

In this way, you can open the window with the first machining method "Turning - ROUGHING".

With the softkeys "Page +1" or "Page –1" you can page through the 11-page cutting value table.

If you press the "Jump" softkey the window for selecting the machining method is displayed immediately.

Use the cursor keys here and press "OK" to reach the required machining method faster.

Changes to the cutting values are always irreversible! You must therefore copy the original material and change the copy! The default cutting values stored in the AutoTurn Graphic Programming System are tested and optimized data. You can change them but you must have sound knowledge of the technology!
4.1.3 Description

Turning - ROUGHING
Here you can enter up to 8 different cutter materials. Cutting speeds must be specified for coarse and fine, external and internal machining. You can also specify the maximum depth of cut and the roughing feedrate for internal and external machining.

Turning - FINISHING
Depending on the surface finish required, you can set the finishing feedrates as a function of the cutting radius.

Grooving - ROUGHING
For "Grooving - ROUGHING" up to 8 different cutting materials are possible that determine the cutting speed for coarse and fine external and internal grooving. Depending on the width \( T \) of the grooving insert (see Section 2.2.5 "Grooving tools") the roughing feedrates will vary.

Grooving - FINISHING
For "Grooving - FINISHING" the finishing feedrate will be different depending on the surface finish required and the width of the grooving tool.

Threads
Here you can enter the cutting speeds for external and internal threads for up to 8 cutting materials. In the lower half of the table, the number of chasing passes has already been optimized and predefined as a function of the pitch.
### Drilling with twist drills
You can make up to 5 entries in the column "Cutter materials". The feedrates will differ for a certain cutting speed depending on the tool diameter D (see Section 2.2.7 "Drilling tools").

### Drilling with insert drills
Here, you specify the same data as for the previous machining process for large drilling diameters.

### Reaming
### Sinking
You can enter up to 5 different cutter materials for these methods. The feedrates (for the same cutting speed) are again a function of the tool diameter D.

### Tapping
For "Tapping", you can enter up to 5 cutter materials per workpiece material, each with a cutting speed assigned to it.

### End milling - ROUGHING
### End milling - FINISHING
You can make up to 5 entries in the column "Cutter materials". Depending on the milling diameter D1 (see Subsection 2.2.8 "Milling tools"), different feedrates F are obtained for constant cutting rate V.
4.2 Cutting values dialog

4.2.1 Changing material data

These sections explain how to handle the cutting values described above:

Sequence of operations

Having left the initial menu of AutoTurn you are in the "Cutting values" menu and have a selection of materials in front of you.

Use the cursor keys to select your material

and press the "Change" softkey.

The first of 11 input pages for 12 machining methods is displayed on the screen ("Sinking" and "Tapping" are grouped together on Page 9/11!).

Move the cursor to the required input point,

write your value

and confirm the entry with "Input".

Press the "OK" softkey and the query: "Save altered values?" is displayed

that you can answer by pressing the appropriate softkey. This returns you to the initial window for the cutting values.
Note

The control informs you about incorrect inputs in a message box which is displayed in the middle of the screen.

You can acknowledge these messages with the "Clear error" softkey. Check the values you have entered!

You can delete cutter materials entered unintentionally by pressing the "Space" key in the field for that material.

Incorrect numeric entries can be overwritten only with the value "zero".

Sequence of operations

You can page back and forward in the cutting value tables with these softkeys,

or with the "Jump" softkey

and then make a selection with the cursor keys

and then press "OK" to jump straight to the required machining method.
4.2.2 Copy material data

**Function**

Here, you can **copy the entire cutting value file into the new material** under the new material designation and adapt each machining method to your new material by changing factors.

**Sequence of operations**

First make a selection from the "Cutting values" selection list.

Select the (similar) material,

to whose cutting values the data of the new material must be adapted (with a few changes) and press the "Copy" softkey.

Enter the name of the new material in the window:

"Enter new material specification: ..."

and confirm with "Input".

A table appears on the screen containing the factor 1.000 with reference to the cutting velocity \( V \) and the feedrate \( F \) for each machining method.
Sequence of operations

Use the cursor keys to select the special factors for your new material,

overwrite the values

and confirm your entry with "Input".

Confirm with the "OK" softkey and when the query "Create new material: ...?" is displayed,

answer it with the appropriate softkey.
4.2.3 Delete material

Sequence of operations

You are in the initial menu of AutoTurn and press the "Cutting values" softkey.

Select the material to be deleted

and press the "Delete" softkey.

The following query is now displayed:
"Delete new material: ...?"
If you answer it by pressing the "Yes" softkey, the stored material will be deleted irrevocably from your cutting value file!
Functions

5.1 Overview .................................................................................................................... 5-70
5.1.1 Setup functions ........................................................................................................ 5-70
5.1.2 Programming functions ....................................................................................... 5-70

5.2 Programming ................................................................................................................. 5-71
5.2.1 Description ............................................................................................................. 5-71
5.2.2 Operation ................................................................................................................. 5-71

5.3 Program management .................................................................................................. 5-73
5.3.1 Description ............................................................................................................. 5-73
5.3.2 Operation ................................................................................................................. 5-74

5.4 Blank ............................................................................................................................ 5-78
5.4.1 Blank stocks and dimensions .................................................................................. 5-78
5.4.2 Free blank stocks ..................................................................................................... 5-79
5.4.3 Material selection ..................................................................................................... 5-79

5.5 Finished part ................................................................................................................ 5-80
5.5.1 Dimensions / Positioning ......................................................................................... 5-80
5.5.2 Display ..................................................................................................................... 5-83
5.5.3 Outside and inside contour ....................................................................................... 5-85
5.5.4 End face and peripheral surface ............................................................................. 5-86
5.5.5 Cylinder/Taper/Circle ............................................................................................. 5-91
5.5.6 Contour definition ..................................................................................................... 5-94
5.5.7 Form element ........................................................................................................... 5-97
5.5.8 Chamfer/Rounding/Roughness height/Fit class ....................................................... 5-98
5.5.9 Thread ...................................................................................................................... 5-100
5.5.10 Correction .............................................................................................................. 5-103

5.6 Mirror workpiece/machining position ............................................................................. 5-108

5.7 Cut-off part ................................................................................................................... 5-108

5.8 Workholders ................................................................................................................ 5-110
5.8.1 Machining side ....................................................................................................... 5-110
5.8.2 Select workholder ................................................................................................... 5-111
5.8.3 Tailstock .................................................................................................................. 5-114

5.9 Defining machining limits ............................................................................................. 5-115
5.1 Overview

5.1.1 Setup functions

The contents of the following functions represent technological system settings for your machine tool and should not be altered by the machine operator!

The machine setter alone is responsible for making any changes to these data!

Setup functions include the following:

- Tools
- Tooling (toolholder)
- Workholders
- Cutting values with material, cutter material and feed files

5.1.2 Programming functions

The following functions are available to the machine operator:

- Program handling (management)
- Blank part
- Finished part
- Chucking
- Automatic program generation
- Interactive programming
- Graphic simulation
- Machining plan editor
- Setup
- NC program generation and load/start NC program
5.2 Programming

5.2.1 Description

**Function**

"Programming" enables you to create a new file in AutoTurn. This serves as the basis for creating a new part program for a workpiece.

**New program**

A new program is created with a freely selectable name. The name must not be longer than 17 characters.

The "Programming" softkey allows you to create a part program. From this point on, AutoTurn guides you through the full process, from selecting the blank right up to the setup of the machine.

5.2.2 Operation

**Sequence of operations**

After you have called the AutoTurn Graphic Programming System you will find yourself in the main menu (see Section 1.4.1).

When you select the "Programming" softkey, a window is opened in which you enter a **program name** at the position of the flashing cursor via the alphanumeric keyboard.

Enter, for example, PART5678 and confirm the input by means of the "Input" key.
The program name may have a maximum of 17 characters and must not contain any blanks or special characters.

If you enter a program name which already exists, then the following query appears:

Program name: ...
WOP program exists already!
Edit WOP program?

If you answer the inquiry by selecting the "Yes" softkey, then you enter the program processing mode.

If you select the "No" softkey, then a new program name is requested.

The "Recall" key has the same meaning as the "No" key.
5.3 Program management

5.3.1 Description

Function

The "Program handling" function allows you to modify, copy, delete and edit all the part programs stored in the AutoTurn program memory.

Program handling

The workpiece to be machined - from the blank part definition to the program generation - is made available to you for complete revision and is then stored with all changes under the same program name.

Copy program

You can use this function to store a part program that you have already generated under a new program name with no changes to the program contents.

Use this function to create back-up copies of original workpieces.

Never copy AutoTurn workpieces outside AutoTurn (e.g. under Services), you will no longer be able to open them with AutoTurn.

Delete program

- This function deletes all the part programs relating to this workpiece from the memory.
- CAUTION! Please check carefully whether you really want to delete a part program before activating this function!
- The program to be deleted should not be selected for processing!

Edit program

This function allows you to edit all the data generated by the AutoTurn Graphic Programming System:

- Part programs
- TOA data (optional)
- Time calculations (optional)
- Setup sheet (optional)

In this case, "Optional" means "As defined in the machine adaptation"
5.3.2 Operation

Program selection

Sequence of operations

You are in the AutoTurn main menu and select the "Program handling" softkey.

A window is then opened containing all the part programs currently stored in the program memory with their date and time of creation.

When you now position the cursor bar on the desired program using the page and cursor keys, the program is highlighted in black in the first line as the program released for machining.

Example:

<table>
<thead>
<tr>
<th>Program</th>
<th>Date/Time</th>
</tr>
</thead>
<tbody>
<tr>
<td>PART5678</td>
<td>8/08/95 5:34 AM</td>
</tr>
<tr>
<td>PART0815</td>
<td>9/08/95 6:31 AM</td>
</tr>
<tr>
<td>TEST</td>
<td>7/18/95 10:06 PM</td>
</tr>
<tr>
<td>PART1</td>
<td>7/25/95 11:50 PM</td>
</tr>
</tbody>
</table>

You can also enter the program name directly or select the program by combining direct inputs with paging.

For example, if the desired program name starts with a "T", press the "T" key and the cursor bar will then be positioned on the first program which starts with a "T".

You can either repeat this process or continue paging.
Program handling

Sequence of operations

When you select the "Machining" softkey, the finished part representation of the selected workpiece is displayed on the screen. From this point in the menu, you have the full range of modification options available so that you can, for example,

- alter the blank data of the workpiece by selecting the "Blank" softkey or
- change the finished part contour by selecting the "Fin. part" softkey.

When you have finished modifying the workpiece data, select the "End" softkey to terminate the operation,

- store the modified workpiece by selecting the "Yes" softkey and store the NC data as well by selecting the "Yes" softkey again.

If you look under "Program handling" again, you will see that the modified part program is listed with the date/time of modification.

Copy program

Selection of the "Copy" softkey opens the following window:

Program name: ... Copy to
Program name: ...

Using the alphanumeric keyboard, type the new program name at the flashing cursor position under which the selected workpiece must be stored with no changes.

The "Input" key initiates the store operation and you will then see your new part program appear in the displayed part program list.
Delete program

Sequence of operations

After you have chosen the part program to be deleted as described in Subsection 5.2.2, select the "Delete" softkey to confirm the operation.

The following inquiry is output in the displayed screenform:
  Program name: PART5678
  Delete WOP/NC program?

If you now acknowledge this inquiry with the "Yes" softkey, the selected workpiece program will be irrevocably removed from the part program memory.

Make sure that the workpiece program to be deleted is not currently being processed!

Edit program

Sequence of operations

After you have selected the workpiece as described in Section 5.2.2,

select the "Edit" softkey and you will receive a list of all the files belonging to the workpiece.
### Sequence of operations

<table>
<thead>
<tr>
<th>Name</th>
<th>Meaning</th>
<th>Write protection</th>
</tr>
</thead>
<tbody>
<tr>
<td>name.041</td>
<td>AutoTurn program</td>
<td>no</td>
</tr>
<tr>
<td>mpf 1001.mpf</td>
<td>NC program 1st side</td>
<td>yes</td>
</tr>
<tr>
<td>mpf 1002.mpf</td>
<td>NC program 2nd side</td>
<td>yes</td>
</tr>
<tr>
<td>toa11.spf</td>
<td>Tool offset data 1st side</td>
<td>yes</td>
</tr>
<tr>
<td>toa22.spf</td>
<td>Tool offset data 2nd side</td>
<td>yes</td>
</tr>
<tr>
<td>tools 1.dat</td>
<td>Setting sheet 1st side</td>
<td>yes</td>
</tr>
<tr>
<td>tools 2.dat</td>
<td>Setting sheet 2nd side</td>
<td>yes</td>
</tr>
<tr>
<td>time.dat</td>
<td>Time calculation</td>
<td>yes</td>
</tr>
</tbody>
</table>

Changes in these files are not accepted by AutoTurn, i.e. changed part programs may be overwritten the next time they are processed with AutoTurn!

To prevent existing program modification being lost when generating again, it is recommended that you rename the part programs "mpf 1001.mpf" and "mpf 1002.mpf".

You select another file as described in Section 5.2.2.

When you select the "OK" softkey, the file you have chosen is edited in the SINUMERIK 840D-ASCII editor. Please refer to the general Operator's Guide for the SINUMERIK 840D/810D for instructions on how to operate the editor.
5.4 Blank

5.4.1 Blank stocks and dimensions

Sequence of operations

In its present version, the AutoTurn Graphic Programming System can offer the user 4 types of blank stock:

Cylinder  Hollow cylinder  Square  Hexagon

These 4 types are defined by the following inputs:

Cylinder:  Blank diameter  Blank length
Hollow cylinder:  as for cylinder, plus inner diameter
Square:  Side length  Blank length
Hexagon:  Width across flats  Blank length

The maximum permissible blank length is 9999.999.

You must separately confirm every value you enter by selecting the "Input" key or else your new data will not be transferred to the system.

You can use the cursor keys to move from one input value to the next and to alter values which are already stored.

Store all the blank dimensions by selecting the "OK" softkey.

The square blank is displayed along axis D in the diagonal diameter:

Side length x factor 1.4142

and the hexagon blank along axis D with diagonal diameter:

Width across flats x factor 1.1547

will then be displayed on the screen.
5.4.2 Free blank stocks

Free blank forms are to be found under "Programming / Blank ...":

<table>
<thead>
<tr>
<th>Cylinder</th>
<th>Hollow cylinder</th>
<th>Square</th>
<th>Hexagon</th>
<th>Free definition</th>
<th>Allowance finished part</th>
</tr>
</thead>
</table>

All the options for creating a finished part geometry are available for this free blank form.

- Cylinder/Taper/Circle/Contour/Form element/Correct
- (s. Section 5.5 Finished part)

- Max. finished part diameter
- Equidistant allowance
- Finished part length F

You will find a detailed example about freely definable blanks in Chapter 7 "Examples of Program Sequences" (s. Section 7.2 Programming example: any blank forms).

5.4.3 Material selection

Sequence of operations

The AutoTurn Graphic Programming System contains a selection list of commonly used materials which is displayed on the screen either after the blank dimensions have been confirmed with "OK" or when the "Cutting values" softkey (in the main menu) is selected.

Instructions on how to handle materials (change, copy, delete) are given for your machine setter in Section 4.2.
5.5 Finished part

5.5.1 Dimensions / Positioning

Sequence of operations

After you have specified the blank data, you will be requested to enter the finished part length.

If you enter a value which is greater than the blank length by mistake, the following alarm:

"Finished part length is greater than blank length."

is output to indicate the mistake.

This error display must be reset before you proceed to the next programming step.

Once you have entered the finished part dimensions correctly, you can position your finished part within the blank part:

AutoTurn offers you the following options:

Centered  Left justified  To size
Centered positioning

Sequence of operations

When you select the "Centered" softkey, the finished part is positioned inside the blank in such a way that the differential dimension corresponding to "Blank length minus finished part length" appears halved into equal parts to the right and left of the finished part, i.e. it is centered exactly.

Programming example

Blank length: 72 mm
Finished part length: 70 mm
Centered position: Allowance on left and right of finished part = 1 mm

Left-justified positioning

Sequence of operations

When you select the "Left justified" softkey, you position the finished part inside the blank such that the left-side end faces of both parts are flush with one another. The dimension to the right of the finished part corresponds in full to "Blank length minus finished part length".

Programming example

Blank length: 72 mm
Finished part length: 70 mm
Left justified position: Allowance of 2 mm to right
Defining your own dimensions

Sequence of operations

If you want to define the allowance of the right machining side yourself, press the "Acc. to scale" softkey,

enter allowance A in the window which opens

and confirm by selecting "Input".

If the allowance you set is greater than the dimension corresponding to "Blank length minus finished part length", then the error message "Distance too large" appears. You must acknowledge this message by selecting "Clear error". You can then enter a new value.

If you specify an allowance A which is smaller than the dimension "Blank length minus finished part length", then the difference between the two values is entered as the allowance to the left of the finished part.

Programming example

Blank length: 72 mm
Finished part length: 70 mm
To size: A = 1.5 mm
Allowance on right: 1.5 mm
Allowance on left: 0.5 mm
5.5.2 Display

Function

You can change the diagrams of your workpiece when you see the following softkeys displayed in the AutoTurn Graphic Programming System in the vertical softkey bar on the screen:

- "Window"
- "Zoom +"
- "Zoom -"
- "Move"
- "Zero point"
- "Original":

You cannot use this function if other programming functions are currently active (for example, simulation in progress, input of a contour element).

Sequence of operations

Your workpiece is returned to its original position on the screen when you select the "Original" softkey.

Select the "Window" softkey,

use the cursor keys to position the cursor cross, which appears at the zero point, to the first corner of the desired section and confirm the selection with "OK".

Then use the cursor keys to pull the border of the window section to the diagonally opposite corner, which must again be confirmed with "OK".

The zoomed section can be seen on the screen.
Sequence of operations

Every time you select these softkeys, the diagram increased or decreased by approx. 20 per cent.

By selecting the "Move" softkey
and then pressing the cursor keys, you can move the workpiece around the screen
and then confirm its position with the "OK" softkey.

When you select the "Zero point" softkey, the zero point is moved to the opposite end point of the finished part on the longitudinal axis so that the sign of the positions along the longitudinal axis is reversed (from minus to plus).
Set the zero point according to the workpiece drawing. If you need to make calculations, you can do so directly in the input screenform.

The geometry definition is not oriented according to the position of the zero point, but always from right to left!

You can return the zero point to its original position by selecting this softkey again.
5.5.3 Outside and inside contour

To assist you in programming the external contour and internal contour of your workpiece, you are provided with the following:

- Geometry elements: Cylinder, taper, circle
- Contour definitions: Straight line, circle
- Form elements: Undercut, radial groove, face recess cylinder with undercut, concatenated O-ring profiles

It does not matter in which order you program the contours, i.e. you can start with the external or internal contour. The following points should be observed:

- The contour you enter must be closed, i.e. all elements must be interlinked!
- It must always be programmed right through to the end of the finished part.

The AutoTurn Graphic Programming System automatically defines the optimum sequence of operations when it generates the NC program!
5.5.4 End face and peripheral surface

The following contours can be programmed:

<table>
<thead>
<tr>
<th>Contour</th>
<th>On end face</th>
<th>On peripheral surface</th>
</tr>
</thead>
<tbody>
<tr>
<td>Hole</td>
<td>Automatic</td>
<td>Automatic</td>
</tr>
<tr>
<td>Thread hole</td>
<td>Automatic</td>
<td>Automatic</td>
</tr>
<tr>
<td>Free contour - Pocket</td>
<td>Automatic</td>
<td>Single machining step</td>
</tr>
<tr>
<td>Free contour - Island</td>
<td>Automatic</td>
<td></td>
</tr>
<tr>
<td>Free contour - Cylinder path</td>
<td>Automatic</td>
<td></td>
</tr>
<tr>
<td>Form element - Slot longitudinal</td>
<td>Automatic</td>
<td></td>
</tr>
<tr>
<td>Form element - Cylinder path</td>
<td>Automatic</td>
<td></td>
</tr>
<tr>
<td>Form element - Slot linear</td>
<td>Automatic</td>
<td></td>
</tr>
<tr>
<td>Form element - Slot semi-circular</td>
<td>Automatic</td>
<td></td>
</tr>
<tr>
<td>Form element - Sector</td>
<td>Automatic</td>
<td></td>
</tr>
<tr>
<td>Form element - Surface</td>
<td>Automatic</td>
<td></td>
</tr>
<tr>
<td>Form element - Pocket milling 1</td>
<td>Automatic</td>
<td></td>
</tr>
</tbody>
</table>

Cylindrical paths on the peripheral surface must not be closed. Program a path that consists of two or more single contours.

Contour 1

For example:

Contour 2

AutoTurn also offers the following new contour planes of finished part geometry:

<table>
<thead>
<tr>
<th>End face left</th>
<th>End face right</th>
</tr>
</thead>
<tbody>
<tr>
<td>Peripheral</td>
<td>Peripheral</td>
</tr>
</tbody>
</table>

Peripheral

Peripheral surface
On each of these 4 new contour planes, you can program:

Holes  Thread holes  Free contours  Form elements.

It does not matter in which order you program the contour.

The AutoTurn Graphic Programming System automatically defines the optimum sequence of operations when it generates the NC program!

The Y axis that appears on the screen when you program with AutoTurn is not a real machine axis!
- In end face machining this axis is simulated in the generated NC program by TRANSMIT.
- For peripheral surface machining, this axis corresponds to the degrees of the C axis position (mapped in the NC program generated in TRACYL).

End face (radius programming)  Peripheral surface (processing)

End face programming should be started in the positive quadrant so that TRANSMIT generates a positive X axis movement.
Drilled holes / Tapped holes

Sequence of operations

When you press the softkeys you call a menu in which you can define drilled holes/tapped holes on the peripheral surface or end faces.

You can make your selection from the following:
- Even distribution of holes
- Uneven distribution
- Entry of coordinates of any holes

You define the circle of holes you want with the softkeys "Selection 1" to "Selection 3".

A dimensioned representation of the selected circle of holes is displayed. Enter the required parameters (pitch diameter, pitch circle center, number of holes and angle between holes, coordinates of holes) through the keyboard and confirm your entries with "Input".

Complete your entries with "OK".

You gain access to the dimension drawing of the hole or thread hole. Here you define the diameter of the holes, their length and thread length and chamfer.

Complete this procedure with "OK", too.
Free contours/Form elements

Sequence of operations

You can program pockets and islands on the end faces and pocket and cylinder machining on the peripheral surface.

You can choose between any form (via "Free contour" softkey) and predefined elements (via "Form element" softkey).

Free contour

If you choose a free contour,

the softkeys "Pockets" and "Islands" for end face machining.

or

the softkeys "Pockets" and "Machine cylinder" for peripheral machining are available.

When you press one of the softkeys, a dimensioned representation of the workpiece is displayed. Enter the depth T and the reference length L here. This defines the position for machining.

After pressing "OK", enter the desired contour. You can work with the "Construction geometry" function or enter the coordinates of the contour points directly.

The options described in Section 5.5.6 are available for entering the contour points.

In the sample program: Extended functions in Section 7.2, a detailed explanation is given to assist you in your work with construction geometries.
Form element

Sequence of operations

If you choose a "Form element", the softkeys "Pockets" and "Islands"

or for peripheral surface machining only the options "Selection 1" and "Selection 2" for longitudinal groove and inclined groove are available.

When you press the softkeys "Pockets" or "Islands", you call a menu in which you can define pockets or islands, which you can call with the "Selection ..." softkeys.

A dimension drawing is displayed for each selection. Enter your values for the parameters displayed and confirm your input by selecting "Input".

When you have completed your entry, press "OK". The programmed form element is now incorporated in the finished part geometry.

You will find extensive programming examples for machining end faces and the peripheral surface in Section 7.2, Sample program: Extended functions.
5.5.5 Cylinder/Taper/Circle

These contour elements must always be defined fully. Every time an input is requested, you must enter a value or else the error message: "Input required" appears!

Sequence of operations

You want to program a cylinder.

Before you enter the cylinder diameter, you can select a cylindrical thread (Metric, Whitworth or Unified).

Please refer to Section 5.5.9 for a description of the threads!

Sequence of operations

A cylinder is defined with:
- Diameter EP:
- End length:

The center line (axis of symmetry) is always positioned along longitudinal axis L.

The end length can be transferred to the system by means of the "Distance to go" softkey.

After you have entered the end length, you can add the following to the cylinder:
- Roundings left / right and
- Chamfers left / right

as described above for the cylinder.

If it is not possible to add roundings or chamfers to the right, then the appropriate softkeys will not be displayed.

Since the following geometry is not known at this point, roundings and chamfers on the left-hand input side are initially only represented by a symbol.

You can continue at this point with "Roughness height" (Section 5.5.8) or enter a "Fit class".
Sequence of operations

You want to program an arc.

A taper is defined by:
- Start diameter: (alternatively: Angle)
- End diameter: (alternatively: Angle)
- End length:

If you do not wish to specify the initial and end diameters, confirm the input field with "Input". Alternatively, you can program the angle of the taper:

The counting direction is positive in the counterclockwise direction starting at the horizontal zero point.

The end length can be transferred to the system with "Distance to go".

After you have entered the end length, you can add the following to the taper:
- Roundings left / right
- Chamfers left / right
as described above for the cylinder.

You can also input a roughness height for the taper (Section 5.5.8).
Sequence of operations

You want to program an arc.

A circle is defined by its:
- Clockwise/counterclockwise direction of rotation:
- Start diameter:
- End diameter:
- End length:
- Radius:

You can also add a rounding or the roughness height here.
5.5.6 Contour definition

The essential characteristic of the contour definition is that the end point of the preceding contour is always the start point of the next contour. You therefore only ever specify the end points of new contour elements.

Contour definitions can also contain up to 4 undefined elements (see Chapter 10 Undefined Contour Elements).

Contour definitions in the AutoTurn Graphic Programming System can comprise straight lines and circles.

Sequence of operations

You want to program a straight line and can choose between four different lines:

A straight line in any direction

is defined by two of the three following values:

- Diameter:
- End length:
- Angle:

Only one of the following values

- Diameter:
- Line length:

needs to be specified for the lines depicted on the left, while this straight line requires only the

- End length:
- or the
- Line length:
Sequence of operations

You can program a rounding, chamfer or roughness height if you wish to.

The contour definitions are transferred to the system on selection of "OK".

Sequence of operations

An arc as a contour element can be programmed either in a clockwise or counterclockwise direction, and either without a tangential link (abrupt contour transition) or with a tangential link (smooth contour transition).

A circle without a tangential transition can be defined by:
- the Center position coordinates Diameter CP:
  - Length CP:
- the End position coordinates Diameter EP:
  - Length EP:
  - Radius:
  - End angle:

Please also note that diameter programming is used! All values along axis D are double radii!

You can define a circle with tangential link by entering

Radius:
Diameter EP:
Length EP:
End angle:
Sequence of operations

When you have a choice of programming options, select either the next end point in the direction of rotation with the "White" softkey or select the end point which is 180 degrees distant in the direction of rotation with the "Blue" softkey.

You can again add a rounding or roughness height

and finally confirm the inputs with "OK".
5.5.7 Form element

The following form elements are available in the AutoTurn Graphic Programming System:

- Undercuts to
  - DIN 509E
  - DIN 509F (with relief cut)
  - DIN 76C (for thread)

- Radial undercuts
  - Circular groove to right
  - Circular groove to left
  - Groove with beveled edges
  - Groove with beveled edges and allowance

- Face grooving:
  - Right-angled cylindrical face groove

- Other:
  - Cylinder plus undercut to DIN 509E
  - Cylinder plus undercut to DIN 509F
  - Concatenated O-ring profiles (external contour only)

Insertion of these elements into an existing contour is described under the function "Correction" (see Section 5.5.10)!

Sequence of operations

You want to program an "Undercut" to DIN 509E.

Press "Selection 1" softkey to call up a graphic of an undercut of this type onto the screen.

You enter:
- Width F:
- Depth T:
- Machining allowance Z:
- Angle W:
- Radius R:

and transfer the values to the system by selecting "OK".
5.5.8 Chamfer/Rounding/Roughness height/Fit class

Chamfers and roundings represent transition elements between geometric contours.

Sequence of operations

A chamfer can be inserted on the left or right (for example, on a cylinder) of the contour to be programmed.

Chamfers are defined by

Chamfer length (left/right): Length x chamfer angle
Chamfer angles of 45 degrees are the default setting.
Chamfer angles must be greater than zero and less than 90 degrees!
The counting direction of the chamfer angle is defined positively from the longitudinal axis L to the next contour.

The chamfer you have entered is displayed only as part of the next programmed element.

If the chamfer or chamfer angle you enter is too large, making it impossible to reach the next contour, the error message: "Tangent for chamfer is too small" is displayed and you will have to program the geometry element again!
Sequence of operations

A rounding can likewise be added to the left or right of the contour you have just programmed. In this case, the transition to the next contour must always be tangential and the aperture angle of the rounding 90°.

Roundings are defined by the
Rounding radius (left/right):

You will not see the rounding you have entered until you program the next contour.
Example: ↓ R4

Sequence of operations

You can define the maximum roughness heights for cylinder, taper and circle contours.
Example: Input 5 µm

Sequence of operations

You can only enter fits for cylinders.
You must type the letter "H" in front of the number!
Example: Input h10
5.5.9 Thread

The AutoTurn Graphic Programming System supports the programming of **cylindrical threads**.

**Sequence of operations**

Starting with external or internal machining, you can select one of the following threads after selecting "Cylinder" softkey:

<table>
<thead>
<tr>
<th>Metric thread</th>
<th>Whitworth thread</th>
<th>UN thread</th>
<th>NPT-thread</th>
</tr>
</thead>
<tbody>
<tr>
<td>M4 x 0.7</td>
<td>M14 x 2.0</td>
<td>M16 x 2.0</td>
<td>M18 x 2.5</td>
</tr>
<tr>
<td>M5 x 0.8</td>
<td>M16 x 2.0</td>
<td>M18 x 2.5</td>
<td>M20 x 2.5</td>
</tr>
<tr>
<td>M6 x 1.0</td>
<td>M18 x 2.5</td>
<td>M22 x 2.5</td>
<td>M24 x 3.0</td>
</tr>
<tr>
<td>M8 x 1.25</td>
<td>M20 x 2.5</td>
<td>M22 x 2.5</td>
<td>M24 x 3.0</td>
</tr>
<tr>
<td>M10 x 1.5</td>
<td>M22 x 2.5</td>
<td></td>
<td></td>
</tr>
<tr>
<td>M12 x 1.75</td>
<td>M24 x 3.0</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Press the "Metric thread" softkey and you will receive a list of metric threads when you press the "Help" softkey.

Select one of the proposed threads by means of the cursor keys and confirm your selection with the "OK" key.

You can terminate the process without storing your inputs by selecting "Recall".

You can define the thread lead at the flashing cursor position and then store the entire thread by selecting "Input".
Sequence of operations

If you wish to program a thread which is not included in the selection list, then enter the thread data in the free input:

Nominal diameter x pitch and
Nominal diameter x pitch and
End length:

with the link elements:

Chamfers right x chamfer angle
Chamfer left x chamfer angle or
thread undercut left.

An appropriate thread lead is suggested for all thread types up to certain rated diameters. You can however modify the suggested lead data!

Thread undercuts are the link contours provided for metric and inch threads.
AutoTurn will calculate the optimum thread undercut from the thread data you specify.

Sequence of operations

You can choose from four thread types when you press the "Whitworth thread" softkey:

Standard Pipe (G) Pipe (R) BSF

You can select standard threads in the "Help" window:

1/4 x 20 1/2 x 12 1 x 8
5/16 x 18 5/8 x 11 1 1/8 x 7
3/8 x 16 3/4 x 10 1 1/4 x 7
7/16 x 14 7/8 x 9 1 3/8 x 6

Pipe (G) inch threads (for example: 1/4 x 19 or 5/8 x 14) and fine threads BSF (for example: 7/32 x 28 or 9/32 x 26) are also made available for you to select under "Help".
Three groups are available under UN threads:

**UNC** | **UNF** | **UNEF**,

which are offered as the following types in the "Help" screen form:

- **UNC** 1/4 x 20
- **UNF** 1/4 x 28
- **UNEF** 1/4 x 32

You can program **multiple threads** by selecting the "Multiple thread" softkey:

You then enter the number of threads and confirm your input by selecting "Input".

Metric and inch threads can also be programmed as a **left-hand thread**. When you select this softkey, a right-hand thread is turned into a left-hand thread.

The default setting of all threads in the AutoTurn Graphic Programming System is **right-hand thread**!

The explanations regarding thread undercuts also apply to threads based on the inch system (Whitworth and UN threads)!
5.5.10 Correction

**Sequence of operations**

After you have entered a contour or a contour definition in the finished part geometry and confirmed your inputs with "OK", the "Correct" softkey will be displayed. You can correct the entire workpiece geometry with the following functions:
- Delete
- Insert
- Change
- Allowance
- Dimension
which you will find in this menu!

**Delete**

**Sequence of operations**

You can choose various options for negating elements under the "Delete" softkey.

The element on the far left is always deleted when you select the "Last element" softkey.

After selecting the "Element" softkey, you can choose the element to be deleted by means of the cursor keys and then remove the color-highlighted element by selecting the "Delete" softkey.
Depending on the individual geometric structure, the gap resulting from the deletion will be filled (example: deletion of an undercut) or you may need to insert a geometry element which fits the gap exactly!

**Sequence of operations**

The transition elements "Rounding" and "Chamfer" are selected with the cursor keys and then deleted.

You can cancel the delete operation by selecting "Recall".

Attributes are additional features such as **roughness heights** or **fit classes** which you can select by means of the cursor keys and then delete individually.

When you select the "All" softkey, the inquiry: "Delete complete geometry (possibly with machining) Y/N ?" is activated. If you acknowledge it with the "Yes" softkey, **the entire contour you have programmed will be deleted irrevocably.**

**Sequence of operations**

When you choose "Insert"

you can select the insertion position for an element with the "Element" softkey and the cursor keys (element is inserted to left of cursor position)

and confirm your input with this softkey. You can then program an element from the softkey selection: **Cylinder Taper Circle Contour Other form**

and then insert it by selecting "Input".
How does geometry respond to insertion of new element?
Insertion of a new element is similar to overwriting. The element to be replaced is overwritten by the contour element you insert. It is not possible to "shift" the following contour to make room for the new element.
If the element to be inserted is to extend beyond the element boundaries at the insertion point, you must delete the following entries.

**Sequence of operations**

Apart from "Element", the softkeys listed on the left are also provided to help you insert elements:

Select the appropriate softkey,
select the point of insertion for the new element with the cursor keys,
confirm the insertion point with "Position OK" and enter the dimensions.

The contour you have inserted will then be displayed when you select "Input".

You can exit from the "Insert" menu by selecting "Recall".

Selection of this softkey will return you to the "Correct" menu.
Change

**Sequence of operations**

By selecting the "Change" softkey, you can alter dimensions for roundings and chamfers or modify attributes. You can do this by selecting the feature to be modified with the cursor keys, confirming the position and then overwriting the displayed values. Your changes are then immediately effective.

By selecting "Recall", you can exit this function without transferring changes to the system.

Allowance

**Sequence of operations**

You can also apply additional allowances either to specific contour sections or to the whole contour: You can define the starting point using the cursor keys, then confirm it with this softkey and finally locate the end point and confirm it with this softkey.

Then use these keys to enter the equidistant allowance (note sign!). The displayed diagram gives you a "preview" of the contour you are creating.

You can transfer the newly displayed contour to the system by selecting "Yes" or reject it by selecting "No" and then enter new values.

You can exit from "Allowance" without storing any changes by selecting "Recall".
Dimension

Sequence of operations

If you select the "Dimension" softkey and then select the measuring point by means of the cursor keys or softkeys,

you will have the opportunity to check the calculated dimensions along diameter axis D and longitudinal axis L to the right and left of the selected contour section on the screen.

In this case, the measuring point is always positioned on the left and the selected, color-highlighted contour to the right of it.

It is always the last contour to be programmed, e.g. for internal machining, which is measured.

You can return to the "Correct" menu by selecting this softkey.

"Recall" has the same function as "Back".
5.6 Mirror workpiece/machining position

Once you have programmed the finished part contour you can mirror the entire workpiece with the softkeys "Position 1" or "Position 2".

This function allows you to machine a side of your choice as the first side.

**Further notes**

If the finished part contour has already been entered, you can change its position later using the "Workpiece position" softkey.

5.7 Cut-off part

**Sequence of operations**

If your workpiece is a cut-off part, then please answer the query: "Cut-off part? (Y/N)"

by selecting the "Yes" softkey.

A diagrammatic representation of a finished part in a blank with the cut-off allowance AB:

will then be displayed on the screen.

You can transfer the default setting to the system or enter the dimension you require.
The cut-off allowance specifies the quantity of residual material on the left-hand end face of the finished part after the parting process. The programming sequence is continued with selection of the workholder.

If you select a collet, the system assumes that you want to machine a bar. The following screen windows are then opened during the automatic program generation run.

You must enter the appropriate conditions in the input fields of these windows and confirm your inputs with "OK".

**Sequence of operations**

- **Bar-pulling into working position**
  - (0) Do not pull
  - (1) Hand (after progr. stop) to stop
  - (2) Bar advance to fixed stop
    - (3) with workpiece gripper.

- **Cut off part**
  - (0) No removal, continue: Part drops off
  - (1) Remove with receiving pan, continue: Cut off and remove part with swiveling receiving pan
  - (2) Remove via NC gantry, continue: Cut off and remove part with NC gantry

- **Define cut-off parameters**
- **Define cut-off tool**
  - Transfer to system with "OK" or enter new values,
  - Define cutter material.

The automatic machining plan calculations are then executed. The NC code parameterized for your machine is then generated according to the settings defined above.

You will find an explanation of the machining plan in Section 8.1.2!
5.8 Workholders

Workholders are chucks and tailstocks. In this section, you will learn what you need to know about selecting workholders for the example workpiece. Once you have programmed a workpiece in AutoTurn, select a suitable workholder. Your machine setter will find full details about the workholder data dialog in Subsection 3.2.5.

5.8.1 Machining side

Workpiece clamping on 1st side

**Sequence of operations**

Use the "Workpiece position" softkey to select the position for clamping the workpiece.

Press the softkeys "Position 1" or "Position 2" to specify the position.

If you have not made any selection, Position 1 is automatically taken when you press the "Continue" softkey.
Workpiece machining on the 2nd side

Sequence of operations

With the softkeys you specify whether the finished part is to be machined from one or two sides. If you have selected machining from two sides you then choose between machining on the main spindle or counterspindle.

If you are only machining one side, press the softkey "no 2nd side".

If machining is carried out in the main spindle, choose the "Main spindle" softkey.

If you machine the workpiece in the counterspindle, choose the "Counterspindle" softkey.

Once you have selected the type of machining, you can select whether the workpiece is to be cut off or not with the "Yes" or "No" softkey.

Details of cut-off are given in Section 5.7, Cut-off part

5.8.2 Select workholder

Sequence of operations

AutoTurn offers you the following types of workholder:

Outside jaws (1) Outside jaws (2) Internal jaws (3) Collet (4) Face driver (5)

(without stop) (with stop) jaws

The workholders you select should be the same as those on your machine!

Please note that the machining limits are automatically set in front of the workholder!

Please select the workholder available on your machine.
5 Functions
5.8 Workholders

External/internal jaws

Sequence of operations

If you select a jaw (external or internal),

you can choose a soft or a hard jaw.

Once you have confirmed the jaw type, the workholder you have selected is displayed.

A list of practical external jaws is displayed in addition to the dimensioned representation of the workholder.

Select one with the cursor keys

and confirm by selecting "Input".

The relationship "Chuck height H" + "Jaw length L" - "Clamping depth T" represents the zero offset between the machine zero and the workpiece zero in the machine coordinate system with a workpiece zero on the left.

When the workpiece zero is on the right, the workpiece length must also be added.
Collet

**Sequence of operations**

When you press the "Collet" softkey, a dimensioned representation of the workholder is displayed.

Select a suitable collet from the list with the cursor keys

and confirm your selection with "Input"

You must now enter the values for the faceplate width H and the clamping depth.

Face driver

**Sequence of operations**

When you press the "Face driver" softkey, a dimensioned representation of the face driver is displayed.

Select a suitable face driver from the list with the cursor keys

and confirm your selection with "Input"

You must now enter the values for distance A from machine zero to the left end face of the workpiece.
5.8.3 Tailstock

**Sequence of operations**

Once you have selected the workholder you can choose whether you want to work with a tailstock. You can make your selection from four tailstock types with differently shaped tailstock tips.

If you want to machine without a tailstock press the "Continue" softkey

If you want to use a tailstock, select a suitable one ("Type A" to "Type D") by pressing the relevant softkey.

AutoTurn displays a list of available tailstocks for each tailstock type.

Select one from the lists using the cursor keys and confirm with "Input".
5.9 Defining machining limits

Sequence of operations

On the screen under "Programming/Machining 1st side/Limits ...

you will see the machining limits of the workpiece when clamped for machining operations:

- Outside roughing
  L ...
- Outside finishing
  L ...
- Inside drilling
  L ...
- Inside finishing.
  L ...
- D ...

Further notes

The machining limit defines up to which z coordinate machining is to be carried out. Contours that lie outside this limit (in the direction of the workholder) are not considered when the program is created and are not machined.

The AutoTurn Graphic Programming System applies the following strategy to define the machining limits:

- The limit is set either 1 mm in front of the chuck or
- 1 mm behind the last element.

Please note that it may not be possible to "finish" some relief-cut elements owing to the automatic setup of machining limits! In such cases, you must adjust the machining limits or machine the workpiece from two sides!
Sequence of operations

This is how you can modify the machining limits:

For example, you select "Outside roughing"

and can then use the cursor keys to shift the machining limits either by means of the displayed

grid softkeys from "1mm" to "100mm"

or move them element by element.

You can also enter the limit as a direct numerical input.

Use these keys to select vertical/horizontal machining limits. Horizontal machining limits are displayed only for relief cuts.

You must confirm the coordinates by selecting "Position OK".

The set machining limits are taken into account when the NC program is generated so that you can safely assume that no workpiece machining will take place beyond the limit values (in the direction of the workholder)!

You can check that this safety function works in the dynamic graphic simulation!
Toolholder Preassignment

6.1 Description ................................................................................................................. 6-118
6.2 Operation................................................................................................................... .... 6-119
6.3 Toolholder without counterspindle................................................................................. 6-121
6.4 Counterspindle machining with 1 or 2 toolholders......................................................... 6-122
6.1 Description

Function

Under this menu you make sure that the toolholder setting of the AutoTurn Graphic Programming System really matches that of your machine.

Before you test the part programs developed on the AutoTurn Graphic Programming System on your machine, you must always check that the theoretical toolholder set matches the real memory assignment!

If the wrong tools are used, human injury and damage to machinery and equipment can result!
6.2 Operation

Sequence of operations

You are in the AutoTurn main menu and press the "Toolholder preassign." softkey.

The horizontal softkey bar with the following keys: "Programming/Machine set-up/Preset/Toolholder 1" lists the tool groups already described in Subsection 2.1.1.

Using the cursor keys, you select

or enter a letter directly to choose a tool group

and confirm the selection with "OK". The existing tool is now displayed with its data.

If other tools are also available in the group, you can select them

with the softkeys

and the message "Tool No. ... of ..." is displayed.

Press "OK" to accept a tool and the following acknowledgement is displayed:

"Number of selected tools: ..."
Sequence of operations

The "Set-up" softkey opens a window containing the tools selected for fitting on the left and the toolholders with the tools already fitted on the right.

With the "Zoom in" softkey you can view the tool with all the data relevant for machining,

before you put it in the toolholder by pressing the "Tool OK" softkey (select the position with the cursor)

and then the "Exchange" softkey.

With this softkey you can toggle between the presetting left and the toolholder right

and with "Exchange" you can swap the assignment of any two tool locations:

Use the cursor keys to select the first tool to be exchanged (light background) and trigger the exchange.

The second tool to be swapped round is selected and the swap is completed.

"Delete" deletes a tool from the toolholder.

Various settings can be saved under a user-defined name and loaded again later.

Underneath the "Memory" softkey you can select between the "Read", "Save" and "Delete" softkeys.

The functions refer to the memory.
If all the tools required for machining exist, this is indicated by the following message (even without a tool change!):
"All tools have been exchanged!".

6.3 Toolholder without counterspindle

Under the path \WOP\Dat200.007\Ma770.ppc, you will find a script that describes the fundamental machine. It defines whether the machine is equipped with or without steady rest, receiving unit or tailstock. The relevant parameters are described in the comments.

```plaintext
// ****************** SIEMENS AG ***********************
// *** PP code 836
// *** CREATED 09.10.97 // JOR
// DESCRIPTION OF THE MACHINE with 1 toolholder without counterspindle
// Spindle opening 60 mm

// General parameters
#WZVER 0  // Tool meas. on machine 0=no 1=yes
#LUN 0  // Steady rest 0=no 1=yes
#ABNEHM 2  // Receiving unit 0=none 1=rec.pan 2=NC gantry 3=gripper TH2
#RSTRUH 0  // Tailstock rest position 0=no t.

// *************** Spindle 1 ***********************
#SPI 1  // Spindle number
#SMAX 1111  // Max. spindle speed
#AUSR 1  // Orient and switching dev. 0=none 1=f.mot 2=Dir.dr 3=rot. sw.

// TOOLHOLDER 1
#WZT 1  // Spindle number
#EILG 27000  // Rapid traverse velocity
#REF 205 710  // Toolholder reference point TH (x/z)
#ARBMAX 111 111  // Working range (x/z max)
#ARBMIN 11 11  // Working range (x/z min)
#TANZ 6  // Number of stations 1= no rev.
#CANZ 1  // Number of paths
#CVER.1 0 0  // Conversion of 1st path (x/z)
#SANMAX 4000  // Max. speed of drive. 0=no drive
#KZW1 450  // Tool transformation angle
#CAX 2  // C axis 0=no 1=without 2=with tr.
#YAX 2  // Y axis 0=no 1=yes 2=real
```

© Siemens AG, 2002. All rights reserved.
You can set 1 or 2 toolholders for counterspindle machining with AutoTurn Plus.

Modify the file \WOP\Dat200.007\Ma770.ppc as described below, if you want to set 1 toolholder. You must activate or deactivate the lines that are commented out ("//") according to your tools (here 1 toolholder).

1 toolholder:

```
#BAX  1          // B axis 0=no 1=yes
#GBOOAF  1      // Rigid tapping  0=no 1=yes
```

// ******************* SIEMENS AG **********************
// *** PP code  836
// *** CREATED 09.10.97 // JOR
// DESCRIPTION OF THE MACHINE with 2 toolholders and counterspindle
// Spindle opening 60 mm
#
#
#EXE        "WPP"                         // EXE to be executed
#MATEXT     "MANUFACTURER + Ggs/Y/B"      // Machine description
#VERS       "V4.2+"                       // Version
#DATE       "09.10.97"                    // Date
#MATYP      "G300"                        // Machine type
#
#
#WZVER 0                         // Tool meas. on machine 0=no 1=yes
#LUN 0                         // Steady rest 0=no 1=yes
#ABNEHM 2                         // Receiving unit 0=none 1=rec.pan
#RSTRUH 0                        // Tailstock rest position 0=no t.
#
#
# *** Spindle 1 ********************************
#
#SPI 1                       // Spindle number
#SMAX 1111                    // Max. spindle speed
#AUSR 1                      // Orient and switching dev. 0=none 1=f.mot
#               2=Dir.dr 3=rot. sw.
#
#
# *** TOOLHOLDER 1
#
#WZT 1                       // Spindle number
#EILG 27000                   // Rapid traverse velocity
#REF 205 710                   // Toolholder reference point TH (x/z)
#ARMAX 111 111               // Working range (x/z max)
#ARMIN 11 11                  // Working range (x/z min)
#TANZ 6                       // Number of stations 1= no rev.
#CANZ 1                       // Number of paths
6.4 Counterspindle machining with 1 or 2 toolholders

MMC 103  PCU 50

//  TOOLHOLDER 2
//  #WZT 2  // Spindle number
//  #EILG 27000  // Rapid traverse velocity
//  #REF 195 710  // Toolholder reference point TH (x/z)
//  #ARBMAX 222 222  // Working range (x/z max)
//  #ARBMIN 22 22  // Working range (x/z min)
//  #TANZ 9  // Number of stations 1= no rev.
//  #CANZ 1  // Number of paths
//  #CVER.1 0 0  // Conversion of 1st path (x/z)
//  #SANMAX 4000  // Max. speed of drive. 0=no drive
//  #WZTWI 90  // Tool transformation angle
//  #CAX 2  // C axis 0=no 1=without 2=with tr.
//  #YAX 0 0  // Y axis 0=no 1=yes 2=real
//  #BAX 0 0  // B axis 0=no 1=yes
//  #GBOOAF 1  // Rigid tapping 0=no 1=yes

//  TOOLHOLDER 2
//  #WZT 2  // Spindle number
//  #EILG 27000  // Rapid traverse velocity
//  #REF 195 710  // Toolholder reference point TH (x/z)
//  #ARBMAX 222 222  // Working range (x/z max)
//  #ARBMIN 22 22  // Working range (x/z min)
//  #TANZ 9  // Number of stations 1= no rev.
//  #CANZ 1  // Number of paths
//  #CVER.1 0 0  // Conversion of 1st path (x/z)
//  #SANMAX 4000  // Max. speed of drive. 0=no drive
//  #WZTWI 90  // Tool transformation angle
//  #CAX 2  // C axis 0=no 1=without 2=with tr.
//  #YAX 0 0  // Y axis 0=no 1=yes 2=real
//  #BAX 0 0  // B axis 0=no 1=yes
//  #GBOOAF 1  // Rigid tapping 0=no 1=yes

//  TOOLHOLDER 2
//  #WZT 2  // Spindle number
//  #EILG 27000  // Rapid traverse velocity
Modify the file \WOP\Dat200.007\Ma770.ppc as described below, if you want to set 2 toolholders. You must activate or deactivate the lines that are commented out ("//") according to your tools (here 2 toolholders).

2 toolholders:

```plaintext
// ****************** SIEMENS AG ***********************
// *** PP code  836
// *** CREATED 09.10.97 // JOR
// DESCRIPTION OF THE MACHINE with 2 toolholders and counterspindle.
// Spindle opening 60 mm
//
// #EXE        "WPP"                         // EXE to be executed
#MATEXT     ?MANUFACTURER + Ggs/Y/B"      // Machine description
#VERS       "V4.2+"                       // Version
#DATE       "09.10.97"                    // Date
#MATYP      "G300"                        // Machine type
#
// General parameters
#WZVER          0                         // Tool meas. on machine 0=no 1=yes
#LUN            0                         // Steady rest 0=no 1=yes
#ABNEHM         2                         // Receiving unit 0=none 1=pan
// 2=NC gantry 3=gripper TH2
#RSTRUH         0                         // Tailstock rest position 0=no t.
#
// Spindle 1
#
#SPI               1                       // Spindle number
#SMAX           1111                       // Max. spindle speed
#AUSR              1                       // Orient and switching dev. 0=none 1=f.mot
// 2=Dir.dr 3=rot. sw.
#
# TOOLHOLDER 1
#
#WZT               1                       // Spindle number
#EILG          27000                       // Rapid traverse velocity
#REF             205            710        // Toolholder reference point TH (x/z)
#ARMAX           111            111        // Working range (x/z max)
#ARMIN           11             11        // Working range (x/z min)
#TANZ             18                       // Number of stations 1= no rev.
#CANZ              1                       // Number of paths
```

```plaintext
Modify the file \WOP\Dat200.007\Ma770.ppc as described below, if you want to set 2 toolholders. You must activate or deactivate the lines that are commented out ("//") according to your tools (here 2 toolholders).

2 toolholders:

```plaintext
// ****************** SIEMENS AG ***********************
// *** PP code  836
// *** CREATED 09.10.97 // JOR
// DESCRIPTION OF THE MACHINE with 2 toolholders and counterspindle.
// Spindle opening 60 mm
//
// #EXE        "WPP"                         // EXE to be executed
#MATEXT     ?MANUFACTURER + Ggs/Y/B"      // Machine description
#VERS       "V4.2+"                       // Version
#DATE       "09.10.97"                    // Date
#MATYP      "G300"                        // Machine type
#
// General parameters
#WZVER          0                         // Tool meas. on machine 0=no 1=yes
#LUN            0                         // Steady rest 0=no 1=yes
#ABNEHM         2                         // Receiving unit 0=none 1=pan
// 2=NC gantry 3=gripper TH2
#RSTRUH         0                         // Tailstock rest position 0=no t.
#
// Spindle 1
#
#SPI               1                       // Spindle number
#SMAX           1111                       // Max. spindle speed
#AUSR              1                       // Orient and switching dev. 0=none 1=f.mot
// 2=Dir.dr 3=rot. sw.
#
# TOOLHOLDER 1
#
#WZT               1                       // Spindle number
#EILG          27000                       // Rapid traverse velocity
#REF             205            710        // Toolholder reference point TH (x/z)
#ARMAX           111            111        // Working range (x/z max)
#ARMIN           11             11        // Working range (x/z min)
#TANZ             18                       // Number of stations 1= no rev.
#CANZ              1                       // Number of paths
```
#CVER.1 0 0 // Conversion of 1st path (x/z)
#SANMAX 4000 // Max. speed of drive. 0=no drive
#WZTWI 450 // Tool transformation angle
#CAX 2 // C axis 0=no 1=without 2=with tr.
#YAX 2 // Y axis 0=no 1=yes 2=real
#BAX 1 // B axis 0=no 1=yes
#GBOODAF 1 // Rigid tapping 0=no 1=yes

//
// TOOLHOLDER 2
#WZT 2 // Spindle number
#EILG 27000 // Rapid traverse velocity
#REF 195 710 // Toolholder reference point TH (x/z)
#ARBMAX 222 222 // Working range (x/z max)
#ARBMIN 22 22 // Working range (x/z min)
#TANZ 12 // Number of stations 1= no rev.
#CANZ 1 // Number of paths
#CVER.1 0 0 // Conversion of 1st path (x/z)
#SANMAX 4000 // Max. speed of drive. 0=no drive
#WZTWI 90 // Tool transformation angle
#CAX 2 // C axis 0=no 1=without 2=with tr.
#YAX 0 0 // Y axis 0=no 1=yes 2=real
#BAX 0 0 // B axis 0=no 1=yes
#GBOODAF 1 // Rigid tapping 0=no 1=yes

//
//************************ Spindle 2 ************************
//
#SPI 2 // Spindle number
#SPITYP 2 // Spindle type 1=synchronous spindle >1 counterspind.
// 2=with 3=without electronic shaft
#SPIABST 640 // Machine zero distance main/counterspind.
#SMAX 2222 // Max. spindle speed
#AUSR 1 // Orient and switching dev. 0=none 1=f.mot

//
// TOOLHOLDER 1
#WZT 1 // Spindle number
#EILG 27000 // Rapid traverse velocity
#REF 205 710 // Toolholder reference point TH (x/z)
#ARBMAX 111 111 // Working range (x/z max)
#ARBMIN 11 11 // Working range (x/z min)
#TANZ 18 // Number of stations 1= no rev.
#CANZ 1 // Number of paths
#CVER.1 0 0 // Conversion of 1st path (x/z)
#SANMAX 4000 // Max. speed of drive. 0=no drive
#WZTWI 90 // Tool transformation angle
#CAX 2 // C axis 0=no 1=without 2=with tr.
#YAX 2 // Y axis 0=no 1=yes 2=real
#BAX 1 // B axis 0=no 1=yes
#GBOODAF 1 // Rigid tapping 0=no 1=yes

//
// TOOLHOLDER 2
#WZT 2 // Spindle number
#EILG 27000 // Rapid traverse velocity
### Counterspindle machining with 1 or 2 toolholders

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value 1</th>
<th>Value 2</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>#REF</td>
<td>195</td>
<td>710</td>
<td>Toolholder reference point TH (x/z)</td>
</tr>
<tr>
<td>#ARBMAX</td>
<td>222</td>
<td>222</td>
<td>Working range (x/z max)</td>
</tr>
<tr>
<td>#ARBMIN</td>
<td>22</td>
<td>22</td>
<td>Working range (x/z min)</td>
</tr>
<tr>
<td>#TANZ</td>
<td>12</td>
<td></td>
<td>Number of stations 1= no rev.</td>
</tr>
<tr>
<td>#CANZ</td>
<td>1</td>
<td></td>
<td>Number of paths</td>
</tr>
<tr>
<td>#CVER.1</td>
<td>0</td>
<td>0</td>
<td>Conversion of 1st. path (x/z)</td>
</tr>
<tr>
<td>#SANMAX</td>
<td>4000</td>
<td></td>
<td>Max. speed of drive. 0=no drive</td>
</tr>
<tr>
<td>#WZTWI</td>
<td>90</td>
<td></td>
<td>Tool transformation angle</td>
</tr>
<tr>
<td>#CAX</td>
<td>2</td>
<td></td>
<td>C axis 0=no 1=without 2=with tr.</td>
</tr>
<tr>
<td>#YAX</td>
<td>0</td>
<td>0</td>
<td>Y axis 0=no 1=yes 2=real</td>
</tr>
<tr>
<td>#BAX</td>
<td>0</td>
<td>0</td>
<td>B axis 0=no 1=yes</td>
</tr>
<tr>
<td>#GBOOAF</td>
<td>1</td>
<td></td>
<td>Rigid tapping 0=no 1=yes</td>
</tr>
</tbody>
</table>
Examples of Program Sequences

7.1 Programming example: Basic functions ................................................................. 7-128
  7.1.1 General .............................................................................................................. 7-128
  7.1.2 Program selection ............................................................................................. 7-130
  7.1.3 Defining the blank ............................................................................................ 7-131
  7.1.4 Entering the finished part geometry ................................................................. 7-133
  7.1.5 Selecting the workholder for the workpiece ..................................................... 7-143

7.2 Programming example: Any blank forms ............................................................. 7-146

7.3 Programming example: End face machining ........................................................... 7-150

7.4 Programming example: Machining on the peripheral surface ............................... 7-159
7.1 Programming example: Basic functions

7.1.1 General

Two examples are used to explain part programming in full detail (Sections 7.1 and 7.2) using programming steps applied in practice (step by step). The machining stages have been divided into subsections to help you find your way through the programming sequence.

If it is possible to choose alternative solutions and operator actions at certain stages in the programming sequence, reference is made to the description in the relevant function under the "Cross-reference" symbol. This means that you are immediately able to program other parts in a similar way to the example workpiece.
You want to program a new workpiece and you have the workpiece drawing in front of you:

Blank: cylinder Φ 120, L 72
End face: 6 threaded holes M 6x17, hole circle 60

**Parameters**

The workpiece
- must be machined outside and inside,
- will have an outside contour comprising the geometry elements "cylinder" and "taper",
- will have an inside contour programmed by contour definition (possibly using cylinder/taper)
- will include roundings, chamfers, an undercut and
- will be provided with a metric thread on the outside.
7.1 Programming example: Basic functions

Data
You need to know the following data:
- Dimensions of the blank,
- material to be machined,
- finished part contour,
- toolholder assignment and
- type of workholder

7.1.2 Program selection

Sequence of operations
After you have called the AutoTurn Graphic Programming System you will find yourself in the main menu (see Section 1.4).

When you select the "Programming" softkey, a window is opened in which you enter a program name at the position of the flashing cursor via the alphanumeric keyboard.

Enter, for example, PART5678 and confirm the input by means of the "Input" key.

The program name may have a maximum of 17 characters and must not contain any blanks or special characters.

If you answer the inquiry by selecting the "Yes" softkey, you switch to the program processing mode.
7.1.3 Defining the blank

Selecting the blank stock

Once you have defined the program name in the last section and confirmed with "Input", you will now see a selection of "Programming / Blank ..."

the following types of blank stock:

| Cylinder | Hollow cylinder | Square | Hexagon | Free definition | Allowance finish. part |

Sequence of operations

Your blank is a cylinder.

Now enter the
Blank diameter: 120 and the
Blank length: 72

and confirm each with "Input".

You can use the cursor keys to switch input fields and to alter values (as described above).

Confirm with the "OK" softkey.

You will now be given a selection list of materials:

  ALCUMGPB
  C45
  C60

Please refer to Chapter 5.4 for a description of the function "Blank".
Selecting the material

Sequence of operations

Select the material you want by means of the paging and cursor keys (example: C60)

and confirm your input by selecting "OK".

You have now completed input of the blank data!

If you have input an incorrect value by mistake, you can alter the entered values by selecting the softkeys "Recall" and "Blank" or confirm them if appropriate.

If your application requires the entry of new materials in this list, then ask your machine setter to make the appropriate entries according to the instructions in Chapter 4 "Cutting values".
7.1.4 Entering the finished part geometry

Positioning the finished part

After you have entered the last "OK" confirmation of the blank data in Section 5.4, the blank representation appears on the screen under: "Programming/Finished part/Length ..." in a coordinate system with the longitudinal axis length \( L \) and the facing axis (transverse axis) diameter \( D \) and a zero point on the right. Dimensions along the longitudinal axis have a negative sign to the left.

The blank length you have already stored is displayed.

All variations of the finished part function are described in Section 5.5.

In this menu, you will have your first opportunity in the programming sequence to influence the representation (Zoom and Move) of the workpiece according to the explanatory information in Section 5.5.2.
**Sequence of operations**

Enter the required finished part length as an absolute value (without sign): 70.

Confirm by selecting "OK"

and you will be offered 3 options on the screen for positioning the finished part in the blank:

- **Centered**
- **Left justified**
- **To size**

The finished part must be positioned flush with the blank on the left. Select the "Left justified" softkey and you will see this view of the blank representation.

You can change the position of the finished part in relation to the blank again by means of keys "Recall" and "Fin. part" (confirm with "OK").

Pressing the "Zero point" softkey defines the workpiece zero on the left or right side of the end face of the finished part.

The **geometry definition** is not oriented according to the position of the zero point, but always from right to left!

You can cancel the zero point shift by selecting this softkey again.
Selecting contours

Once you have entered the data for the blank and finished parts and the position of the finished part, you need to define the finished part geometry.

Under "Programming/Finished part ..."

you are offered the following contours:

Outside  Inside  End face  End face  Peripheral  Periph. surface
  left     right

You will find programming examples for the last 4 contours in Section 7.3 Machining on the end faces and Section 7.4 Machining on the peripheral surface.

Sequence of operations

Start with the external contour

Under "Programming/Finished part/Outside ...

the horizontal softkey bar with the following keys now appears on the screen:

Cylinder  Taper  Circle  Contour  Form  Correction
  definition  element

Please refer to Section 5.5, Finished part function for a detailed description of all these elements.
Programming the thread

Sequence of operations

Since the first external element starting from the right is a cylindrical metric RH thread, select the "Cylinder" softkey and then the "Metric thread" softkey.

By selecting the "Help" softkey, you can open a screen form with a selection of threads which are frequently required.

You can exit the Help form by selecting "Recall".

You want to program a thread of M80 x 2. Go to screen form: "Programming/Finished part/External/Cylinder/Metr. thread"

and enter the rated diameter: 80 and the thread lead: 2.

The end length must be entered with a sign:

End length: -25

The thread starts on the right with a chamfer

Chamfer length right: 1.5

and a chamfer angle of 45 degrees.
7.02 Examples of Program Sequences

7.1 Programming example: Basic functions

Sequence of operations

There is to be no chamfer on the left, "Chamfer left Y/N ?"

but a thread undercut
"Thread undercut left Y/N ?"

A dimensioned representation of a thread undercut according to DIN 76 form C now appears on the screen.

The AutoTurn Graphic Programming System automatically calculates the thread undercut values! You can also edit the values manually.

Confirm with the "OK" softkey.

Since you want to program an RH thread, please select the "OK" softkey.

The metric cylindrical external contour with chamfer on the right and thread undercut on the left now appears on the screen as the first finished part contour inside the blank.

You have now completed programming of the thread.

Please refer to Section 5.5.9 for further information about other thread forms and parameters!
Taper

Sequence of operations

The next contour after the thread is a taper.

Select the "Link" softkey

OR: Enter the start diameter: 80.
The last end dimension then becomes the new initial dimension!

The end diameter: 85 and the end length: -35 must still be entered.

You can enter a roughness height in mm or inch here.

The parameters of the function block geometry elements cylinder, taper and circle are described in Section 5.5.5.

You transfer the taper to the system by selecting the "OK" softkey.
The roughness height is displayed for this contour element.
Cylinder

Sequence of operations

The next contour element in the geometry definition is a cylinder.

Select the "Link" softkey and transfer the end diameter of the taper as the cylinder diameter.

Then enter the end length: -45.

You can use roughness heights and fit classes again here.
Please read the explanatory information in Section 5.5.8!

The rounding on the left is activated with this softkey.

Enter the radius (left): 4

and store the cylinder with rounding of which the contour is now displayed on the screen.

The last element of the external contour is again a cylinder.

Enter the diameter: 120,
which corresponds to the blank diameter.

The input of the end length: -70 is transferred to the system by means of the "Distance to go" softkey,

Complete the inputs for the external contour by selecting "OK". The entire contour appears on the screen.

You can modify the contour by means of the "Correct" key.

Press "Back" to return to the finished part.
Contour definition

Sequence of operations

Now you continue with the internal contour.

Contour definitions are characterized by the fact that the end point of the preceding contour element is always the initial point of the next contour element. If the first element of the external or internal contour is a contour definition, then the definition always begins at the starting point on the right.

Part of the internal contour must be programmed as a contour definition. In the geometry definition, however, a contour definition can only implement chamfers on the left-hand side. For this reason, simply start here with a cylinder.

Press the "Cylinder" softkey and enter the diameter: 40.
The end length is: -25.

Call "Chamfer right" and enter the chamfer length: 2,
accept the chamfer angle 45 (degrees) and store the cylinder contour with chamfer.

Press "OK" to confirm and continue programming the contour definitions.
Sequence of operations

Press the "Contour definition" softkey

and choose a "Straight line"

with any direction.

Only with this softkey that you have just selected will you obtain optional straight lines in the plane, the other 3 softkeys always create lines that are parallel to an axis!

Enter the diameter at the end point of the straight line: 30 (the start point is defined by the preceding contour element!) and enter the end length: -45.

The rounding which is added on the left

is programmed with a radius (left): 8.

Transfer the optional straight line to the system with the "OK" softkey.

The next contour element is also a straight line

which runs to the left in parallel to an axis.
7.1 Programming example: Basic functions

Sequence of operations

You don't need to enter any end length here. The "Distance to go" softkey transfers the finished part dimension to the system.

Complete your inputs for the internal contour by selecting "OK"

and exit from the menu "Contour definition" and the "Internal" contour definition by selecting the "Back" softkey twice.

You have now entered all the contour data for the finished part!

Before you select the "Back" softkey twice in the previous section, you can modify your workpiece geometry before transferring it to the system by selecting the "Correct" softkey.

The correction inputs are described in detail in Section 5.5.10.
7.1.5 Selecting the workholder for the workpiece

Sequence of operations

After you have defined your finished part contour, proceed to the workholder selection menu by selecting "Continue". The workpiece is to be machined from one side only, so answer the inquiry:
"Machining of 2nd side?"

by selecting the "Continue" softkey again.
The "No 2nd side" softkey with the blue background is the default setting and therefore has the same meaning as "Continue".
Nor is there a cut-off part which means you must acknowledge the inquiry:
"Cut-off part? (Y/N)"
by selecting "Continue".

On the screen under
"Programming/Machining 1st side/Workholders..."
you will now see a selection of 5 workholders:

Outside jaws (1) Outside jaws (2) Inside jaws (3) Collet (4) Face driver (5)

The workholders you select should, of course, be the same as those on your machine!
Please note that the machining limits are automatically set in front of the workholder!
Select the workholder on your machine and perform the following sequence of operations accordingly.
Sequence of operations

You decide to select "Outside jaws (2)"

and "Outside jaws, soft with stop (2)".

In addition to the dimensioned drawing of this workholder, you are also provided with a selection list of practical outside jaw chucks. Using the cursor keys, you select:

E / 77.12 / Q70 L40 T12.

After selecting "Input", you can modify dimension H (width of face plate).

The relationship "Chuck height H" + "Jaw length L" - "Clamping depth T" + "Workpiece length" represents the zero offset between the machine zero and the workpiece zero.

Confirm the workholder data with "OK". Four types of tailstock with various shaped tailstock tips then appear on the screen.
Sequence of operations

Since no tailstock is required for this short workpiece, acknowledge with the "Continue" softkey.

On the screen under "Programming/Machining 1st side/Limits ..."

you will see the machining limits of the workpiece when clamped for machining operations:

- Outside roughing
- Outside finishing
- Inside drilling
- Inside finishing.

The machining limits remain unchanged for this example. Press "Continue" to proceed.

You have now completed programming for the example. To generate an executable part program, you must create a work plan and select the tools to be used.

Information about program creation and setup is given in Chapter 8, Program Generation and Setup.
7.2 Programming example: Any blank forms

This chapter presents two versions: The "free blank definition" and the "allowance machined part".

Under: "Programming / Blank ..." you are offered the following:

- Cylinder
- Hollow cylinder
- Square
- Hexagon

Free definition

Allowance finished part

rchapter presents two versions: The "free blank definition" and the "allowance machined part".

Under: "Programming / Blank ..." you are offered the following:

- Cylinder
- Hollow cylinder
- Square
- Hexagon

Free definition

Allowance finished part
7.2 Programming example: Any blank forms

**Sequence of operations**

Your blank type is freely defined:

Enter the blank diameter: 125 and the blank length: 170

and confirm these values with "OK".

In the coordinate system of length L 0 ... -170 and diameter D 0 ... 125 now displayed continue to program the blank contour:

The outside contour is described starting from the zero point located on the right with a cylinder, with the diameter: 85, and end length: -62.5 and a rounding left with the radius: 10.

Accept the cylinder with "OK".
Examples of Program Sequences

7.2 Programming example: Any blank forms

Sequence of operations

Another cylinder is added on the left
with the diameter: 125
and an end length: -120
Accept by selecting the "OK" softkey.

Another cylinder is added on the left in the direction of programming
with the diameter: 120,
an end length: -145
and a rounding to the right
with a radius: 2.
Store this cylinder by selecting "OK".

The last contour section of this freely defined blank
is again a cylinder
with the diameter: 100
and the remaining distance to go as the end length.
Accept this cylinder too by selecting the "OK" softkey.
You have now defined your first "free blank".

The "Back" softkey takes you back to the menu "Outside/Inside",
where you can program an additional inside geometry
for your new blank, if required.
Press "Continue" to switch to material selection.
(see Subsection 5.4.3).

Once you have assigned a material to the new blank
and entered the finished part length: 165,
position the finished part in the middle (centered).
7.2 Programming example: Any blank forms

**Alternative operating sequence**

If your blank has an **equidistant allowance** around the finished part contour you can define the blank geometry with the "Allowance Fin. part" softkey instead of the "Free definition" softkey.

Finally, enter the maximum finished part diameter: 120,

the equidistant allowance: 2.5,

the finished part length: 165

and confirm with "OK".

After selecting the material continue with the description of the outside contour of the machined part:

In the example:

- Cylinder Diameter: 80   End length: -62.5   rounding left: 10.0
- Cylinder Diameter: 120   End length: -115
- Cylinder Diameter: 105   End length: -140   Rounding right: 7.5
- Cylinder Diameter: 95    End length: -165

In this case, you can only return to the finished part programming to make a correction via the "Blank" softkey.
7.3 Programming example: End face machining

In addition to the 2 contours of the finished part geometry
External  Internal
four additional contour planes are suggested:
End face right  End face left  Peripheral  Peripheral surface

You can program holes, thread holes, slots and free contours
(pockets, islands, open contours) on each of these 4 new contour
planes.

In this section, this is demonstrated to you using the example
of a pocket placed on the right end face with a free contour.

Machining of a workpiece peripheral surface is explained in
Section 7.4.

The premachining step "Longitudinal and face turning" required before
face and peripheral machining can begin is omitted from this
description for reasons of clarity.
Sequence of operations

Continue programming your workpiece by pressing the "End face right" softkey which gives you the choice of the following elements:

<table>
<thead>
<tr>
<th>Hole</th>
<th>Thread hole</th>
<th>Free contour</th>
<th>Form element</th>
</tr>
</thead>
</table>

You press the "Free contour" softkey and want to program a "Pocket".

With "Input" you accept the reference length: 0,

enter a depth: 5

and confirm with "OK".

You can now enter explicit values or make a selection from the construction geometries:

Using the construction geometries the AutoTurn Graphic Programming System calculates the unknown contour points.

In this example, initially the entire free pocket contour is put together from construction geometries and then the contour that you actually want to machine is put together by selecting geometry elements and their links with intermediate points SP1 to SP5.
7.3 Programming example: End face machining
7.3 Programming example: End face machining

Sequence of operations

Construction line A:
X value P1: 0  Y value P1: -30  Angle: 120  Distance to P1: 0
After you have made the last of 4 entries the construction line A appears on the screen
(P1 always refers to the construction line!).

Construction line B:
X value P1: 0  Y value P1: -30  Angle: 45  Distance to P1: 0
Construction circle C:

X value MP: -17  Y value MP: 15  Radius: 15

Construction circle D:

X value MP: 17  Y value MP: 0  Radius: 15
7.3 Programming example: End face machining

Construction line E:

X value P1: 0  Y value P1: 15  Angle: 0  Distance to P1: 0

You have now created all the construction geometries and can now continue programming by entering the geometry elements.
7.3 Programming example: End face machining

Sequence of operations

Select intersection point SP1 with these softkeys and confirm your selection with "OK". This contour intersection point is then displayed as the starting point on the contour.

If you continue programming in the clockwise direction, the next contour is a circle with clockwise direction of rotation.

You enter:
X value MP: -17   Y value MP: 15

X value EP: ? The end point of this circle is unknown and is calculated using a new construction geometry:

Press the "Construction geometries" softkey to recall the previously defined construction geometries.
Select intersection point SP2 and confirm it with "OK".

After the next confirmation with "OK" (no rounding, roughness height) the circle is displayed as the first geometry element.
**Sequence of operations**

The next geometry element is a straight line pointing to the right.

X value EP: ?
The unknown end point is again calculated from the construction geometry:

Select SP3

and confirm with "OK".
No inputs are required for the rounding, chamfer and roughness height so you can press the "OK" softkey again.

Next follows a circle

in the clockwise direction with a tangential transition and a radius: 15.

X value EP: ?
The unknown end point (SP4) is calculated by AutoTurn:

Select the point of intersection SP4

and confirm with "OK".
Sequence of operations

The next element joined to the circle is a straight line
with any direction
and the end point values

The transition from the straight line to the next geometry element
is to be formed by a rounding
with a radius (r) of: 6.
The rounding with R6 is accepted with "OK".

The freely programmed pocket contour is completed
with a straight line
in any direction
for which, again, the unknown end point X value EP: ?
is calculated from the construction geometry:

Select the starting point (SP1) from the contour
and confirm with "OK".

You have now completed definition of the pocket contour.

Save the pocket which you have programmed on the right end face of
your workpiece by answering the prompt "Save object?" with "Yes"
which appears after you press the "Back" softkey.

The entire pocket contour is now displayed.

A contour of this type is machined with the single machining step
function.
This function is described in more detail in the next Section.
7.4 Programming example: Machining on the peripheral surface

Sequence of operations

Once you have programmed the contour on the end face described in the previous section you can continue by creating the contour on the peripheral surface.

Again, you are given the following technology options:

<table>
<thead>
<tr>
<th>Hole</th>
<th>Thread hole</th>
<th>Free contour</th>
<th>Form element</th>
</tr>
</thead>
<tbody>
<tr>
<td>Form element</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

You want to program a "Form element"

Press this softkey to select a "Tapered groove".

In the screenform that is now displayed enter the following dimensions for the cylinder path on the peripheral surface:

- Angle W1: -30°  X1: -100
- Angle W2: -80°  X2: -100
- Width W:  12
- Depth D:  6

Please note:
All dimensions are entered negatively on the peripheral surface because the zero point on the development is located top right.

Confirm these values with "OK".
Sequence of operations

If you press this softkey you return to the initial menu via:
"Define position" - "Cut-off Yes/No" - "Clamp"

and press the "Automatic" softkey
to let AutoTurn create the optimum NC program (see Section 8.1).

Please check the machining limits for "Ext. rough" and "Ext. finish",
since the groove will otherwise not be machined.
# Program Generation and Setup

<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>8.1 Program generation/Machining plan</td>
<td>8-162</td>
</tr>
<tr>
<td>8.1.1 Program generation</td>
<td>8-162</td>
</tr>
<tr>
<td>8.1.2 Machining plan</td>
<td>8-164</td>
</tr>
<tr>
<td>8.2 Simulation</td>
<td>8-166</td>
</tr>
<tr>
<td>8.3 Setup dialog</td>
<td>8-167</td>
</tr>
<tr>
<td>8.4 Program handling</td>
<td>8-168</td>
</tr>
</tbody>
</table>
8.1 Program generation/Machining plan

8.1.1 Program generation

Sequence of operations

You have now performed the following programming steps for the example workpiece:

- definition of blank,
- definition of finished part geometry,
- selection of workholder and
- definition of machining limits.

After you have checked the machining limits for your workpiece, select the "Continue" softkey and you will go to the menu in which the machining operations for the contour defined above can be established. These operations are either performed automatically by the AutoTurn Graphic Programming System or initiated by your inputs under "Single machining operations" and "Macros".

"Single machining operations" and "Macros" are described in Chapter 9 - Interactive Programming!

Leave AutoTurn to generate the optimum program and select the "Automatic" softkey.

When you select this key, a window is opened on the screen in which you are offered default values for:

- speed limits,
- clamping pressures and
- with/without operator action.
Sequence of operations

Transfer these values to the system by selecting "OK".

During the automatic run, you will be informed about the generated machining steps. When all machining operations for the defined contour have been completed, the following text is displayed:

"... END OF AUTOMATIC...(0) Clamping:1".

If an error message occurs while Automatic mode is active, please refer to Chapter 17 for the causes of errors and how to eliminate them.

You have now created a whole new NC program!

If you attempt to execute a new automatic run with an NC program that already exists, the error message

"Part has already been machined => delete with softkey "Delete Machining" is displayed.

Acknowledge by selecting "Clear error".

Now select this softkey, answer the inquiry

"Do you really want to delete machining?"

with "Yes" and start a new automatic run.

The Automatic run can be influenced.

- Enter a "1" in the relevant field of the automatic start form to active the next form "Control run". By entering "1" for specific types of machining you can influence tool selection, e.g. if the tool of your preference is not selected.
- Enter "?" to query all operations without defining them individually again.
8.1.2 Machining plan

Sequence of operations

The programmed workpiece is machined in several operations which you can call by selecting the "Machining plan" softkey. This machining plan has been created in the course of automatic program generation and includes:

- all operations,
- the tool identification numbers,
- the cutting velocities in m/min and
- the feedrates in mm/rev.

You have the opportunity at this stage to modify these machining data if you wish to.

The cutting velocity and feedrate can be altered separately for each machining operation.

Please use the cursor keys for this purpose in order to select the appropriate field.

Enter the new values and confirm them by selecting "Input".

Transfer these values to the system by selecting "OK".

You need to archive all changes to the machining plan by selecting the "Save" softkey. They are activated as soon as you exit the plan by selecting "Back".
Sequence of operations

The "Machining plan" menu offers other functionalities such as
• deletion of operations,
• rescheduling of operations,
• modification of operations.

You **delete** an operation by positioning the cursor on the appropriate step using the cursor keys and then select the softkey "Delete operation".

An operation is **rescheduled** in two separate steps:

First, choose the operation to be moved using the cursor keys and mark it with the "Mark" softkey. A "***" is inserted before the machining operation to indicate that it has been highlighted.

Position the cursor on the relevant work step to modify a machining operation. Pressing the "Change mach. oper." softkey calls a screenform in which you can change several parameters.

**Sequence of operations**

Now use the cursor keys again to select the position at which the operation is to be inserted (it is always inserted in front of the cursor) and then select the "Paste" softkey. The position of the selected machining operation in the machining plan has been altered.

If you exit the machining plan via "Back" **without saving** the new data, **your changes will be rejected** after an acknowledgment request from AutoTurn.
8.2  Simulation

Sequence of operations

You will be able to watch your workpiece being machined in a graphic-dynamic simulation.

After you have selected simulation, you can either view each machining operation separately

or start a complete simulation process

in which you can control the velocity

and traverse a "Single block" while selecting the next block by means of "Input".

When you select this softkey, the updated blank is refreshed up to the point of the machining step that you have selected.

You can trace the tool paths by selecting this softkey (select once: ON, select again OFF).

By selecting the "Restart" softkey, you can reestablish the initial state prior to cutting and restart the simulation process.
8.3 Setup dialog

Sequence of operations

By selecting the "Continue" softkey, you go to the tool setup dialog:
- **On the left** you can see the tools which AutoTurn has selected when generating the program and which are not yet included in the toolholder.
- The preset toolholder is displayed on the right.
- All tools against a colored background in the magazine are required to machine the special workpiece!

You select the tools to be added on the left of the screen and confirm your input by selecting "Tool OK".

Then you transfer them to the toolholder on the right by means of the "Load tool" softkey.

Tools and tooling are described in detail in Chapters 2 and 6 respectively.

If you do not insert any tools in the toolholder error messages will be output when the NC data are generated (see Chapter 17).

Before you test on your machine a part program created in the Graphic Programming System, it is essential to check whether the theoretical toolholder placement corresponds to the practical tool memory assignment!

You can use the setup sheet generated by AutoTurn for this.

If the wrong tools are used, human injury and damage to machinery and equipment can result!

Sequence of operations

You can terminate the setup dialog by selecting the "Continue" softkey.
8.4 Program handling

Sequence of operations

You can now check all program steps from the "Blank" definition to the "Finished part" geometry and "Setup" of the toolholder.
This menu is also the same as the screen layout after a specific workpiece has been loaded.

You can terminate your workpiece inputs by selecting the "End" softkey.

You should answer the inquiry:
"Save WOP program?"
with "Yes" and acknowledge the question
"Generate NC data?"
with "Yes".

If you answer the question:
"Transfer data to NC?" with "Yes", the workpiece you have created is selected for machining and can be started directly by means of NC start.
AutoTurn also offers you the option of editing all files. To do so, select the "Edit" softkey.

You have now programmed your new workpiece successfully and you are back in the AutoTurn main menu.
Your new workpiece PART5678 is now entered with date and time of creation in the part program list under the "Program handling" softkey.
Interactive Programming

9.1 Single-step machining ........................................................................................................ 9-170
  9.1.1 Introduction ........................................................................................................... 9-170
  9.1.2 Operation ............................................................................................................. 9-175

9.2 Macros .......................................................................................................................... 9-178
  9.2.1 General notes ........................................................................................................ 9-178
  9.2.2 Milling .................................................................................................................... 9-183
  9.2.3 Macro programming example .............................................................................. 9-185
9.1 Single-step machining

9.1.1 Introduction

Besides automatic programming, the operator can define his/her own programming steps by means of "interactive programming" and even define the entire machining process. You will need this programming method for complex workpieces. In some cases, an automatically generated program may not be capable of machining the entire contour. Some residual material is left over which must be machined by means of interactive programming.

The following is a typical error message:

"Error occurred: Remaining surface ..."

The error must be reset and the remaining surface eliminated by a single machining operation.

Two types of interactive programming are available:

- **Single-step machining:**
  Individual operations (e.g.: turning, plunge-cutting) are added and referenced to the programmed geometry.

- **Macros:**
  Machining operations are added without reference to the programmed geometry (e.g.: facing, milling).

Function overviews for single machining operations are given on the following pages. Please refer to Subsection 9.2.1 for the equivalent function overviews for macros!
Overview of Single-step machining functions

Turning menu

TURNING

External rough turning
- Rough turning longit. to spindle
- Rough turning longit. from spindle
- Turn contour/offset
- Rough turning face to left
- Rough turning face to right
- Rough face blank

External finish turning
- Finish turning to left
- Finish turning to right
- Turn contour offset to left
- Turn contour offset to right

Machining mode
>Automatic<
- Ext. 1. rough turning longit. (180°)
- Ext. 1. finish turning longit.

Internal rough turning
- Rough turning longit. to left
- Rough turning longit. to right
- Rough turning face to left top
- Rough turning face to left bottom

Internal finish turning
- Finish turning to left
- Finish turning to right
- Turn contour offset to left
- Turn contour offset to right

MMC 103  PCU 50
9.1 Single-step machining

Grooving menu

- GROOVING
  - Cutting off
    - Cutting off with receiving cradles
    - Cutting off with NC gantry
    - Cutting off/counterspindle pick up
    - Move forward with counterspindle, cutting off
  - External radial grooving
    - Rough grooving
      - Finish grooving to left and right
      - Finish grooving to left
      - Finish grooving to right
      - Parting off
  - Internal radial grooving
    - Rough grooving
      - Finish grooving to left and right
      - Finish grooving to left
  - External face grooving
    - Rough grooving
      - Finish grooving
      - External contour grooving upwards
      - External contour grooving downwards
      - Parting off
  - Internal face grooving
    - Rough grooving
      - Finish grooving
      - Internal contour grooving downwards
      - Internal contour grooving upwards
      - Parting off
Drilling menu

- DRILLING
  - Centrical to axis of rotation
    - Centering
      - Spot drilling on face
      - Drilling on face
      - Deep-hole drilling
      - Drilling with insert drill
      - Drilling, sinking, reaming
        - Thread tap on face
        - Sinking
        - Reaming
  - C axis
    - Face end
  - Face indexing plate
  - C axis
    - Peripheral surface
  - Machining mode
    - > Automatic<
      - Single spot drilling, Drilling, Twist dr, Upl
      - G17 Spot drilling, Drilling, Deep-hole, Thread tap
      - G19 Spot drilling, Drilling, Deep-hole, Thread tap
Thread menu

<table>
<thead>
<tr>
<th>THREAD</th>
<th>External thread turning right</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>External thread turning left</td>
</tr>
<tr>
<td></td>
<td>Internal thread turning</td>
</tr>
<tr>
<td></td>
<td>Threading tool</td>
</tr>
<tr>
<td></td>
<td>Tapping face</td>
</tr>
</tbody>
</table>

Milling menu

<table>
<thead>
<tr>
<th>MILLING 1)</th>
<th>Groove milling on face</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Pocket/island on face</td>
</tr>
<tr>
<td></td>
<td>Contour milling on face (Marked Points)</td>
</tr>
<tr>
<td></td>
<td>Groove milling peripheral surface</td>
</tr>
<tr>
<td></td>
<td>Pocket/island peripheral surface</td>
</tr>
<tr>
<td></td>
<td>Contour milling peripheral surface (Marked Points)</td>
</tr>
</tbody>
</table>

Non-cutting handling menu

<table>
<thead>
<tr>
<th>NON-CUTTING HANDLING/OTHER</th>
<th>Gripping/Link Bar in position</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Stock stop manual</td>
</tr>
<tr>
<td></td>
<td>Stock stop and feed-out</td>
</tr>
<tr>
<td></td>
<td>Gripping bar</td>
</tr>
<tr>
<td></td>
<td>Remove part from main spindle</td>
</tr>
<tr>
<td></td>
<td>Move part forward with counterspindle</td>
</tr>
<tr>
<td></td>
<td>Remove part from counterspindle, 2nd revolver</td>
</tr>
<tr>
<td></td>
<td>Remove part from counterspindle, NC gantry</td>
</tr>
<tr>
<td></td>
<td>Remove part from counterspindle manually</td>
</tr>
</tbody>
</table>

1) from SW 4
9.1.2 Operation

There are many methods by which you can program interactively! It is not possible to explain all these variants within the scope of this Guide. Operator guidance should be sufficient for our purposes.

Macro programming is described in Subsection 9.2.3 on the basis of an example!

Please refer to Chapter 17 for an explanation of the error messages relating to macro programming.

Sequence of operations

Starting in the menu in which you activate your automatic program generation function, select the "Single machining" softkey.

A selection of 6 machining operations will appear on the screen:

1. Turning
2. Grooving
3. Drilling
4. Thread
5. Milling
6. Non-cutting handling (other)
9.1 Single-step machining

Sequence of operations

If, for example, you select "Turning", a new window is opened:
1  External rough turning
2  External finish turning
3  Machining mode Automatic
4  Internal rough turning
5  Internal finish turning

When you select "Selection 1", the following screenform appears:
1  Rough turning longitudinal to spindle
2  Rough turning longitudinal from spindle
3  Turn contour offset
4  Rough turning face to left
5  Rough turning face to right
6  Rough facing blank

You want to continue working with "Selection 1"

and select the starting point on your contour with the cursor keys,

confirm your selection with "Starting point OK",

then define the end point,

and finally link contour start and contour end to the blank.
The example below shows how to join the contour to the blank.
Sequence of operations

Confirm the link to the blank you have selected by selecting the "Point OK" softkey. A tool is then offered.

You can press "OK" to accept this tool, choose a different one or enter a temporary tool with "Tool entry".

In a new display, you will now be presented with cutting values which you can also transfer to the system with "OK" since these are values which have been optimally calculated.

When the cutting values are entered, the single machining operation is activated in the contour section defined and color-highlighted above and then added as the last machining section to a part program already generated in an automatic run.

This additional single machining operation can easily be tracked in the simulation.

When you check the work plan now, you will find the programmed single machining and can shift it if necessary.

You can also program parts which are made up of single machining operations.

If you were to do so, however, you would not make use of the full range of practical experience integrated in the automatic program generation function of the AutoTurn Graphic Programming System!

Moreover, the risk of technological errors is increased through individual inputs and you will need more time to generate your program!

The remaining machining operations from "2 Grooving" to "6 Non-cutting handling" must be programmed in the same way as "1 Turning".

Since the screen windows are user-friendly and self-explanatory, a more detailed description of all the possible combinations will not be given at this point!
9.2 Macros

9.2.1 General notes

Overview of Macros functions

Turning/Grooving menus

As of SW 4, you can program machining operations for contours individually using the Single-step machining function.

Exception: – Free NC blocks
– Knurling
– Oil flute grooving
– Text rolling

TURNING

- Rough face blank
- Turn undercut
- Drill chamfer

GROOVING

- Cut-off
- Cut-off with receiving pan
- Cut-off with NC gantry
- Cut-off/pick up counterspindle ¹
- Move forward counterspindle/cut-off ¹
- Recessing and chamfer external
- Recessing and chamfer internal
- Groove prior to finish grooving
- Groove Form R
- Groove Form V

¹ from SW 4
Overview of Macros functions

Drilling menu

```
DRILLING
  Centrical to axis of rotation
    Centering
      Spot drilling on face
      Drilling with insert drill
      Drilling on face
      Deep hole drilling
      Tapping/sinking/reaming

  C axis, face
    Spot drilling on face
    Drilling on face
    Tapping on face
    Metric thread complete
    Step drilling

  C axis, radial, peripheral surface
    Spot drilling, radial
    Drilling, radial
    Thread tapping, radial
    Metric thread complete

  Locating
    Indexing plate
      Spot drilling on face
      Drilling on face
      Tapping on face

  Locating
    Indexing plate
      Radial, peripheral surface
      Spot drilling, radial
      Drilling, radial
      Tapping, radial
```
Milling/Non-cutting handling menus

MILLING
- Groove milling face
  - Peripheral surface milling
  - Milling indexing plate
    - Groove milling indexing plate G17
    - Groove milling indexing plate G19
- Groove milling
  - Milling single flat
- Groove milling
  - Milling single flat
  - Contour outside milling
  - Face cutting

NON-CUTTING HANDLING/OTHERS
- Bar in position
  - Stock stop, bar, manual
  - Stock stop, bar, feed-out
  - Grip bar
- Workpiece removal, handling
  - Part removal from main spindle with counterspindle
    - Move part forward on main spindle with counterspindle
    - Remove part from counterspindle with NC gantry
    - Remove part from counterspindle manually
- Tailstock, quill
  - Tailstock: Quill forward
  - Tailstock: Quill back
  - Tailstock: Tailstock forward
  - Tailstock: Tailstock back
  - Revolver: Quill forward
  - Revolver: Quill back
  - Counterspindle: Quill forward and back
- Non-cutting machining and others
  - Text rolling, radial
  - Text rolling, face
  - Knurling external
  - Oil flute grooving, internally
  - NC block

1) from SW 4
Sequence of operations

When you select the "Macros" softkey, a selection of 6 machining operations appears on the screen:

1. Turning
2. Grooving
3. Drilling
4. Thread
5. Milling
6. Non-cutting handling

If you wish to select "Turning", you can choose between:

1. Rough facing blank
2. Turning undercut and
3. Drilling chamfer.

When you select the "Selection 1" softkey, the window "Blank rough facing" is opened.
Check the proposed values
- Approach diameter \( D_1 \)
- Face diameter \( D_2 \)
- Machining allowance \( Z \)
and then transfer them to the system with "OK".
Sequence of operations

You will then be given a tool recommendation for this macro which you can assign to it by selecting "OK".

As with a single machining operation, you will be offered the calculated optimum cutting values in the next screenform: "Cutting data enter/complete" which you can, of course, change before you select "OK".

When you select this last "OK", the selected macro takes effect independently of any machining contour.

Please check this additional macro machining operation under "Simulation".

You will find that the macro has been added under "Machining plan".

Note about the macro "Groove rough grooving/finish grooving": This is a grooving operation performed once only, i.e. the grooving tool width must correspond to that of the groove itself. The grooving operation is performed without tool radius compensation.
9.2.2 Milling

If your machine tool is equipped with a driven tool or a position-controlled spindle (C axis), then the "Milling" macro will be of particular interest to you!

**Sequence of operations**

You can activate milling operations by selecting the "Selection 5" softkey.

A selection of 3 different milling options will appear on the screen:

1. Face milling
2. Peripheral milling
4. Milling – indexing plate

Press the "Selection 1" softkey to access the 5 machining modes:

1. Groove milling
2. Milling single flat
3. 2-edge surface milling – Milling indexing plate
4. Contour outside milling
5. Face milling

You want, for example, to mill an external **hexagonal contour**.

In the input fields of the opened screenform:

"Mill contour (polygon 3...nn_side)"

Enter your **example values** for a hexagonal contour:

- No. of corners: 6
- Distance A (Half width across flats): 80
- Infeed depth T: –2
- Approach S: 10
- Overlapping U: 5

And transfer them with "OK".
Sequence of operations

After you have entered the dimensions for the macro, you will be offered an end mill for the macro machining operation.

By selecting the "Tool entry" softkey, you can create a temporary workpiece

and then confirm your input with "OK".

The cutter material "HSS" is also transferred to the system with "OK".

In the cutting value window now open for the machining operation "C-AXIS MILL SURFACE G17" (mill surface on plane G17 with C axis) you can change the values for Cutting velocity V in m/min and Feedrate F in mm/rev.

The "OK" softkey stores the added macro so that you will now find it in the machining plan.
9.2.3 Macro programming example

Using the workpiece pictured below as an example, you will now learn how to add two machining macros “Milling” and “Drilling” to the existing workpiece after the program generation sequence in automatic programming mode.

Workpiece:  
Blank:  
Diameter 30 mm  
Length 40 mm  
Blank material: C 45

As of SW 4 you can program this machining operation as an end face contour and machine it with the automatic and single-step machining function.
Sequence of operations

According to Chapter 8.1.1 "Automatic Programming", the part shown is defined by the following inputs:

<table>
<thead>
<tr>
<th>Programming step</th>
<th>Inputs</th>
</tr>
</thead>
<tbody>
<tr>
<td>1. Program name</td>
<td>MACRO</td>
</tr>
<tr>
<td>2. Blank</td>
<td>Cylinder</td>
</tr>
<tr>
<td>3. Blank dimensions</td>
<td>Ø30 x 40</td>
</tr>
<tr>
<td>4. Material</td>
<td>C 45</td>
</tr>
<tr>
<td>5. Finished part dimension</td>
<td>30</td>
</tr>
<tr>
<td>6. Finished part position</td>
<td>To size: 1 mm</td>
</tr>
<tr>
<td>7. External geometry</td>
<td>Cylinder</td>
</tr>
<tr>
<td></td>
<td>Diameter: 28</td>
</tr>
<tr>
<td></td>
<td>Length: Residual length (~30)</td>
</tr>
<tr>
<td>8. Workholder selection</td>
<td>Optional</td>
</tr>
<tr>
<td>9. Tailstock</td>
<td>None</td>
</tr>
<tr>
<td>10. Machining limits</td>
<td>Check</td>
</tr>
<tr>
<td>11. Machining mode</td>
<td>Automatic</td>
</tr>
<tr>
<td>12. Simulation</td>
<td>All</td>
</tr>
</tbody>
</table>

Your next inputs will be to program the milling and drilling macros.
Sequence of operations

Select the "Macros" softkey in menu "Program/Machin. 1st side", enter the speed limit for the spindle and select the milling technology from the available macros

1  Turning
2  Grooving
3  Drilling
4  Thread
5  Milling
6  Non-cutting handling

by means of "Selection 5" softkey.

AutoTurn offers you the 3 milling types:

1  Face milling
2  Peripheral milling
3  Milling – indexing plate

The "Selection 1" softkey now provides access to 5 milling macros for "face milling":

1  Groove milling
2  Milling single flat
3  2-edge surface milling – Milling indexing plate
4  Contour outside milling
5  Face milling

You want to cut a square in the end face and therefore select "4".

AutoTurn displays a screen form with the parameterizing inputs for this end face machining operation. A number of input values are assigned defaults. Set your parameters according to the workpiece drawing:

No. of corners: 4 (input)
Distance A: 10 (input)
Infeed depth T: –8 (input)
Center point Xm: 0
Center point Zm: 0
Machining direction (1/2): 1
Approach path S: 10
Overlapping U: 5

and then confirm your milling operation inputs with "OK".
Sequence of operations

AutoTurn then requests you to specify
- a groove end mill,
- cutter materials and
- cutting values.

To simplify programming, you should store the preset values.

The milling macro is now programmed!

Now you need to program the symmetrical circle of holes.
Starting in the same menu "Program/Machin. 1st side", select the "Macros" softkey again

and then select the technology "Drilling" with "Selection 3".

You want to machine a symmetrical circle of holes in the end face (G17 plane) of your workpiece, so select "Selection 2".

For drilling operations on the end face, AutoTurn again offers you a wide range of options:
1. Spot drilling face
2. Drilling face
3. Thread tapping face
4. Metric thread complete
5. Step drilling

Select the end face drilling operation with "Selection 2".

AutoTurn now displays a screenform with parameterizing inputs to define the individual holes. A number of input values are assigned defaults.

Set the following parameters according to your drawing:

Drill_diameter D: 5 (input)
Reference length L: 0 (input)
Lead Z1: (no entry) (input)
Approach Z2: 2 (input)
Drilling depth T: 8 (input)

and then confirm your inputs with the "OK" softkey.
Sequence of operations

Once you have defined one hole, you only need to specify the pattern in which a number of holes are to be drilled on the end face.
AutoTurn offers you the following options for this purpose:

- 0   Use present circle of holes
- 1   Pitch circle division regular
- 2   Pitch circle division irregular
- 3   Coordinates

A circular division around the construction circle with a diameter of 17 mm is used for this workpiece.

Press "1" and confirm your input with "OK".

AutoTurn now provides you with a screenform with parameterizing inputs for the circle of holes:

- Name circle of holes: B1 (input)
- Pitch circle diam._D: 17 (input)
- No. of holes: 4 (input)
- Angle of 1st hole: 45 (input)

Confirm your circle of holes by selecting "OK".
Sequence of operations

The system then requests you to specify
- a twist drill,
- a cutter material and
- cutting data.

You can modify the default settings or transfer them to the system with "OK".

The workpiece is now fully defined with the additional milling operations and can, as we have already mentioned, now be simulated (see Section 8.2 Simulation).
The end face machining operations are shown in a separate simulation display.
Undefined Contour Elements

10.1 Introduction .............................................................................................................. 10-192
10.2 Programming example ............................................................................................... 10-193
10.1 Introduction

Taking the programming example described in Section 10.2, you can check the performance of the AutoTurn contour calculators (geometry processor).
For this purpose, you program a workpiece with 4 undefined contour elements, undefined and then use the geometry help provided by AutoTurn to resolve the contour.
10.2 Programming example

The contour definition of the workpiece pictured below consists of 2 cylinders and 4 undefined contour elements.

Workpiece: Blank: Diameter 20 mm
Length 30 mm
Blank material: C 45

The angle of 110° is calculated as follows in AutoTurn (relative to the machining direction):

AutoTurn angle = 180° + (90° – 20°) = 250°
## Sequence of operations

The workpiece shown in the diagram is defined by the following inputs according to Chapter 8.1.1 "Automatic programming":

### Programming step | Input
---|---
1. Program name | HELP
2. Blank | Cylinder
3. Blank dimensions | ∅20 x 30
4. Material | C 45
5. Finished part dimension | 20
6. Finished part position | To size: 1 mm

Your next inputs will be to program the contour of the workpiece as a combination of contour elements and a contour definition:

Start by selecting the "External" softkey.

The first contour element is a cylinder which is fully defined on the basis of its dimensions:

- Diameter: 14 (input)
- End length: −4.5 (input)

Select the "Chamfer right" softkey and define the chamfer by entering the following:

- Chamfer length (r): 1 (input)
- Angle: 45 (input)

Store the cylinder you have entered by selecting "OK".
Sequence of operations

Since the following contour is not defined, it must be programmed by means of a contour definition.

Start by selecting an upward straight line.

The starting point of this upward straight line is already defined by the cylinder.

AutoTurn requests you to enter the

Diameter: 17.6 (input).

Confirm this input by selecting the "OK" softkey and the element you have entered will appear on the screen.

All of the 4 following geometry elements are only partially defined. To calculate their resolution, use the geometry help function provided by AutoTurn.

The first element is a counterclockwise tangential circle with a radius of 1. To program it, select the following softkeys:

and enter:

Radius: 1 (input)
Diameter EP: (input) (no data)
Length EP: (input) "
End angle: (input) "

Confirm your inputs by selecting "OK".

The undefined circle you have entered cannot be displayed graphically, but is indicated in the top right-hand window by the symbol □.
Sequence of operations

The next element is a clockwise tangential circle linked to the first element with a radius of 5. Select the softkeys:

Circle

AutoTurn requests you to enter the following data which you can only partially specify:

Diameter CP: (input) (no data!)
Length CP: (input) "
Radius: 5 (input)
Diameter EP: (input) (no data!)
Length EP: (input) "
End angle: (input) "

Confirm your inputs by selecting "OK".

This circle cannot be displayed graphically either. However, it is displayed in the top right-hand window with the symbol.

The next element is another counterclockwise tangential circle with the radius 1.
You therefore select the softkeys:

Circle

AutoTurn will again request you to enter data which you can only partially specify:

Diameter: 17.6 (input)
Length CP: −15.65 (input)
Radius: 1 (input)
Diameter EP: (input) (no data!)
Length EP: (input) "
End angle: (input) "

Confirm your inputs by selecting "OK".

Since the undefined circle you have entered cannot be displayed graphically, it is represented by an icon on the screen.
Sequence of operations

The next element is an optional straight line.

You can only partially specify the data requested now by AutoTurn

- Diameter: 14 (input)
- End length: (input) (no data)
- Angle: 250 (input)

Confirm these inputs by selecting the "OK" softkey.

The undefined straight line cannot be displayed graphically, but is therefore symbolized.

Now try to resolve the undefined contour using the AutoTurn geometry help function:
Select the "Geometric help" softkey to activate this function.

The symbols in the upper section of the screen window are now displayed against a color background.
AutoTurn now requests data about the marked elements.
If you do not know the answer to the question, please just select the "Input" key:

- End angle: (input)
- End length: (input)
- Diameter: (input)

The system then requests the following data for the next element:

- Diameter auxiliary tangent bottom: (input)
- Diameter auxiliary tangent top: (input)
- Length auxiliary tangent left: (input)
- Length auxiliary tangent right: (input)
- Tangential at end? Y
Sequence of operations

The last step is to define the cylinder on the left:

You can define the cylinder without specifying numerical data by means of the softkeys "Link" and "Distance to go",

and then confirm your inputs with "OK".

Since the remaining programming procedure has already been described, it will not be described again here. When defining the machining limits, you should pay attention to the vertical machining limits. If the workpiece is to be machined on one side only, you should shift the vertical limit downwards to ensure that no sections of the contour are left unmachined.
Two-Sided Machining/Counterspindle

11.1 Introduction............................................................................................................... 11-200
11.2 Programming example: Machining on the main spindle (two-sided machining)........... 11-201
11.3 Programming example: Machining on the counterspindle ......................................... 11-208
   11.3.1 Special features of working with 2 toolholders............................................... 11-214
   11.3.2 Cut off and pull forward with counterspindle............................................... 11-215
11.4 Progr. ex.: Machining on the main spindle and counterspindle................................ 11-217
11.1 Introduction

With AutoTurn you can also machine workpieces from two sides on the main spindle. By defining the variable machining limits, you can determine up to what element the workpiece is machined in the first clamping (right-hand side of workpiece). The remainder of the machining operation is performed in the second clamping.

The sequence of operations required for this function is explained on the basis of a programming example in Section 11.2.
11.2 Programming example: Machining on the main spindle (two-sided machining)

The workpiece pictured below must be machined from 2 sides.

Workpiece:
- Blank:
  - Diameter: 85 mm
  - Length: 182 mm
- Blank material: C 60
Sequence of operations

The workpiece shown in the diagram is defined by the following inputs according to Chapter 8.1.1 "Automatic programming":

**Programming step** | **Input**
--- | ---
1. Program name | 2SIDES
2. Blank | Cylinder
3. Blank dimensions | ∅85 x 182
4. Material | C 60
5. Finished part dimension | 180
6. Finished part position | Centered
7. External geometry | According to drawing

After you have programmed the workpiece contour, you reach the two-sided machining menu by means of the softkeys "Back" and "Continue".

With "Side 1" or "Side 2" you define which side of the workpiece is to be machined first.

Answer the system inquiry "Machining 2nd side" by selecting the "Main spindle" softkey, thus confirming that the workpiece must be machined from two sides.

Then you are asked whether the workpiece is a cut-off part. Answer this question with "Continue", as "No" is the default.

Now define your workholder in the normal way. You may possibly be able to determine through your clamping data up to what element you wish the workpiece to be machined in the first clamping operation (see Subsection 7.1.5).
11.2 Programming example: Machining on the main spindle

Sequence of operations

By setting the machining limits for external rough turning and external finishing,

you can specify exactly up to which element you wish the workpiece to be machined in the first clamping operation.

By entering:

Outside rough: –120 (input)

Outside finish: –120 (input)

you position the machining limit behind the 2nd element (from right!).

You then need to start machining of the 1st side in the normal way by selecting the "Automatic" softkey.

You can then execute (as described in Chapters 8 and 9) the functions

• Simulation
• Machining plan
• Single machining operations
• Macros

for this first machining side.
Sequence of operations

You then start to program machining of the second side:

By selecting the "Continue" softkey, you automatically branch to the workholder selection for the 2nd machining side (and not to the setup dialog as you do for single-side machining operations).

Select the workholder for machining of the 2nd side and use the cursor keys to position the holder at the optimum position on the workpiece which has now been rotated through 180 degrees.

You can move the workholder:
- element by element
- in 1 mm steps
- in 5 mm steps
- in 100 mm steps.

Confirm the position you have selected by selecting the "Position OK" softkey. The workholder is then positioned on the first side which has already been machined.

Selection of the "Open workholder" softkey releases the chuck again.
### Sequence of operations

You now need to define the machining limits for the second side:

AutoTurn normally calculates the machining limits automatically, but you should check them anyway.

When you select the "Automatic" softkey, AutoTurn automatically generates the machining program for the 2nd workpiece side.

You can, however, define the machining operation yourself by using the interactive programming function.

AutoTurn then provides you with the following functions for the second machining side:
- Simulation
- Machining plan
- Single machining operations
- Macros.

If you now select the "Continue" softkey, you will go to the setup dialog for the first machining side

and, if you select it again, to the setup dialog for the second machining side.
Sequence of operations

After you have made all the necessary inputs in the setup dialogs for both machining sides, select the "Continue" softkey again to reach the next menu which will allow you to program functions which you already know, i.e.

- Blank (definition of blank)
- Finished part (definition of finished part)
- Setup (setup dialog)

as well as functions specific to two-side machining, i.e.

- Machining 1st side (machining of right-hand workpiece side)
- Machining 2nd side (machining of left-hand workpiece side).

Finish your programming inputs by selecting the "End" softkey, store the NC program and generate the NC data.

At this stage, the following programs are generated for two-side machining operations:

- MPF1001.MPF NC program 1st machining side
- MPF1002.MPF NC program 2nd machining side

The following are generated optionally (see machine adaptation):

- TOA11.SPF TOA data 1st machining side
- TOA22.SPF TOA data 2nd machining side
- TIME.DAT Time calculations
- TOOLS1.DAT Setting sheet 1st machining side
- TOOLS2.DAT Setting sheet 2nd machining side
Sequence of operations

If you answer the question "Transfer data to NC?" with "Yes", the workpiece program you have generated is selected for machining. You can then initiate a program run directly by means of NC start.

AutoTurn also allows you to edit all the files listed above in an editor via the "Edit" softkey. The selection menu which appears after you have selected the softkey includes all the files that have been generated.

Use the cursor keys to mark the file that you wish to edit and confirm your input by selecting "OK".

Please refer to the general Operator's Guide for the SINUMERIK 840D/810D system for instructions on how to use the editor.
11.3 Programming example: Machining on the counterspindle

The example only applies to AutoTurn Plus!

The workpiece pictured below must be machined from 2 sides.

Workpiece: Blank part: Diameter 85 mm
            Length 182 mm
Blank material: C 60

The 2nd side of the workpiece in Section 11.2 can also be machined on the counterspindle with AutoTurn Plus (SW 4.x and higher).

Sequence of operations

Operation is the same as for the workpiece used in Section 11.2.

After you have programmed the workpiece contour, you reach the two-sided machining menu by means of the softkeys "Back" and "Continue".

With "Side 1" and "Side 2" you define which side of the workpiece is to be machined first.
Answer the system inquiry "Machining 2nd side" by selecting the "Counterspindle" softkey, thus confirming that the workpiece must be machined from two sides.

Answer the system inquiry "Cut-off part (Y/N)" with "No".

Now define your workholder in the normal way. You may possibly be able to determine through your clamping data up to what element you wish the workpiece to be machined in the first clamping operation.

**Sequence of operations**

By setting the machining limits for external rough turning and external finish turning,

you can specify exactly up to which element you wish the workpiece to be machined in the first clamping operation.

By entering:

Outside rough: \(-120\) (input)

Outside finish: \(-120\) (input)

you position the machining limit behind the 2nd element (from right!).

You then need to start machining of the 1st side in the normal way by selecting the "Automatic" softkey.

Unlike machining on the main spindle, here you will be asked to define the workholder for the counterspindle when you have completed machining of the 1st side.

Handling is the same as for the main spindle, but mirrored. Select the workholder that you have defined for the counterspindle.
Sequence of operations

You then start to program machining of the second side:

By selecting the "Continue" softkey, you automatically branch to the workholder selection for the 2nd machining side (and not to the setup dialog as you do for single-side machining operations).

Select the workholder for the counterspindle and use the cursor keys to position the workholder at the optimum position on the workpiece. You can move the workholder
- element by element
- in 1 mm steps
- in 5 mm steps
- in 100 mm steps.

Confirm the position you have selected by selecting the "Position OK" softkey. The workholder is then positioned on the first side which has already been machined.

Selection of the "Open workholder" softkey releases the chuck again.
Sequence of operations

Mark the counterspindle
The workpiece is picked off by the counterspindle.
"Advance counterspindle/Remove part"
Here, you define a retraction position and a release position for the turret.

Travel against fixed stop is currently not supported (setting "N").

Set the option "Pick off positioned part" to "Y".
In the following form you enter an angular offset of 180°.
This angular offset is derived from the coordinate position used in the graphic programming.
When you have pressed "OK" and "Continue", you start machining on the counterspindle.

You now need to define the machining limits for the second side:
AutoTurn normally calculates the machining limits automatically, but you should check them anyway.

When you select the "Continue" and "Automatic" softkeys, AutoTurn automatically generates the machining program for the 2nd workpiece side.

You can, however, define the machining operation yourself by using the interactive programming function.

AutoTurn then provides you with the following functions for the second machining side:
- Simulation
- Machining plan
- Single machining operations
- Macros
Simulation starts for the counterspindle as a mirror image.

If you now select the "Continue" softkey, you will go to the setup dialog for the main spindle and counterspindle.

**Sequence of operations**

After you have made all the necessary inputs in the setup dialogs for both machining sides, select the "Continue" softkey again to reach the next menu which will allow you to program functions which you already know, i.e.

- Blank (definition of blank)
- Finished part (definition of finished part)
- Setup (setup dialog)

as well as functions specific to **two-side** machining, i.e.

- Machining 1st side (machining of right-hand workpiece side)
- Machining 2nd side (machining of left-hand workpiece side)

Conclude the programming with "End", save the NC program you have created and generate the NC data.

At this stage, the following programs are generated for two-side machining operations:

- MPF1001.MPF NC program 1st machining side
- MPF1002.MPF NC program 2nd machining side

The following are generated optionally (see machine adaptation):

- TOA11.SPF TOA data 1st machining side
- TOA22.SPF TOA data 2nd machining side
- TIME.DAT Time calculations
- TOOLS1.DAT Setting sheet 1st machining side
- TOOLS2.DAT Setting sheet 2nd machining side
Sequence of operations

If you answer the question "Transfer data to NC?" with "Yes", the workpiece program you have generated is selected for machining. You can then initiate a program run directly by means of NC start.

AutoTurn also allows you to edit all the files listed above in an editor via the "Edit" softkey. The selection menu which appears after you have selected the softkey includes all the files that have been generated.

Use the cursor keys to mark the file that you wish to edit and confirm your input by selecting "OK".

Please refer to the general Operator's Guide for the SINUMERIK 840D/810D system for instructions on how to use the editor.
11.3 Programming example: Machining on the counterspindle

11.3.1 Special features of working with 2 toolholders

If your system is preset for 2 toolholders (see Chapter 6), this results in the following differences from the procedure described in Section 11.3.

- When you first define machining of the 1st side, you define how the workpiece is to be removed from the counterspindle.
  (screenshot)
- When machining of the 2nd side started, the 2nd toolholder is also released.
  (cf. holder 1)

During simulation, the tool of the 2nd holder is shown in the mounting position in front of the turning center.

- When Automatic comes to an end, the synchronization table for toolholder 1 and toolholder 2 is displayed.
  The "Wait" markers automatically inserted for synchronization of the two operation sequences are listed here.
  With the "Synchronize" softkey, additional "Wait" markers can be set or existing markers deleted with "Synchronization off".

- The setup dialog is called when you select "Continue". Here, you can switch from the 1st to the 2nd toolholder with the "Toolholder 2" softkey.
  Once you have set up the tools, continue operation in the usual way (see Section 11.3.).
11.3.2 Cut off and pull forward with counterspindle

- Define a longer blank for the example workpiece (e.g. 200). Answer the query "Cut-off part? Y/N" in the finished part area with "Yes" and confirm the cut-off allowance AB with "OK".

- Select a collet as the workholder and position the finished part with T = 185.

- If you want to machine without a tailstock press the "Continue" softkey.

- The machining limits are maintained and confirmed with "Continue".

- When you start Automatic, the screenform "Pull bar into machining position" is displayed. The following options are available to you:
  0: Pull bar forward
  1: Pull bar forward manually to stop
  2: Use a bar feed
  3: Use a gripper
  and 4: Pull forward with counterspindle

Select "4", for example, (pull forward with counterspindle). Automatic mode runs. When the automatic run is complete, the screenform "Strategy for removing part on counterspindle side" is displayed.
• Remove part from counterspindle e.g. manually (option 1).

• Workholder selection for counter spindle as already described.

• Now the screenform for pull forward and cut-off appears: Here again, you select the option "Pick off positioned part" (Yes) and an angular offset of 180°.

• When you have selected the cut-off tool, machining of the 1st side is terminated and the parting operation is displayed.

• After that, you parameterize machining of the 2nd side in the usual way (see Section 11.3).
11.4 Progr. ex.: Machining on the main spindle and counterspindle

The example described in Section 7.2 is now machined in the main and counterspindle. The differences compared to conventional programming are identified and explained. Take the contour data from Section 7.2.

Sequence of operations

Please enter the following data to describe the finished part contour:

<table>
<thead>
<tr>
<th>Cylinder</th>
<th>Diameter:</th>
<th>End length:</th>
<th>Rounding</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>80</td>
<td>–62.5</td>
<td>10.0</td>
</tr>
<tr>
<td></td>
<td>120</td>
<td>–115</td>
<td></td>
</tr>
<tr>
<td></td>
<td>105</td>
<td>–140</td>
<td>7.5</td>
</tr>
<tr>
<td></td>
<td>95</td>
<td>–165</td>
<td></td>
</tr>
</tbody>
</table>

Press "Back" and "Continue" to go to the display of the 1st clamping and acknowledge proposed position 1 by selecting the "Continue" softkey.

When the display of the 2nd workpiece clamping appears on your screen, press the "Counterspindle" softkey and acknowledge "No cut-off part" by selecting the "Continue" softkey.
Sequence of operations

As described in Section 5.8 above, select the chuck for the 1st machining side, set the machining limits and leave AutoTurn to generate an optimum program for the first machining side.

While the NC program is being generated, the following system queries are displayed:
Reject "Pull bar into machining position" by entering: 0 and acknowledge default "(0) Turn contour in one run, spot drill first" in input menu "Ext. finish turning 1st machining side" by entering "OK".

In response to query "Strategy part removal on counterspindle side" please enter: 2 (by NC gantry).

The 1st machining operation on the workpiece is now displayed on your screen.
By selecting the "Continue" softkey, you can switch to the menu for selecting the counterspindle chuck.
11.4 Progr. ex.: Machining on the main spindle and counterspindle

Sequence of operations

You can now enter settings for the following parameters in the input display "Advance counterspindle/Pick up part":
- Activate 1st toolholder (behind turning center) to station T ...
- Retraction position Z
- Angular offset W (between main and counterspindles)
Pick-up position: ... is displayed.

Automatic programming of the 1st clamping is now completed.
You can now start processing the 2nd clamping.

Select the "Automatic" softkey.
Since you have decided to have parts removed from the counterspindle by means of an NC gantry, you now need to enter the following settings in display "Pick up part from counterspindle with gantry":
- Pick up position L
- Retraction from chuck A
- Activation of 2nd toolholder (in front of turning centre) to station T ...
- and retraction position Z
and confirm your inputs with "OK".

Automatic processing of the 2nd clamping is now complete.
Sequence of operations

When you select the "Continue" softkey, the operations executed with toolholders TH 1 and TH 2 are displayed under "Synchronization" together with the corresponding synchronization commands (WAIT markers).

You can of course modify the machining plans at this point if you wish to do so.

With SW 4 of AutoTurn, however, you also have the option of programming workpieces on a machine with a main spindle and counterspindle that has only one toolholder. If only one toolholder is installed, then only one NC program is generated for both spindle machining operations.

In other words, if only one toolholder is installed for both the main spindle and the counterspindle, then only one NC program is generated and there is no need to synchronize operations.
Link to CAD

12.1 Link to CAD systems (SW 4 and higher) ................................................................. 12-222
12.2 DXF boundary conditions (SW 4 and higher) ......................................................... 12-222
12.1 Link to CAD systems (SW 4 and higher)

The CAD link is used to process complex parts designed by the customer on CAD systems with AutoTurn. The CAD information must be stored in DXF format. AutoTurn reads in the DXF files and filters out contour-specific information such as dimensions and hatching etc. A blank and finished part description is then no longer required.

Sequence of operations

Procedure on the MMC 103 and PCU 50:
Go into the area "Services", "Manage data" and then copy the data from the diskette or via computer link into the buffer. These files are then displayed in the DXF window of AutoTurn. Once you have selected your file with "OK", a workpiece of the same name is created and an AutoTurn file (*.041) generated. Now you can process this workpiece in the normal way with AutoTurn.

12.2 DXF boundary conditions (SW 4 and higher)

In order to ensure correct conversion of DXF files to the finished part geometry of AutoTurn, it is important to follow the special conditions of DXF format. The basic structure of DXF files is described below.

Boundary conditions of DXF format for the generation of data for blank and finished part geometries in AutoTurn:

1. The blank and finished part must lie on different layers.
2. The finished part should be described in the 1st or 2nd quadrant, the drawing origin (X coordinate) is taken as the workpiece zero (X coordinate).
3. The contour must not contain any gaps.
4. The blank/finished part contour must not be covered by detail drawings or "enlargements".
5. The blank and finished part must not be drawn as one with workholders or connected to other parts.
6. Workpieces must be drawn in the correct position.
Sample Program: Sprocket Shaft

13.1 Creating a program ................................................................. 13-224
13.2 Definition blank/machined part................................................. 13-227
13.3 Creating the outside contour ..................................................... 13-228
13.4 Machining the end face ............................................................. 13-234
  13.4.1 Drilled holes ................................................................. 13-234
  13.4.2 Pocket ............................................................................ 13-236
13.5 Workholder selection .............................................................. 13-240
13.6 Setting the machining limits ..................................................... 13-241
13.7 Single machining operations ................................................... 13-242
  13.7.1 Rough face turning ......................................................... 13-243
  13.7.2 Rough turning longitudinal .............................................. 13-245
  13.7.3 Rough grooving recess .................................................... 13-247
  13.7.4 Finish grooving ............................................................... 13-249
  13.7.5 Drilling ............................................................................ 13-250
  13.7.6 Milling ............................................................................... 13-251
13.8 Machining plan ................................................................. 13-252
13.1 Creating a program

Prerequisites

- For this example to work properly, please set up the following tools prior to programming (see Chapter 2):

<table>
<thead>
<tr>
<th>ID number</th>
<th>ASTE2</th>
<th>SPI5.0ANGETR</th>
<th>SF6</th>
</tr>
</thead>
<tbody>
<tr>
<td>Tool group</td>
<td>External grooving tools</td>
<td>Drills</td>
<td>End mills</td>
</tr>
<tr>
<td>Name</td>
<td>Extern grooving tool 2</td>
<td>R twist drill</td>
<td>End mill</td>
</tr>
<tr>
<td>Cutter material</td>
<td>HM-U</td>
<td>HSS</td>
<td>HSS</td>
</tr>
<tr>
<td>Setting dimension X</td>
<td>–70</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Setting dimension Z</td>
<td>–110</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Holder diameter DH</td>
<td>55</td>
<td>95</td>
<td></td>
</tr>
<tr>
<td>Grooving tool width ST</td>
<td>2</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Radius I</td>
<td>0.2</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Collision dimens. K</td>
<td>90</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Diameter D1</td>
<td>5</td>
<td>6</td>
<td></td>
</tr>
<tr>
<td>Max. drilling depth</td>
<td>80</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Angle W1</td>
<td>118</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Transmission ratio</td>
<td>1</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Number of teeth</td>
<td></td>
<td></td>
<td>3</td>
</tr>
</tbody>
</table>

- Please use AutoTurn for one toolholder. If you have to use AutoTurn Plus, you must choose between toolholder 1 and toolholder 2 during single machining.

Requirement
In this example, you will learn some important functions of the AutoTurn Graphic Programming System:

- Create any blank
- Removing the stock remaining between blank and machined part
- Drilling on the end face
- Milling on the end face

**Sequence of operations**

You are in the main menu of AutoTurn (see Section 1.4). Press F1 to start programming.

Now enter the name of the program.
13.1 Creating a program

MMC 103  PCU 50
13.2 Definition blank/machined part

You must select the blank form first.

- Cylinder
  - Blank diameter: 95
  - Blank length: 150

Then select the desired blank material 16MnCr5 with the cursor keys.

After you have entered the machined part length, you must also define the position of the machined part in the blank.

Machined part length F: 100

Allowance A: 5
13.3 Creating the outside contour

External
Contour
Straight
line
Diameter 42

Start diameter: 42
End diameter: 48
End length:
Angle: 135

OK
Back Taper

Sample Program: Sprocket Shaft
13.3 Creating the outside contour

External
Contour
Straight
line
Diameter 42

Start diameter: 42
End diameter: 48
End length:
Angle: 135

OK
Back Taper

Autoturn CHAN1 Auto
Program aborted

External
Contour
Straight
line
Diameter 42

Start diameter: 42
End diameter: 48
End length:
Angle: 135

OK
Back Taper

Autoturn CHAN1 Auto
Program aborted
13.3 Creating the outside contour

Contour

Straight line

End length:

Line length:

OK

Circle

Diameter CP: 80

Length CP: –35

Radius: 23

Diameter EP: 60

Length EP:
13.3 Creating the outside contour

Selecting the starting point

As of Version 5.2.10, the starting point is no longer selected via the "Gray/Blue" softkey; rather via "1/2".
13.3 Creating the outside contour

Link
End diameter: 90
End length: -80
Rounding left
Radius (1): 4
OK

Contour
Straight line
End length: -100
OK
The outside contour is completed with a recess.

The recess is made in the form of a circular groove.
Adding a rounding at the start of a circle.

The outside contour is completed.
13.4 Machining the end face

13.4.1 Drilled holes

- **Name drilling pattern P B1**
- **Pitch circle diameter D 32**
- **Pitch circle center Xm 0**
- **Pitch circle center Ym 0**
- **Number of holes I 4**
- **Angle 1st hole W 0**

The 1st step is to program the holes.
You have now created the drilling pattern.

You can still perform the following actions:
- The F1 and F2 keys provide access to the coordinates of the individual holes.
- F5 resets the input of the drilling pattern to the 1st hole (x 16, y 0).
- F6 deletes the entries.

Then enter the data for the individual holes.

Hole diameter D 5
Chamfer width F 0
Chamfer angle W 0
Datum length L 0
Drilling depth T 10
Fit type dimension P

OK
13.4.2 Pocket

The 2nd step in end face machining is pocket programming.

To create the chamfer contour, you first define the starting point of the contour. The quadratic form of the pocket is defined by the single straight lines.
Sample Program: Sprocket Shaft

13.4 Machining the end face

Straight line

X: 11.5

Rounding

Radius (1): 4

OK

Straight line

Y: -11.5

Rounding

Radius (1): 4

OK
Sample Program: Sprocket Shaft 02.02

13.4 Machining the end face

Straight line

X: -11.5

Rounding

Radius (1): 4

OK

Straight line

Y: 11.5

Rounding

Radius (1): 4

OK
When you close the chamfer contour, the outside contour is completed.
13.5 Workholder selection

Now select the workholder.

Selecting the workholder.

The input mask for the tailstock is skipped.
13.6 Setting the machining limits

**Outside rough**

<table>
<thead>
<tr>
<th>Element</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Default values</td>
<td></td>
</tr>
<tr>
<td>External rough: ( -100.000 )</td>
<td></td>
</tr>
<tr>
<td>Position OK</td>
<td></td>
</tr>
</tbody>
</table>

**Outside finish**

| External finish: \( -100.000 \) |  |
| Position OK |  |

**Bore**

| Internal rough: \( -10 \) |  |
| Position OK |  |

| Internal finish |  |
| Internal finish: \( -10 \) |  |
| Position OK |  |

For general information about machining limits, please refer to Section 5.9. Positive values are reset to zero.
13.7 Single machining operations

After you have defined the machining limits, start single machining by specifying the speed limit.

Speed limit 3000

Pulling the bar forward: 0

You can accept the default value here.
13.7.1 Rough face turning

The end face is machined first.

Selecting the start and end point for machining.

Starting point OK

Endpoint OK
Connecting the start and end of the contour to the blank.

The first suitable tool for this type of machining appears. Use the F1/F2 keys to select further suitable tools. You can use F3 to select other suitable tools from other groups.

Choose the tool with the ID number PLANLA.

You can accept the default value (down) as the direction of cut.

You may, of course, change the cutting data later.

Rough facing machining is now completed. You can now modify the work steps, cutting velocities, allowance, etc. in the work plan (F5)!
13.7.2 Rough turning longitudinal

The workpiece is now machined parallel to the spindle. In this machining process, the recess is cut first, and then removed in the second step.

First, select the start and end points for machining.

- Element
  - D 42.000
    - Starting point OK
  - D 60.000
    - Skip
- Element
  - L 0.000
  - L –61.800
Then connect the start and end of the contour to the blank.

Now choose the tool with ID number **KOP35**.

You can accept the insertion angle and cutting data.
### 13.7.3 Rough grooving recess

Now, single machining is carried out for the recess.

**Single machining**

**Selection 2**

**Plunge-cutting**

**Selection 2**

**Plunge-cutting radial**

**Selection 1**

**Rough grooving**

**Allowance X 0.5**

**Allowance Z 0.5**

for radial cut: U1 0.2

... on slopes: U2 0.2

Increment infeed DU 0.5

OK
Now connect the start and end points of the contour to the blank.

To complete recess machining, please select the tool with the ID number ASTE2.
13.7.4 Finish grooving

Single-machining finish grooving is now carried out:
- Values for preprocessing L and D are accepted.
- The start and end points are adopted as for rough grooving.

The recess is skipped for finish grooving.

Then select the tool with the ID number ASTE2.
### 13.7 Single machining operations

#### 13.7.5 Drilling

The hole is machined.

You can use the F1 and F2 keys to choose the hole pattern.

Please press F8 to accept the data in the interactive screenform "Drilling with twist drill".

Now select the tool with the ID number **SPI5.0ANGETR**.

To finish hole machining, you now only have to edit or accept the cutting data.
13.7.6 Milling

To finish machining of the workpiece, you now only need to mill the pocket on the end face.

Use the F1 and F2 keys to select the milling object:

![Milling Selections]

- Selection 5: Milling
- Selection 1: Face milling
- Selection 4: Object rough-finished

Do you want to change/reduce the contour?
- No

You can accept or change the values displayed for "Rough and finish milling".

© Siemens AG, 2002. All rights reserved.
SINUMERIK 840D/840Di/810D AutoTurn Programming/Setup (AUP) – 02.02 Edition

13-251
Select the tool **SF6**.

You can accept or change the values for the cutting data. Machining is now completed.

### 13.8 Machining plan

The individual work steps are displayed here:

<table>
<thead>
<tr>
<th>Operation</th>
<th>Tool ID no.</th>
<th>V</th>
<th>F</th>
</tr>
</thead>
<tbody>
<tr>
<td>EXTERNAL BUSH FACETING</td>
<td>PBS526</td>
<td>230.0</td>
<td>0.400</td>
</tr>
<tr>
<td>EXTERNAL BUSH TURING</td>
<td>PBS527</td>
<td>230.0</td>
<td>0.400</td>
</tr>
<tr>
<td>EXTERNAL BUSH GROOVING</td>
<td>AP754</td>
<td>80.0</td>
<td>0.100</td>
</tr>
<tr>
<td>EXTERNAL BUSH GROOVING</td>
<td>AP711C</td>
<td>100.0</td>
<td>0.080</td>
</tr>
<tr>
<td>EXTERNAL BUSH GROOVING</td>
<td>SP51 G50000</td>
<td>24.0</td>
<td>300.0</td>
</tr>
<tr>
<td>EXTERNAL BUSH GROOVING</td>
<td>SF6</td>
<td>24.0</td>
<td>0.017</td>
</tr>
</tbody>
</table>

Changes can still be made. However, care must be taken!
Adaptation to Machine

14.1 Overview ................................................................................................................... 14-254
  14.1.1 Description ........................................................................................................... 14-254
  14.1.2 Operation ............................................................................................................. 14-255

14.2 Settings ................................................................................................................... 14-256
  14.2.1 Description ........................................................................................................... 14-256
  14.2.2 Operation ............................................................................................................. 14-256

14.3 NC words................................................................................................................... 14-260
  14.3.1 Description ........................................................................................................... 14-260
  14.3.2 Operation ............................................................................................................. 14-263

14.4 NC blocks.................................................................................................................. 14-265
  14.4.1 Description ........................................................................................................... 14-265
  14.4.2 Operation ............................................................................................................. 14-267

14.5 Time calculation ........................................................................................................ 14-269
  14.5.1 Description ........................................................................................................... 14-269
  14.5.2 Operation ............................................................................................................. 14-270

14.6 System ..................................................................................................................... 14-271
  14.6.1 Description ........................................................................................................... 14-271
  14.6.2 Operation ............................................................................................................. 14-272
14.1 Overview

14.1.1 Description

Function

With the "Adaptation to Machine" function, the universal AutoTurn Graphic Programming System installed is adapted to your specific machine by parameterization.

Only the machine setter or instructed personnel are authorized to make changes in the "Adaptation to Machine" menu!

Changes to this data always change the technological conditions!

Further notes

The "Setting" softkey is always available to the user!

The access rights and therefore also the display of the 4 softkeys "NC words", "NC blocks", "Time calculation" and "System" of the adaptation to the machine are defined by the machine manufacturer in the "System" menu (see Section 14.6)!

Therefore, if one of the softkeys described below does not appear on your screen, the machine manufacturer has disabled this function!

Nevertheless, this Chapter describes all functions for adaptation to the machine.
14.1.2 Operation

Sequence of operations

You are in the basic AutoTurn menu and press the "Etc." key.

After you have pressed the "Machine adaptation" softkey the "AutoTurn machine adaptation" menu is displayed.

Setting  NC words  NC blocks  Time  System calculation
14.2 Settings

14.2.1 Description

Function

You define the following in the "Settings" menu:

- The output format of your NC programs (e.g.: system of units, block number),
- What data can be output as a file (e.g.: TOA data, setup sheet),
- The type of tool change, speed limitations, clamping pressures,
- The allowances for your workpiece and
- Work area limitations.

14.2.2 Operation

Sequence of operations

You are in the "Machine adaptation" menu and press the "Settings" softkey.

In the opened window of the active vertical "Output" softkey

you can select the following with the cursor keys and the next input block key:

- Output system: "metric" or "inch" (for PC only)
  On the machine, the current setting that is set in the machine data is shown.
- Output resolution: "mm" or "inch" or "inch high-precision"
  ("inch" has one decimal place and "inch high-precision" two decimal places more than the specifications in the NC words)
- Threading cycle: Yes/No
Sequence of operations

Enter the following on the numeric keypad:

Block numbering:  
Start: 10  
Max. block number: 99999  
Increment: 5

You can change or accept these default values.  
The block numbering must have increments large enough to allow 
for the subsequent insertion of NC blocks!

Pressing the vertical "Files" softkey activates the second window of 
the "Setting" menu.

Here you can select your setting using the cursor keys 
and the next input block key:

- TOA data: Yes/No  
  If "Yes", the tool data are generated as a subprogram (SPF) 
  and automatically activated by the main program MPF.

- Time calculation: Yes/No  
  If "Yes", the time that each tool needs for machining is 
  calculated for you.

- Setup sheet: Yes/No  
  The setup sheet (which can be printed out) contains data for 
  setting up the machine for the programmed workpiece (zero 
  offset, tool changing point, tools used etc.).
14.2 Settings

Sequence of operations

If you press the "Machine" softkey, the system suggests the setpoint values for:
- The chuck height of the main spindle,
- The speed limitations of the main spindle and the powered tool,
- The clamping pressure of the main spindle,
- The starting point for a fixed tool change
- or an optimized tool change, the starting point being based on the largest tool and
- a tool search in the tool file only
- or a tool search in the toolholder first and then in the tool file.

Under the "Allowances" softkey, you can enter the allowances for rough turning and rough grooving for inside and outside machining and for the parting (cut-off) of the finished part. AutoTurn takes account of these dimensions in the X and the Z axis during NC program generation. With zero allowance the contour entered is already machined precisely during roughing.

The upper and lower working area limitations in the X and Z axes displayed if you press the "Working area" softkey must be set such that the software limit positions set in the machine data are not exceeded and must match the working zone limitations that are possibly already set in the setting data!

You can save data areas here with AutoTurn by selecting them with the Toggle key. When you press the "Save" softkey, the data are stored in the "Autoturn.dir" directory and can be saved from there via the Clipboard of the data management system onto diskette or via RS232C. Data are loaded in reverse order (RS232C/diskette → Clipboard → Autoturn.dir → "Load" softkey).

Data backups under AutoTurn are stored in the path C:\dh\autoturn.dir\name.dir on the MMC 103 and PCU 50.
Please refer to the information in the upgrade guide (Siemens.txt) when loading data from older software versions.

You store the data with the "OK" softkey.

The "Abort" softkey switches back to the initial display of the machine adaptation without storing the data.
14.3 NC words

14.3.1 Description

Note

The NC words have already been adapted to your machine by the machine manufacturer. It is possible that this function might only be accessible to the manufacturer of your machine!

Function

"NC words" are permanently defined terms in the AutoTurn Graphic Programming System that stand for certain functions. NC blocks are subsequently formed from these NC words. In Chapter 15 you will find a detailed list of all NC words, each with an explanation!

The specifications of the NC words are independent of the dimension system.

The input window for the NC words has the following structure:

<table>
<thead>
<tr>
<th>NC word</th>
<th>Word description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Address letter:</td>
<td>Modal:</td>
</tr>
<tr>
<td>Minimum value:</td>
<td></td>
</tr>
<tr>
<td>Maximum value:</td>
<td></td>
</tr>
<tr>
<td>Smallest increment:</td>
<td></td>
</tr>
<tr>
<td>Integer places:</td>
<td>Function group:</td>
</tr>
<tr>
<td>Decimal places:</td>
<td>Text after number:</td>
</tr>
<tr>
<td>Output leading zeros:</td>
<td></td>
</tr>
<tr>
<td>Output following zeros:</td>
<td>Calculation values:</td>
</tr>
<tr>
<td>Output + sign:</td>
<td>Constants:</td>
</tr>
<tr>
<td>Output – sign:</td>
<td>Factor:</td>
</tr>
<tr>
<td>Output decimal characters:</td>
<td></td>
</tr>
</tbody>
</table>
Explanations

Modality

Modal functions are only output if the output value has changed. However, after a main block (NC word "NM") even modal functions are output again.

Function group

G functions can be grouped together. When an NC word is output the modality of the other NC words in the same function group is reset until the next output. That means that the other NC words are output even if their output value has not changed. In this way, it is possible to control the output of G functions that have a mutual influence on one another.

Example:

CS Constant cutting speed and
CSOFF Deselect constant cutting speed are in the same function group.

The output value of CS is 200. CS is modal. After output of CSOFF the constant cutting speed CS can be output again in a following block although the output value 200 has not changed.

If no group or function group "zero" has been entered, no check is made.

Value calculation

The output value can be influenced by a factor and a constant:

Output value = Factor * Value + Constant

If the value "zero" is entered for "constant" and "factor" only the address letter of the NC word is output.
Parameters

If the output system has been set to "metric or inch" in the "Setting" menu, the input unit is converted to the output unit. If the output unit is "inch", one decimal place is added to the NC code output and if it is "inch high-precision" two decimal places are added.

Up to 30 characters can be entered in these fields.

All NC words already contain default values that you can accept.

Example 1:  GF  Current preparatory function

Address letter:  G
Minimum value:  0
Maximum value:  3
Smallest increment:  1
Leading zeros:  Yes
Modal:  Yes

Example 2:  A1  1st axis horizontal

Address letter:  Z
Minimum value:  –999
Maximum value:  999
Smallest increment:  0.001

Note

The content of the fields with a light (white or yellow) background can be changed or activated/deactivated.

If you want to change the values offered to you, please observe the conditions stated in the Programming Guide!

Values with a dark (gray) background are display values and cannot be altered!
14.3.2 Operation

**Sequence of operations**

After you have pressed the "Etc." key in the AutoTurn initial menu, press these softkeys in the sequence shown:

(if NC words are enabled, see Subsection 14.1.1!) and you will obtain an overview of all NC words.

The left-hand column of the table shows the short designation and the right-hand column the assigned word description.

You can jump from word to word using either the cursor keys and the paging keys, or the alphabetic keyboard to search for a word:

The letter entered marks the NC word to be searched for.

The input window is opened with the "Edit word" softkey.

Alternatively, you can use the "Input" key.

You select a field, enter your machine-specific values and confirm the input with "OK".
Sequence of operations

This input confirmation stores all data of the NC word edited and you return to the table of NC words.

If you press the "Another word" softkey in the window enter, you switch to the NC word table with automatic storage of the "previous" word.

With "Next word" the content of the next word is displayed after automatic storage.

With "Abort" you switch back to the higher-level menu "NC words" without storing the changed data of the NC words.
14.4 NC blocks

14.4.1 Description

The NC blocks have already been adapted to your machine by the machine manufacturer in the same way as the NC words (see Subsection 14.3.1).
The manufacturer of your machine can disable the "NC blocks" function for the machine operator.

Function

"NC blocks" are made up of the "NC words" defined in the AutoTurn Graphic Programming System. In Chapter 16 you will find a detailed list of all NC blocks, each with an explanation!

The input window of the NC blocks has the following structure:

NC block | Block description
--- | ---
{Blocks made up of NC words}
Monitor
(The number 9 is a placeholder!)
(Blocks in NC plaintext)

Explanations

Main block/modality
After a main block, modal functions (otherwise function only on value change) are output again.
Code: NM

Fixed/powered tools
You can have fixed and powered tools together in the NC blocks.
The adaptation to the machine takes account of which spindle is active for output.

Any texts
Any texts are enclosed in quotation marks "...".
They are written directly into the part program in this format. There is no syntax check.
NC blocks describe certain recurring function sequences on your machine.

Example 1: TRET Tool retraction

NM.";(TRET)",
N,CPOS,
N,CSOFF,
N,GF,"D0 G53",TNC,TC2,TC1,COOL,GCOOL

Monitor
;99999 ;(TRET)
N99999 C-999.999
N99999 G95
N99999 G9 D0 G53 G99 X-999.999 Z-999.999 M9 M9

Example 2: DELAY Delay

N,SSP,SPDL,SSPD,SPDLD,GSSP,GSPDL,GSPDLD;
N,COOL,GCOOL,
N, DT

Monitor
N99999 S9999 M9 S3=9999 M3=9 S2=9999 M2=9 M3=9
N99999 M9 M9
N99999 G04 F9999

Example 3: THR Thread cutting

N,THR,A1,PIT1,SPDL

Monitor
N99999 G33 Z-9999.999 K=–999.999 M9

In the plain text underneath "Monitor" you can see the following in the NC words:

- The address letter (G, Z, X, K)
- The output format (sign, integer positions, decimal point, decimal positions) and
- The text after the number
Note

The NC blocks generated are tested and optimized combinations of NC words. Of course, you can make changes to the block structure but you must have sound knowledge of the technology used!

14.4.2 Operation

Sequence of operations

Press the following softkeys from the AutoTurn initial menu once you have pressed the "Etc." key.

(if NC blocks have been enabled, see Subsection 14.1.1!) and the list of all NC blocks appears on the screen with their description.

You can see the content of one of these NC blocks after you have selected it using the cursor or paging keys

or using the alphabetic keyboard (the letter entered indicates the NC block searched for)

by pressing the "Edit block" softkey.

Alternatively, you can use the "Input" softkey.

The upper portion of the screen contains the block structure from the NC words described in Section 14.3 and the lower portion (monitor) the corresponding NC plain text.
**Sequence of operations**

Please make changes in the NC blocks at the blinking cursor position and confirm your input with the "OK" softkey.

In this way, all data of the NC block edited are stored and you can return to the table of NC blocks.

The "Another block" softkey switches back to the NC block table and stores your "previous" NC block.

With "Next block" the content of the next block up in the table is displayed after automatic storage.

With the "Abort" softkey you switch back to the menu "NC blocks" without storage of the changed NC block.

**Error handling**

If you violate the prescribed syntax while inputting NC blocks, the position with the error is highlighted with a blue background. The cursor cannot be moved before the error is remedied! In this case it is recommended to discard the changes and start again. Under "Monitor" you can see the NC blocks up to the position of the error.
14.5 Time calculation

14.5.1 Description

Function

The AutoTurn Graphic Programming System performs calculations of the machining times (active workpiece machining) and downtimes (e.g.: tool change).

The total time for machining includes not only the optimum cutting speeds and feedrates calculated by the system but also the rapid traverses and accelerations of the participating axes and the tool change times as machine values.

You can also enter machining times as free values using any combination of characters (e.g. M commands) to assign times to your machining functions.
14.5.2 Operation

Sequence of operations

You are in the AutoTurn initial menu and, after pressing the "Etc." key, you press the softkeys "Machine adaptation" and then "Time calculation".

Under the active "Machine values" softkey, you will see the values suggested by AutoTurn for:
- The rapid traverse rates of axes X, Z and C,
- The acceleration rates of axes X, Z and C
- The tool change.

You select a field and enter your machine-specific data.

Please check that the values entered for rapid traverse and accelerations of the axes match the values stored in the NC machine data (see Installation Instructions or Installation Lists)! Only if these data match can you expect precise time calculations from AutoTurn that correspond to the real machining times! It is also necessary to enter the tool change times actually measured!

Sequence of operations

Under the "Free values" softkey, you can assign times to any string. These strings could be, for example, machine or auxiliary functions, subroutines. In this function, you can enter, delete or change values.
14.6 System

14.6.1 Description

Function

If the "System" softkey, which has the highest priority, can be selected, you can control the following AutoTurn system settings for your machine tool:

- The access rights for the softkey functions:
  - Settings
  - NC words
  - NC blocks
  - Time calculation
  - System
  Priorities from 1 to 5 can be assigned, number 1 being the highest priority and number 5 the lowest. Equal priorities for several functions are possible. "System" must always have the highest, "Settings" the lowest priority.

- The turret indexing in ascending order or optimized for the shortest path.

- The turret stations in steps of two over 8 to 24 tool locations. Real turrets with fewer than eight locations are converted to a turret with eight locations.
14.6.2 Operation

**Sequence of operations**

From the AutoTurn initial menu you select the following in the order specified: "Etc." key,

"Machine adaptation" and

the "System" softkey.

The "Next input block" key takes you from one input field to the next.

You can assign access rights from 1 to 5 (see Subsection 14.6.1).

The fields for toolholder position and turret indexing must be activated/deactivated using the cursor keys.

Finally, confirm the system settings with "OK".
NC Words

15.1 Formal word description .................................................................................................. 15-274
  15.1.1 Information about formal words .............................................................................. 15-274
  15.1.2 Coordinate computation ....................................................................................... 15-275

15.2 Functions for turning .................................................................................................... 15-276
  15.2.1 General functions ................................................................................................. 15-276
  15.2.2 Tool change ......................................................................................................... 15-277
  15.2.3 Spindle and feedrate ............................................................................................ 15-278
  15.2.4 Thread .................................................................................................................. 15-280
  15.2.5 Insertion texts ...................................................................................................... 15-280

15.3 Special functions for C-axis machining ....................................................................... 15-281

15.4 Special functions with counterspindle (SW 4 and higher) ........................................... 15-282
15.1 Formal word description

15.1.1 Information about formal words

This chapter gives you a clear description of the NC words that are put together to make up the NC blocks during automatic generation by the AutoTurn Graphic Programming System.

The table below is obtained by selecting the **program number** with the "Edit word" softkey from the list of NC words.

### Parameters

<table>
<thead>
<tr>
<th>Formal word description</th>
<th>Formal word description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Address letter</td>
<td>MPF</td>
</tr>
<tr>
<td>Minimum value</td>
<td>1</td>
</tr>
<tr>
<td>Maximum value</td>
<td>999</td>
</tr>
<tr>
<td>Smallest increment</td>
<td>1</td>
</tr>
<tr>
<td>Integer places</td>
<td>3</td>
</tr>
<tr>
<td>Decimal places</td>
<td>0</td>
</tr>
<tr>
<td>Function group</td>
<td></td>
</tr>
<tr>
<td>Text after number</td>
<td>;Machine1</td>
</tr>
</tbody>
</table>

Calculating values

(Value = Value * Factor + Constant)

| Constant   | 0   |
| Factor     | 1   |

Output leading zeros

Output following zeros

Output decimal places

Output + sign

Output – sign

Modal
15.1.2 Coordinate computation

Function

AutoTurn works with the zero offset TRANS (formal word ZOP). The values for the offset (ZOPA2, ZOPA1) are obtained from the workholder and finished part geometry.
### 15.2 Functions for turning

#### 15.2.1 General functions

<table>
<thead>
<tr>
<th>Word</th>
<th>Description</th>
<th>Control</th>
</tr>
</thead>
<tbody>
<tr>
<td>PN</td>
<td>Program number</td>
<td></td>
</tr>
<tr>
<td>SPN</td>
<td>Subprogram number</td>
<td></td>
</tr>
<tr>
<td>PPD</td>
<td>Part program designation (PARTNO)</td>
<td>MSG(&quot;...&quot;)</td>
</tr>
<tr>
<td>PS</td>
<td>Program stop</td>
<td>M0</td>
</tr>
<tr>
<td>PE</td>
<td>End of program</td>
<td>M30</td>
</tr>
<tr>
<td>N</td>
<td>NC block number</td>
<td>N</td>
</tr>
<tr>
<td>NM</td>
<td>NC main block number (all active functions output)</td>
<td>:</td>
</tr>
<tr>
<td>GF</td>
<td>Current preparatory function G function (G00, G01, G02, G03)</td>
<td>G</td>
</tr>
<tr>
<td>A1</td>
<td>1st axis (Z) horizontal; for peripheral surface (Z)</td>
<td>Z</td>
</tr>
<tr>
<td>A2</td>
<td>2nd axis (X) vertical</td>
<td>X</td>
</tr>
<tr>
<td>A1IP</td>
<td>Interpolation parameter for axis 1</td>
<td>K</td>
</tr>
<tr>
<td>A2IP</td>
<td>Interpolation parameter for axis 2</td>
<td>I</td>
</tr>
<tr>
<td>A1_O</td>
<td>Old position coordinate 1st axis</td>
<td>Z</td>
</tr>
<tr>
<td>A2_O</td>
<td>Old position coordinate 2nd axis</td>
<td>X</td>
</tr>
<tr>
<td>FROM1</td>
<td>Start coordinate 1st axis</td>
<td>Z</td>
</tr>
<tr>
<td>FROM2</td>
<td>Start coordinate 2nd axis</td>
<td>X</td>
</tr>
<tr>
<td>SECTEXT</td>
<td>Text of machining section</td>
<td>MSG(&quot;...&quot;)</td>
</tr>
<tr>
<td>ZOS</td>
<td>Settable zero offset</td>
<td>G</td>
</tr>
</tbody>
</table>

Settable zero offset from AutoTurn only 2 ZOs possible (G54/55),
(Constant=53, Factor=1)
Value passed=1: 1st ZO
Value passed=2: 2nd ZO
### 15.2 Functions for turning

<table>
<thead>
<tr>
<th>Word</th>
<th>Description</th>
<th>Control</th>
</tr>
</thead>
<tbody>
<tr>
<td>ZOO</td>
<td>Zero offset Off</td>
<td>G53</td>
</tr>
<tr>
<td>ZOP</td>
<td>Programmed zero offset</td>
<td>ATRANS</td>
</tr>
<tr>
<td>ZOPA1</td>
<td>Programmed zero offset for 1st axis Z</td>
<td>Z</td>
</tr>
<tr>
<td>ZOPA2</td>
<td>Programmed zero offset for 2nd axis X</td>
<td>X</td>
</tr>
<tr>
<td>DIMS</td>
<td>System of units (metric/inch)</td>
<td>G</td>
</tr>
<tr>
<td></td>
<td>Value assignment from INI file</td>
<td></td>
</tr>
<tr>
<td></td>
<td>(Constant=70, Factor=1)</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Value passed=1: inch</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Value passed=0: metric</td>
<td></td>
</tr>
<tr>
<td>LIMA1</td>
<td>Working range limit axis 1</td>
<td>G26Z</td>
</tr>
<tr>
<td>LIMA2</td>
<td>Working range limit axis 2</td>
<td>G26X</td>
</tr>
</tbody>
</table>

### 15.2.2 Tool change

<table>
<thead>
<tr>
<th>Word</th>
<th>Description</th>
<th>Control</th>
</tr>
</thead>
<tbody>
<tr>
<td>TN</td>
<td>Tool number</td>
<td>T</td>
</tr>
<tr>
<td>TO</td>
<td>Tool offset number (D number)</td>
<td>D</td>
</tr>
<tr>
<td>TOR</td>
<td>Tool offset R</td>
<td></td>
</tr>
<tr>
<td>TN1</td>
<td>1st tool number</td>
<td>T</td>
</tr>
<tr>
<td>TN_P</td>
<td>Preceding tool number</td>
<td>T</td>
</tr>
<tr>
<td>TN_N</td>
<td>Next tool number</td>
<td>T</td>
</tr>
<tr>
<td>TC</td>
<td>Tool change (M06)</td>
<td>M06</td>
</tr>
<tr>
<td>TLOC</td>
<td>Magazine location</td>
<td>T</td>
</tr>
</tbody>
</table>

Tool change point: including the longest tool and the protection zone in the calculation

| TC1    | Tool change coordinate axis 1                         | Z       |
| TC2    | Tool change coordinate axis 2                         | X       |

Tool dimensions:

| TXS    | XS dimension                                          |         |
| TYS    | YS dimension                                          |         |
| TRS    | RS dimension (tool nose radius)                       |         |
| TNAME  | Tool name                                             | :NAME   |
| TNAME_P| Name of previous tool                                 | :NAME   |
| TNAME_A| Name of current tool                                  | :NAME   |
| TNAME_N| Name of next tool                                     | :NAME   |
### 15.2.3 Spindle and feedrate

<table>
<thead>
<tr>
<th>Word</th>
<th>Description</th>
<th>Control</th>
</tr>
</thead>
<tbody>
<tr>
<td>FF</td>
<td>G function for feedrate (G94/G95) mm/min or mm/rev.</td>
<td>G</td>
</tr>
<tr>
<td></td>
<td>(Constant=0, Factor=1)</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Value passed=94: mm/min</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Value passed=95: mm/rev</td>
<td></td>
</tr>
<tr>
<td>PATH</td>
<td>Exact stop/continuous-path mode G60/G64</td>
<td>G</td>
</tr>
<tr>
<td></td>
<td>(Constant=60, Factor=4)</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Value passed=0: exact stop</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Value passed=1: continuous-path mode</td>
<td></td>
</tr>
<tr>
<td>CD</td>
<td>Corner deceleration</td>
<td>G9</td>
</tr>
<tr>
<td>FEED</td>
<td>Feedrate in output unit</td>
<td>F</td>
</tr>
<tr>
<td>SPDL</td>
<td>Function for spindle direction (M3...5)</td>
<td>M</td>
</tr>
<tr>
<td></td>
<td>Value passed=3: right</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Value passed=4: left</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Value passed=5: top</td>
<td></td>
</tr>
<tr>
<td>SPDLD</td>
<td>Function for spindle direction of drive unit (3...5)</td>
<td>M2=</td>
</tr>
<tr>
<td>SSP</td>
<td>Spindle rpm main spindle</td>
<td>S</td>
</tr>
<tr>
<td>SSPD</td>
<td>Spindle rpm drive unit</td>
<td>S2=</td>
</tr>
<tr>
<td>LS</td>
<td>Main spindle limit speed to constant cutting rate</td>
<td>LIMS=</td>
</tr>
<tr>
<td>LSD</td>
<td>Drive unit limit speed to constant cutting rate</td>
<td>LIMS2=</td>
</tr>
<tr>
<td>CS</td>
<td>Constant cutting rate of main spindle</td>
<td>G96 S</td>
</tr>
<tr>
<td>CSD</td>
<td>Constant cutting rate of drive unit</td>
<td>G96 S2=</td>
</tr>
<tr>
<td>CSOFF</td>
<td>Deactivate constant cutting rate</td>
<td>G97</td>
</tr>
<tr>
<td>CSOFFD</td>
<td>Deactivate constant cutting rate of drive unit</td>
<td>G97</td>
</tr>
<tr>
<td>AOS</td>
<td>Angle for oriented spindle</td>
<td>SPOS</td>
</tr>
<tr>
<td>SPCL</td>
<td>Clamp/release spindle (M12/M13) (Constant=13, Factor=-1)</td>
<td>M</td>
</tr>
<tr>
<td></td>
<td>Value passed=0: release</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Value passed=1: clamp</td>
<td></td>
</tr>
<tr>
<td>Word</td>
<td>Description</td>
<td>Control</td>
</tr>
<tr>
<td>------</td>
<td>-------------</td>
<td>---------</td>
</tr>
<tr>
<td>TNC</td>
<td>Tool nose radius compensation Off/On (G40/41/42)</td>
<td>G</td>
</tr>
<tr>
<td></td>
<td>Value passed=40: Off</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Value passed=41: left</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Value passed=42: right</td>
<td></td>
</tr>
<tr>
<td>CRC</td>
<td>Cutter radius path compensation Off/On (G40/41/42)</td>
<td>G</td>
</tr>
<tr>
<td></td>
<td>Value passed=40: Off</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Value passed=41: left</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Value passed=42: right</td>
<td></td>
</tr>
<tr>
<td>REF</td>
<td>Reference point approach by program</td>
<td>G74</td>
</tr>
<tr>
<td>COOL</td>
<td>Coolant On/Off (M8/9) (Constant=9, Factor=−1)</td>
<td>M</td>
</tr>
<tr>
<td></td>
<td>Value passed=0: off</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Value passed=1: on</td>
<td></td>
</tr>
<tr>
<td>DT</td>
<td>Dwell time</td>
<td>G04 F</td>
</tr>
<tr>
<td>TFB</td>
<td>Tailstock forward/backward (M64/65) (Constant=65, Factor=−1)</td>
<td>M</td>
</tr>
<tr>
<td></td>
<td>Value passed=0: backward</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Value passed=1: forward</td>
<td></td>
</tr>
<tr>
<td>TOC</td>
<td>Tailstock open/close (M69/68) (Constant=69, Factor=−1)</td>
<td>M</td>
</tr>
<tr>
<td></td>
<td>Value passed=0: open</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Value passed=1: closed</td>
<td></td>
</tr>
<tr>
<td>TSPOS</td>
<td>Tailstock position</td>
<td></td>
</tr>
<tr>
<td>TSH</td>
<td>Tailstock height tailstock point</td>
<td></td>
</tr>
<tr>
<td>DON</td>
<td>Drive on/off</td>
<td>M</td>
</tr>
<tr>
<td></td>
<td>Value passed=0: off</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Value passed=1: on</td>
<td></td>
</tr>
<tr>
<td>RCP</td>
<td>Receiving pan forward/backward (M60/61) (Constant=61, Factor=−1)</td>
<td>M</td>
</tr>
<tr>
<td></td>
<td>Value passed=0: backward</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Value passed=1: forward</td>
<td></td>
</tr>
<tr>
<td>GS</td>
<td>Gear stages (M40...45)</td>
<td></td>
</tr>
<tr>
<td>(not yet available)</td>
<td>Value passed=1: stage 1</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Value passed=2: stage 2</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Value passed=3: stage 3</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Value passed=4: stage 4</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Value passed=5: stage 5</td>
<td></td>
</tr>
</tbody>
</table>
### 15.2 Functions for turning

<table>
<thead>
<tr>
<th>Word</th>
<th>Description</th>
<th>Control</th>
</tr>
</thead>
<tbody>
<tr>
<td>CLAMP</td>
<td>Clamping open/closed (M69/68)</td>
<td>M</td>
</tr>
<tr>
<td></td>
<td>(Constant=69, Factor=−1)</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Value passed=0: open</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Value passed=1: closed</td>
<td></td>
</tr>
<tr>
<td>BARON</td>
<td>Bar advance on</td>
<td>M</td>
</tr>
<tr>
<td>BAROFF</td>
<td>Bar advance off</td>
<td>M</td>
</tr>
<tr>
<td>PTSWPOS</td>
<td>Gantry swing-in position</td>
<td>Z</td>
</tr>
<tr>
<td>PTTOPOS</td>
<td>Gantry pick-up position</td>
<td>Z</td>
</tr>
<tr>
<td>PTCRYFREE</td>
<td>Gantry retraction path</td>
<td>Z</td>
</tr>
<tr>
<td>PTTRWPOS</td>
<td>Gantry eject position</td>
<td>Z</td>
</tr>
</tbody>
</table>

#### 15.2.4 Thread

<table>
<thead>
<tr>
<th>Word</th>
<th>Description</th>
<th>Control</th>
</tr>
</thead>
<tbody>
<tr>
<td>THR</td>
<td>Thread cutting</td>
<td>G33</td>
</tr>
<tr>
<td>TAPS</td>
<td>Tapping with floating tapholders</td>
<td>G63</td>
</tr>
<tr>
<td>PIT1</td>
<td>Thread lead axis 1 (Z)</td>
<td>K</td>
</tr>
<tr>
<td>PIT2</td>
<td>Thread lead axis 2 (X)</td>
<td>I</td>
</tr>
<tr>
<td>TAPR</td>
<td>Rigid tapping</td>
<td>G331/G332</td>
</tr>
</tbody>
</table>

#### 15.2.5 Insertion texts

<table>
<thead>
<tr>
<th>Word</th>
<th>Description</th>
<th>Control</th>
</tr>
</thead>
<tbody>
<tr>
<td>CT</td>
<td>Comment text</td>
<td>MSG(&quot;...&quot;)</td>
</tr>
<tr>
<td>IT</td>
<td>Insertion text (direct output)</td>
<td>...</td>
</tr>
</tbody>
</table>
### 15.3 Special functions for C-axis machining

<table>
<thead>
<tr>
<th>Word</th>
<th>Description</th>
<th>Control</th>
</tr>
</thead>
<tbody>
<tr>
<td>CON</td>
<td>Rotary axis on</td>
<td></td>
</tr>
<tr>
<td>COFF</td>
<td>Rotary axis off</td>
<td></td>
</tr>
<tr>
<td>CPOS</td>
<td>Angle of rotary axis</td>
<td>C</td>
</tr>
<tr>
<td>COP</td>
<td>Couple/decouple powered tools</td>
<td></td>
</tr>
<tr>
<td></td>
<td><strong>Linear axis:</strong></td>
<td></td>
</tr>
<tr>
<td>FA1</td>
<td>Horizontal axis end face</td>
<td>X</td>
</tr>
<tr>
<td>FA2</td>
<td>Vertical axis end face</td>
<td>Y</td>
</tr>
<tr>
<td>FA3</td>
<td>Infeed axis end face</td>
<td>Z</td>
</tr>
<tr>
<td>FA1IP</td>
<td>Interpolation parameter for horizontal axis end face</td>
<td>I</td>
</tr>
<tr>
<td>FA2IP</td>
<td>Interpolation parameter for vertical axis end face</td>
<td>J</td>
</tr>
<tr>
<td>TRMON</td>
<td>Transmit on</td>
<td>TRANSMIT</td>
</tr>
<tr>
<td>TRMOFF</td>
<td>Transmit off</td>
<td>TRAOFF</td>
</tr>
<tr>
<td></td>
<td><strong>Rotary axis:</strong></td>
<td></td>
</tr>
<tr>
<td>FRA1</td>
<td>Horizontal axis end face rotary axis</td>
<td>X</td>
</tr>
<tr>
<td></td>
<td><strong>Peripheral surface:</strong></td>
<td></td>
</tr>
<tr>
<td>PA1</td>
<td>Rotary axis peripheral surface</td>
<td></td>
</tr>
<tr>
<td>PA2</td>
<td>Longitudinal axis peripheral surface</td>
<td></td>
</tr>
<tr>
<td>PA3</td>
<td>Infeed axis peripheral surface</td>
<td></td>
</tr>
<tr>
<td>PC</td>
<td>Circle setting for peripheral surface</td>
<td></td>
</tr>
<tr>
<td>PCR</td>
<td>Reference radius for peripheral surface</td>
<td></td>
</tr>
<tr>
<td>MFA1</td>
<td>Milling: Rotary axis peripheral surface</td>
<td>Y</td>
</tr>
<tr>
<td>MA1IP</td>
<td>Peripheral surface: Interpolation parameter for axis 1</td>
<td>J</td>
</tr>
<tr>
<td>MA2IP</td>
<td>Peripheral surface: Interpolation parameter for axis 2</td>
<td>K</td>
</tr>
</tbody>
</table>
### 15.4 Special functions with counterspindle (SW 4 and higher)

<table>
<thead>
<tr>
<th><strong>Word</strong></th>
<th><strong>Description</strong></th>
<th><strong>Control</strong></th>
</tr>
</thead>
<tbody>
<tr>
<td>WAIT</td>
<td>Synchronization marker</td>
<td></td>
</tr>
<tr>
<td>GTC</td>
<td>CSP: Tool change</td>
<td>M3=M06</td>
</tr>
<tr>
<td>GSPDL</td>
<td>CSP: Function for spindle direction</td>
<td>M3=</td>
</tr>
<tr>
<td>GSPDLD</td>
<td>CSP: Function for spindle direction drive direction</td>
<td>M4=</td>
</tr>
<tr>
<td>GSSP</td>
<td>CSP: Spindle speed</td>
<td>S3=</td>
</tr>
<tr>
<td>GSSPD</td>
<td>CSP: Spindle speed for powered tool</td>
<td>S4=</td>
</tr>
<tr>
<td>GLS</td>
<td>CSP: Limit speed to constant cutting rate</td>
<td>LIMS=</td>
</tr>
<tr>
<td>GLSD</td>
<td>CSP: Limit speed to constant cutting rate of drive unit</td>
<td>LIMS=</td>
</tr>
<tr>
<td>GCS</td>
<td>CSP: Constant cutting rate</td>
<td>G96 S</td>
</tr>
<tr>
<td>GCSD</td>
<td>CSP: Constant cutting rate of drive unit</td>
<td>G96 S</td>
</tr>
<tr>
<td>GCSOFF</td>
<td>CSP: Deactivate constant cutting rate</td>
<td>G97</td>
</tr>
<tr>
<td>GCSOFFD</td>
<td>CSP: Deactivate constant cutting rate of drive unit</td>
<td>G97</td>
</tr>
<tr>
<td>GSPCL</td>
<td>CSP: Clamp/release spindle</td>
<td>M</td>
</tr>
<tr>
<td>GCOP</td>
<td>CSP: Couple/decouple powered tool</td>
<td>M</td>
</tr>
<tr>
<td>GCLAMP</td>
<td>CSP: Workholder open/closed</td>
<td>M</td>
</tr>
<tr>
<td>GHWINK</td>
<td>CSP: Angular offset between counterspindle and main spindle</td>
<td></td>
</tr>
<tr>
<td>GFUMASS</td>
<td>CSP: Chuck length of counterspindle</td>
<td></td>
</tr>
<tr>
<td>GSPAPOS</td>
<td>CSP: Pick-up position</td>
<td></td>
</tr>
<tr>
<td>GANFPOS</td>
<td>CSP: Switchover position</td>
<td></td>
</tr>
<tr>
<td>GSRPOS</td>
<td>CSP: Clamping position</td>
<td>Q1=</td>
</tr>
<tr>
<td>GSVORL</td>
<td>CSP: Advance</td>
<td>Q1=</td>
</tr>
<tr>
<td>GSHOEH</td>
<td>CSP: Height to tip</td>
<td></td>
</tr>
<tr>
<td>GVORLEN</td>
<td>CSP: Re-grip/pull workpiece (length)</td>
<td></td>
</tr>
<tr>
<td>GVORFUNK</td>
<td>CSP: Re-grip/pull workpiece (function)</td>
<td></td>
</tr>
<tr>
<td>GSCPOS</td>
<td>CSP: Rotary axis peripheral surface</td>
<td></td>
</tr>
</tbody>
</table>
NC Blocks

16.1 Formal block description ................................................................. 16-284
16.2 NC blocks with examples ............................................................... 16-285
  16.2.1 General NC blocks ................................................................. 16-285
  16.2.2 Tool change ......................................................................... 16-286
  16.2.3 Normal turning ................................................................. 16-289
  16.2.4 C axis ................................................................................. 16-292
  16.2.5 Counterspindle ................................................................. 16-295
16.3 Assignment between machining operations and blocks called ........ 16-301
16.1 Formal block description

- Each formal block is a separate macro and can comprise several lines.
- Each begins with a block number N or main block number NM.
- Entries in quotation marks "..." are written unchanged into the part program (direct output).
### 16.2 NC blocks with examples

The examples are provided exclusively for illustration purposes. Some of the blocks on your machine may be different.

#### 16.2.1 General NC blocks

<table>
<thead>
<tr>
<th>Formal block name</th>
<th>Formal block content (example)</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Beginning of program:</strong></td>
<td>PB1</td>
</tr>
<tr>
<td><strong>Beginning of program 2nd clamping:</strong></td>
<td>PB2</td>
</tr>
<tr>
<td><strong>End of program:</strong></td>
<td>PE1</td>
</tr>
<tr>
<td><strong>End of program 2nd clamping:</strong></td>
<td>PE2</td>
</tr>
<tr>
<td><strong>Program stop:</strong></td>
<td>PS</td>
</tr>
<tr>
<td><strong>Direct output of NC blocks:</strong></td>
<td>DO</td>
</tr>
</tbody>
</table>
### 16.2.2 Tool change

<table>
<thead>
<tr>
<th>Formal block name</th>
<th>Formal block content (example)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Tool retraction:</td>
<td></td>
</tr>
<tr>
<td>TRET</td>
<td>NM,&quot;;(TRET)&quot;,</td>
</tr>
<tr>
<td></td>
<td>N,CPOS,</td>
</tr>
<tr>
<td></td>
<td>N,CSOFF,SPDLD,SSPD,GSPDL,GSPDLD,</td>
</tr>
<tr>
<td></td>
<td>N,GF,&quot;DO G53&quot;,TNC,TC2,TC1,COOL,GCOOL</td>
</tr>
<tr>
<td>1st tool change</td>
<td></td>
</tr>
<tr>
<td>non-powered tool:</td>
<td></td>
</tr>
<tr>
<td>TC1_N</td>
<td>N,&quot;;(TC1_N)&quot;,</td>
</tr>
<tr>
<td></td>
<td>NM,TN,TO,COOL,TNAME,</td>
</tr>
<tr>
<td></td>
<td>N,&quot;;&quot;,TN_N,</td>
</tr>
<tr>
<td></td>
<td>N,CS,GCS,</td>
</tr>
<tr>
<td></td>
<td>N,SSP,SPDL,GSPDL,GSPDLD</td>
</tr>
<tr>
<td>1st tool change</td>
<td></td>
</tr>
<tr>
<td>powered tool:</td>
<td></td>
</tr>
<tr>
<td>TC1_D</td>
<td>N,&quot;;(TC1_D&quot;,</td>
</tr>
<tr>
<td></td>
<td>NM,TN,TO,COOL,TNAME,</td>
</tr>
<tr>
<td></td>
<td>N,&quot;;&quot;,TN_N,</td>
</tr>
<tr>
<td></td>
<td>N,CSD,GCSD,</td>
</tr>
<tr>
<td></td>
<td>N,SSPD,SPDLD,GSPDL,GSPDLD</td>
</tr>
<tr>
<td>nth tool change</td>
<td></td>
</tr>
<tr>
<td>pwrd/non-pwrd tool:</td>
<td></td>
</tr>
<tr>
<td>TCN_NN</td>
<td>N,&quot;;(TCN_NN)&quot;,</td>
</tr>
<tr>
<td></td>
<td>N,CSOFF,</td>
</tr>
<tr>
<td></td>
<td>NM,TN,TO,COOL,TNAME,</td>
</tr>
<tr>
<td></td>
<td>N,&quot;;&quot;,TN_P,TN_N</td>
</tr>
<tr>
<td></td>
<td>N,CS,GCS,</td>
</tr>
<tr>
<td></td>
<td>N,SSP,SPDL,GSPDL,GSPDLD</td>
</tr>
</tbody>
</table>
### Formal block name

<table>
<thead>
<tr>
<th>nth tool change</th>
<th>Formal block content (example)</th>
</tr>
</thead>
<tbody>
<tr>
<td>pwr. - pwr. tool:</td>
<td>TCN_DD</td>
</tr>
<tr>
<td></td>
<td>N,&quot;;(TCN_DD)&quot;</td>
</tr>
<tr>
<td></td>
<td>N,TRMOFF,</td>
</tr>
<tr>
<td></td>
<td>N,CPOS,</td>
</tr>
<tr>
<td></td>
<td>NM,TN,TO,COOL,GCOLL,TNAME,</td>
</tr>
<tr>
<td></td>
<td>N,&quot;;&quot;,TN_P,TN_N,</td>
</tr>
<tr>
<td></td>
<td>N,CSD,GCSD,</td>
</tr>
<tr>
<td></td>
<td>N,SSPD,SPDLD,GSPDL,GSPDLD</td>
</tr>
<tr>
<td>pwr./ non-pwr. tool:</td>
<td>TCN_DN</td>
</tr>
<tr>
<td></td>
<td>N,&quot;;(TCN_DN)&quot;</td>
</tr>
<tr>
<td></td>
<td>N,TRMOFF,</td>
</tr>
<tr>
<td></td>
<td>N,CPOS,</td>
</tr>
<tr>
<td></td>
<td>NM,TN,TO,COOL,GCOOL,TNAME,</td>
</tr>
<tr>
<td></td>
<td>N,&quot;;&quot;,TN_P,TN_N,</td>
</tr>
<tr>
<td></td>
<td>N,&quot;;SWITCH TO MAIN SPINDLE&quot;;</td>
</tr>
<tr>
<td></td>
<td>N,&quot;SETMS&quot;;</td>
</tr>
<tr>
<td></td>
<td>N,CS,GCSD,</td>
</tr>
<tr>
<td></td>
<td>N,SSP,SPDL</td>
</tr>
<tr>
<td>non-pwr./ pwr. tool:</td>
<td>TCN_ND</td>
</tr>
<tr>
<td></td>
<td>N,&quot;;(TCN_ND)&quot;</td>
</tr>
<tr>
<td></td>
<td>N,CSOFF,GCOSOFF,</td>
</tr>
<tr>
<td></td>
<td>NM,TN,TO,COOL,TNAME,</td>
</tr>
<tr>
<td></td>
<td>N,&quot;;&quot;,TN_P,TN_N,</td>
</tr>
<tr>
<td></td>
<td>N,&quot;;SWITCH TO TOOL DRIVE&quot;;</td>
</tr>
<tr>
<td></td>
<td>N,CSD,GCSD;</td>
</tr>
<tr>
<td></td>
<td>N,SSPD,SPDLD,GSPDL,GSPDLD</td>
</tr>
</tbody>
</table>
**Formal block name**  |  **Formal block content (example)**
--- | ---
Last tool change  |  
non-pwrd./non-pwrd. tool:  |  
**TCLAST_NN**  |  N,";(TCLAST_NN)"
N,CSOFF,GCSOFF,
NM,TN,TO,GCOOL,COOL,TNAME,
N,";",TN_P,TN1,
N,CS,GCS,
N,SSP,SPDL,GSPDL,GSPDL

Last tool change  |  
pwrd./pwrd. tool:  |  
**TCLAST_DD**  |  N,";(TCLAST_DD)"
N,TRMOFF,
N,CPOS,
NM,TN,TO,GCOOL,COOL,TNAME,
N,";",TN_P,TN1,
N,CSD,GCSD,
N,SSPD,SPDL,GSPDL,GSPDL

Last tool change  |  
pwrd./non-pwrd. tool:  |  
**TCLAST_DN**  |  N,";(TCLAST_DN)"
N,TRMOFF,
N,SPDL,
NM,TN,TO,COOL,GCOOL,TNAME,
N,";",TN_P,TN1,
N,";SWITCH TO MAIN SPINDLE"
N,"SETMS",
N,CS,GCS,
N,SSP,SPDL

Last tool change  |  
non-pwrd./pwrd. tool:  |  
**TCLAST_ND**  |  N,";(TCLAST_ND)"
N,CSOFF,GCSOFF,
NM,TN,TO,COOL,GCOOL,TNAME,
N,";",TN_P,TN1,
N,";SWITCH TO TOOL DRIVE"
N,CSD,GCS,
N,SSPD,SPDL,GSPDL,GSPDL
16.2.3 Normal turning

<table>
<thead>
<tr>
<th>Formal block name</th>
<th>Formal block content (example)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Programmable zero offset:</td>
<td>Programmable zero offset:</td>
</tr>
<tr>
<td>ZOP</td>
<td>N,ZOP,ZOPA2,ZOPA1</td>
</tr>
<tr>
<td>Motion at rapid traverse:</td>
<td>Motion at rapid traverse:</td>
</tr>
<tr>
<td>RAP</td>
<td>N,CS,SSP,SPDL,GCS,GSSP,GSPDL, N,CSOFF,SSP,GCOSOFF,GSSP, N,GF,PATH,TNC,TO,A2,A1,COOL,GCOOL</td>
</tr>
<tr>
<td>Motion at feedrate:</td>
<td>Motion at feedrate:</td>
</tr>
<tr>
<td>FED</td>
<td>N,CS,SSP,SPDL,GCS,GSSP,GSPDL, N,CSOFF,SSP,GCOSOFF,GSSP, N,GF,PATH,TNC,TO,FF,CD,A2,A1,FEED</td>
</tr>
<tr>
<td>Circular motion:</td>
<td>Circular motion:</td>
</tr>
<tr>
<td>FEDC</td>
<td>N, GF,TO,A2,A1,A2IP,A1IP,FEED</td>
</tr>
<tr>
<td>Dwell time:</td>
<td>Dwell time:</td>
</tr>
<tr>
<td>DELAY</td>
<td>N,SSP,SPDL,SSPD,SPDLD,GSSP,GSPDL,GSPDLD, N,COOL,GCOOL, N,DT</td>
</tr>
<tr>
<td>Speed, direction of spindle rotation:</td>
<td>Speed, direction of spindle rotation:</td>
</tr>
<tr>
<td>SSD</td>
<td>N,CS,SSP,SPDL,GCS,GSSP,&quot;(SPINDLE)&quot;, N,CSOFF,SSP,GCOSOFF,GSSP</td>
</tr>
<tr>
<td>Speed, direction of spindle rotation drive setting system:</td>
<td>Speed, direction of spindle rotation drive setting system:</td>
</tr>
<tr>
<td>SSDD</td>
<td>N,CSD,SSPD,SPDLD,GCSD,GSSPD,&quot;;SPINDLE PWRD.&quot;, N,SSPD,GSSPD</td>
</tr>
<tr>
<td>Spindle stop:</td>
<td>Spindle stop:</td>
</tr>
<tr>
<td>SST</td>
<td>N,SPDL,SPDLD,SSP,SSPD,GSPDL,GSPDLD,GSSP,GSSPD</td>
</tr>
</tbody>
</table>
### Formal block name | Formal block content (example)
--- | ---
Angle of rotary axis: CPOS | N,GF,CPOS,GSCPOS,A2,A1
Orient main spindle: SPA | N,COFF,
N,"SETMS",
N,AOS
Thread cutting: THR | N,";THREAD CUTTING (THR)",
N,THR,A1,A2,PIT1,PIT2,SPDL
Thread cutting with position-controlled spindle: TAPR | N,";(TAPR)",
N,AOS,
N,TAPR,A1,A2,PIT1,PIT2,SSPD,GSSPD
Tapping with compensating chucks: TAPS | N,";TAPPING WITH COMPENSATING CHUCK (TAPS)",
N,GF,FF,SSP,SSPD,SPDL,SPDLD,GSSP,GSSPD,
GSPDL,GSPDLD,GSPDLD
Tailstock: TAILST | N,TFB,TOC
Tailstock forward: TAILSTFW | N,SPDL,SPDLD,GSPDL,GSPDLD,
N,COOL,
N,"FETCH_TAILSTOCK",
N,TPOS,
N,TSH,
N,"PARK_TAILSTOCK",
N,"M15",
N,"G95"
<table>
<thead>
<tr>
<th>Formal block name</th>
<th>Formal block content (example)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Receiving pan:</td>
<td>N,RCP,&quot;;(RCP)&quot;</td>
</tr>
<tr>
<td>Clamping:</td>
<td>&quot;:(CLAMP)&quot;, N,CLAMP,GCLAMP</td>
</tr>
<tr>
<td>Gantry backward:</td>
<td>N,&quot;;(PTBK)&quot;, N,&quot;PART_OUT&quot;</td>
</tr>
<tr>
<td>Clamp/release spindle:</td>
<td>N,SPCL,GSPCL</td>
</tr>
<tr>
<td>Bar feed:</td>
<td>N,BARON,BAROFF,&quot;;(BAR)&quot;</td>
</tr>
</tbody>
</table>
## 16.2.4 C axis

<table>
<thead>
<tr>
<th>Formal block name</th>
<th>Formal block content (example)</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Rotary axis On:</strong></td>
<td></td>
</tr>
<tr>
<td>CON</td>
<td>N,&quot;;ROTORARY AXIS ON (CON)&quot; ,</td>
</tr>
<tr>
<td></td>
<td>N,&quot;G94&quot; ,</td>
</tr>
<tr>
<td></td>
<td>N,&quot;SPOS=0&quot;</td>
</tr>
<tr>
<td><strong>Rotary axis Off:</strong></td>
<td></td>
</tr>
<tr>
<td>COF</td>
<td>N,&quot;;ROTORARY AXIS OFF (COF)&quot; ,</td>
</tr>
<tr>
<td></td>
<td>N,SPDL,SPDLD,GSPDL,GSPDLD,&quot;G95&quot;</td>
</tr>
<tr>
<td><strong>Angle peripheral surface:</strong></td>
<td></td>
</tr>
<tr>
<td>PSA</td>
<td>N,GF,CPOS</td>
</tr>
<tr>
<td><strong>Angle end face:</strong></td>
<td></td>
</tr>
<tr>
<td>FA</td>
<td>N,GF,CPOS</td>
</tr>
<tr>
<td><strong>Motion on end face TA with rapid traverse:</strong></td>
<td></td>
</tr>
<tr>
<td>FRAP</td>
<td>N,CSD,SSPD,SPDL,GCSD,GSSPD,GSPDL,</td>
</tr>
<tr>
<td></td>
<td>N,CSOFFD,SSPD,GCSOFFD,GSSPD,</td>
</tr>
<tr>
<td></td>
<td>N,GF,PATH,CRC,TO,FA1,FA2,FA3,COOL,GCOOL</td>
</tr>
<tr>
<td>with feedrate:</td>
<td></td>
</tr>
<tr>
<td>FFED</td>
<td>N,CSD,SSPD,SPDL,GCSD,GSSPD,GSPDL,</td>
</tr>
<tr>
<td></td>
<td>N,CSOFFD,SSPD,GCSOFFD,GSSP,</td>
</tr>
<tr>
<td></td>
<td>N,GF,PATH,CRC,TO,FF,&quot;G9&quot;,Fa1,FA2,FA3,FEED,COOL,GCOOL</td>
</tr>
<tr>
<td><strong>Circular motion:</strong></td>
<td></td>
</tr>
<tr>
<td>FFEDC</td>
<td>N,GF,CRC,&quot;G9&quot;,TO,FA1,FA2,FA3,FA1IP,FA2IP,FEED</td>
</tr>
<tr>
<td><strong>Motion to end face RA with rapid traverse:</strong></td>
<td></td>
</tr>
<tr>
<td>FRRAP</td>
<td>N,CSD,SSPD,SPDLD,GCSD,GSSPD,GSPDLD,</td>
</tr>
<tr>
<td></td>
<td>N,CSOFFD,SSPD,GCSOFFD,GSSPD,</td>
</tr>
<tr>
<td></td>
<td>N,GF,PATH,CRC,TO,FRA1,CPOS,FA3,COOL,GCOOL</td>
</tr>
<tr>
<td>with feedrate:</td>
<td></td>
</tr>
<tr>
<td>FRFEED</td>
<td>N,CSD,SSPD,SPDLD,GCSD,GSSPD,GSPDLD,</td>
</tr>
<tr>
<td></td>
<td>N,CSOFFD,SSPD,GCSOFFD,GSSPD,</td>
</tr>
<tr>
<td></td>
<td>N,GF,PATH,CRC,TO,FF,CD,FRA1,CPOS,FA3,FEED</td>
</tr>
<tr>
<td><strong>Circular motion:</strong></td>
<td></td>
</tr>
<tr>
<td>FRFEEDC</td>
<td>N,GF,CRC,TO,FRA1,CPOS,FA3,FA1IP,FA2IP,FEED</td>
</tr>
<tr>
<td>Formal block name</td>
<td>Formal block content (example)</td>
</tr>
<tr>
<td>------------------</td>
<td>--------------------------------</td>
</tr>
<tr>
<td>Motion on peripheral surface with rapid traverse:</td>
<td></td>
</tr>
<tr>
<td><strong>PRAP</strong></td>
<td>N,CSD,SSPD,SPDLD,GCSD,GSSPD,GSPDLD, N,CSOFFD,SSPD,GCSD,GSSPD,GSSPD, N,GF,PATH,CRC,TO,PA1,PA2,PA3,COOL,GCOOL</td>
</tr>
<tr>
<td>with feedrate:</td>
<td></td>
</tr>
<tr>
<td><strong>PFEED</strong></td>
<td>N,CSD,SSPD,SPDLD,GCSD,GSSPD,GSPDLD, N,CSOFFD,SSPD,GCSD,GSSPD,GSSPD, N,GF,PATH,CRC,TO,FF,CD,PA1,PA2,PA3,FEED</td>
</tr>
<tr>
<td>Circular motion:</td>
<td></td>
</tr>
<tr>
<td><strong>PFEEDC</strong></td>
<td>N,GF,CRC,TO,PA1,PA2,PA3,PC,FEED</td>
</tr>
<tr>
<td>Tapping with compensating chuck on peripheral surface:</td>
<td></td>
</tr>
<tr>
<td><strong>PTAPS</strong></td>
<td>N,&quot;;(PTAPS)&quot;, N,GF,FF,SSPD,SPDLD,GSSPD,GSPDLD, N,TAPS,TO,PA2,PA3,FEED</td>
</tr>
<tr>
<td>on end face:</td>
<td></td>
</tr>
<tr>
<td><strong>FTAPS</strong></td>
<td>N,&quot;;(FTAPS)&quot;, N,GF,FF,SSPD,SPDLD,GSSPD,GSPDLD, N,TAPS,TO,FRA1,FA3,FEED</td>
</tr>
<tr>
<td>Tapping on peripheral surface:</td>
<td></td>
</tr>
<tr>
<td><strong>PTAPR</strong></td>
<td>N,&quot;;(PTAPR)&quot;, N,&quot;SETMS(3)&quot;, N,AOS N,TAPR,PA3, PIT2,SSPD,GSSPD</td>
</tr>
<tr>
<td>Tapping on end face:</td>
<td></td>
</tr>
<tr>
<td><strong>FTAPR</strong></td>
<td>N,&quot;;(FTAPR)&quot;, N,&quot;SETMS(3)&quot;, N,AOS N,TAPR,FA3, PIT1,SSPD,GSSPD</td>
</tr>
<tr>
<td>Formal block name</td>
<td>Formal block content (example)</td>
</tr>
<tr>
<td>-------------------</td>
<td>--------------------------------</td>
</tr>
<tr>
<td><strong>End face on:</strong></td>
<td></td>
</tr>
<tr>
<td><strong>FON</strong></td>
<td>N,&quot;;FACE END LINEAR ON (FON)&quot;</td>
</tr>
<tr>
<td></td>
<td>N,&quot;TRAFOFF&quot;</td>
</tr>
<tr>
<td></td>
<td>N,&quot;SPOS=0&quot;</td>
</tr>
<tr>
<td></td>
<td>N,&quot;G40&quot;</td>
</tr>
<tr>
<td></td>
<td>N,&quot;DIAMOF&quot;</td>
</tr>
<tr>
<td></td>
<td>N,&quot;TRANSMIT(1)&quot;</td>
</tr>
<tr>
<td></td>
<td>N,ZOP</td>
</tr>
<tr>
<td></td>
<td>N,&quot;G17&quot;</td>
</tr>
<tr>
<td></td>
<td>N,&quot;G94&quot;</td>
</tr>
<tr>
<td><strong>End face off:</strong></td>
<td></td>
</tr>
<tr>
<td><strong>FOFF</strong></td>
<td>N,&quot;;FACE END LINEAR OFF (FOFF)&quot;</td>
</tr>
<tr>
<td></td>
<td>N,TRMOFF</td>
</tr>
<tr>
<td></td>
<td>N,&quot;DIAMON&quot;</td>
</tr>
<tr>
<td></td>
<td>N,&quot;G18&quot;</td>
</tr>
<tr>
<td></td>
<td>N,&quot;ZOP&quot;</td>
</tr>
<tr>
<td></td>
<td>N,&quot;G95&quot;</td>
</tr>
<tr>
<td><strong>Drive on:</strong></td>
<td></td>
</tr>
<tr>
<td><strong>DRVON</strong></td>
<td>&quot;;DRON&quot;</td>
</tr>
<tr>
<td></td>
<td>N,&quot;SETMS(2)&quot;</td>
</tr>
<tr>
<td></td>
<td>N,SSPD,SPDL</td>
</tr>
<tr>
<td><strong>Drive off:</strong></td>
<td></td>
</tr>
<tr>
<td><strong>DRVOFF</strong></td>
<td>&quot;;DRVOFF&quot;</td>
</tr>
<tr>
<td></td>
<td>N,&quot;SETMS&quot;</td>
</tr>
<tr>
<td></td>
<td>fin,SPDL</td>
</tr>
</tbody>
</table>
### 16.2.5 Counterspindle

<table>
<thead>
<tr>
<th>Formal block name</th>
<th>Formal block content (example)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Set synchronization marker:</td>
<td></td>
</tr>
<tr>
<td><strong>WAITM</strong></td>
<td>N,&quot;WAITM[&quot;,WAIT&quot;,1,2]&quot;</td>
</tr>
<tr>
<td>CSP: Beginning of program</td>
<td></td>
</tr>
<tr>
<td><strong>GPB1</strong></td>
<td>N,PPD, N,&quot;;(GPB1)&quot;, N,GF,ZOO,DIMS,FROM2,FROM1, N,ZOPA2,ZOPA1,&quot;;TRANS FROM COUNTERSPINDLE&quot;, N,&quot;R48=520-R48;CALCULATE DIST. OF WCS FROM MCS&quot;, N,ZOP,&quot;;ZERO OFFSET FOR COUNTERSPINDLE&quot;, N,&quot;SETMS(3)&quot;, N,GLS,&quot;;GSP: SPEED LIMIT&quot;</td>
</tr>
<tr>
<td>CSP: Program end</td>
<td></td>
</tr>
<tr>
<td><strong>GPE1</strong></td>
<td>N,&quot;;---FINISH 2. CLAMPING---(GPEI)“, N,GCOOL,GSPDL,GSPDDL, N,GF,ZOO,TNC,FROM2,FROM1, N,PE</td>
</tr>
<tr>
<td>CSP: 1st tool change powered tool</td>
<td></td>
</tr>
<tr>
<td><strong>GTC1</strong></td>
<td>N,&quot;;1. TOOL CHANGE DRIVEN TOOL (GTC1_D)“, NM,TN,TO,GCOOL,TNAME, N,&quot;;”,TN_N, N,GCSD, N,GSSPD,GSPDDL</td>
</tr>
</tbody>
</table>
### Formal block name | Formal block content (example)
---|---
CSP: nth tool change  
non-pwrd./pwrd. tool
**GTCN_DN**  
N,";(GTCN_DN)",
N,TRMOFF,
N,GSPDLD,
NM,TN,TO,GCOOL,TNAME,
N,";",TN_P,TN_N,
N,";SWITCH TO MAIN SPINDLE,GSP",
N,"SETMS",
N,GCS,
N,GSSP,GSPDL

CSP: nth tool change  
non-pwrd./pwrd. tool
**GTCN_ND**  
N,";(GTCN_ND)",
N,GCSOFF,
NM,TN,TO,GCOOL,TNAME,
N,";",TN_P,TN_N,
N,";TOOL DRIVE ON",
N,GCSD,
N,GSSPD,GSPDLD

CSP: last tool change pwrd./  
non-pwrd. tool
**GTCLAST_DN**  
N,";(TLAST_DN)",
N,TRMOFF,
NM,TN,TO,GCOOL,TNAME,
N,";",TN_P,TN1,
N,";SWITCH TO MAIN SPINDLE,GSP",
N,"SETMS",
N,GCS,
N,GSSP,GSPDL
<table>
<thead>
<tr>
<th>Formal block name</th>
<th>Formal block content (example)</th>
</tr>
</thead>
<tbody>
<tr>
<td>CSP: last tool change non-pwrld./pwrld. tool</td>
<td>N,&quot;;(GTCLAST_ND)&quot;, N,GCSOFF, NM,TN,TO,GCOOL,TNAME, N,&quot;&quot;,TN_P,TN1, N,&quot;;TOOL DRIVE ON,GSP&quot;, N,GCSD, N,GSSPD,GSPDLD</td>
</tr>
<tr>
<td>GTCLAST_ND</td>
<td></td>
</tr>
<tr>
<td>CSP: Align counterspindle</td>
<td>N,GCSOFF, N,&quot;SETMS(3);(GSPA)&quot;, N,AOS</td>
</tr>
<tr>
<td>GSPA</td>
<td></td>
</tr>
<tr>
<td>GFON</td>
<td></td>
</tr>
<tr>
<td>Drive on</td>
<td>N,&quot;;DRIVE UNIT ON (GDRVON)&quot;, N,&quot;SETMS(2)&quot;, N,GSSPD,GSPDLD</td>
</tr>
<tr>
<td>GDRVON</td>
<td></td>
</tr>
<tr>
<td>Formal block name</td>
<td>Formal block content (example)</td>
</tr>
<tr>
<td>-------------------</td>
<td>--------------------------------</td>
</tr>
<tr>
<td>CSP: Drive off</td>
<td>&quot;;DRIVE UNIT OFF (GDRVOFF)&quot;,</td>
</tr>
<tr>
<td>GDRVOFF</td>
<td>N,&quot;SETMS(3)&quot;,</td>
</tr>
<tr>
<td></td>
<td>N,GSPDLD</td>
</tr>
<tr>
<td>PARTRMAN</td>
<td>N,&quot;M0&quot;</td>
</tr>
<tr>
<td></td>
<td>= &quot;</td>
</tr>
<tr>
<td>CSP: Removal of workpiece with gripper</td>
<td>N,&quot;;PARTRGR&quot;,</td>
</tr>
<tr>
<td>PARTRGR</td>
<td>N,GSPAPOS,&quot;;Grip position relative to counterspindle&quot;,</td>
</tr>
<tr>
<td></td>
<td>N,PTCRYFREE,&quot;;Return path from chuck&quot;</td>
</tr>
<tr>
<td>CSP: Remove workpiece from MSP with CSP</td>
<td>N,&quot;REMOVE PART FROM FROM MSP with CSP&quot;</td>
</tr>
<tr>
<td>PARTRHSGS</td>
<td></td>
</tr>
<tr>
<td>CSP: Counterspindle as tailstock (forward)</td>
<td>N,&quot;;COUNTERSPINLDE AS TAILSTOCK FWD(GSTAILSTF),</td>
</tr>
<tr>
<td>GSTAILSTF</td>
<td>N,SPDL,SSP,</td>
</tr>
<tr>
<td></td>
<td>N,&quot;COUPON(C3,C1,&quot; ,GHWINK, &quot;);COUPLING ON&quot;,</td>
</tr>
<tr>
<td></td>
<td>N,&quot;R99=&quot;,&quot;GSHOEH,GSRPOS,</td>
</tr>
<tr>
<td></td>
<td>N,&quot;G0 Q1=-520+R99&quot;,GSVORL,</td>
</tr>
<tr>
<td></td>
<td>N,&quot;G01 F0.5 Q1=-520+R99&quot;</td>
</tr>
</tbody>
</table>
### Formal block name | Formal block content (example)
--- | ---
CSP: Advance counterspindle/pull workpiece | **GSDRPULL**
N,";GSDRPULL"
CSP: Cut off, remove workpiece with CSP | **CUTRGS**
N,";CUTRGS"
CSP: Counterspindle as tailstock (back) | **GSTAILSTB**
N,";COUNTERSPINDLE BACK AS TAILSTOCK", N,"G0 Q1=0",
N,"SYNC_OFF"
CSP: Counterspindle forward | **GSFORWARD**
N,";COUNTERSPINDLE FWD",
N,"G95",
N,"SYNC_ON",
N,"COUPON(C3,C1,"GHWINK,") ;COUPLING ON (GSFORWARD)",
N,"M49",
N,"R98="GFUMASS,
N,"R99=R98",GSPAPOS,
N,"G0 Q1=520+R98+R48",GANFPOS,";SWITCHOVER POSITION FROM G0 TO G1",
N,"G1 F0.5 Q1=520+R99+R48;POSITION ON WORKPIECE AT CUTTING DEPTH WITH G1",
N,"M48"
CSP: Counterspindle backward | **GSBACKWARD**
N,"G0 Q1=0; COUNTERSPINDLE BACKWARD",
N,"SYN_OFF",
N,"M68; CLOSE MSP CHUCK"
CSP: Re-grip/pull workpiece | **WKSNFAS**
NM,";RE-GRIP/PULL WORKPIECE (WKSNFAS)
N,"M69 ;OPEN MSP CHUCK",
N,"G01 F0.5 G9 Q1="GVORLEN,"(length of pull)"
N,"G90",
N,"M68;CLOSED MSP CHUCK"
## Formual block name | Formal block content (example)
--- | ---
CSP: Retraction/tool retraction | RETRACT
N,";(RETRACT)",
N,CPOS,GSCPOS,
N,CSPF,SPDL,SSPD,GSPDL,GSPDLD,
N,GF,"DO G53",TNC,TC2,
N,TC1,COOL,GCOOL,"D1",
= =

CSP: End of program 1/
Program change to 2nd clamp | GWP
N,";---FINISH 1. CLAMPING---(GWP)",
N,COOL,SPDL,SPDLD,
N,GF,ZOO,TNC,FROM2,FROM1
### 16.3 Assignment between machining operations and blocks called

<table>
<thead>
<tr>
<th>Type of machining</th>
<th>Block called</th>
</tr>
</thead>
<tbody>
<tr>
<td>Grip workpiece with counterspindle from main spindle</td>
<td>PARTRHSGS</td>
</tr>
<tr>
<td></td>
<td>GSFORWARD</td>
</tr>
<tr>
<td></td>
<td>GSBACKWARD</td>
</tr>
<tr>
<td>Cut off workpiece and grip with counterspindle</td>
<td>CUTGRS</td>
</tr>
<tr>
<td></td>
<td>GSFORWARD</td>
</tr>
<tr>
<td></td>
<td>GSBACKWARD</td>
</tr>
<tr>
<td>Pull workpiece forward and cut off</td>
<td>CUTGRS</td>
</tr>
<tr>
<td></td>
<td>GSFORWARD</td>
</tr>
<tr>
<td></td>
<td>GSDRPPULL</td>
</tr>
<tr>
<td></td>
<td>WKSNFAS</td>
</tr>
<tr>
<td></td>
<td>GSBACKWARD</td>
</tr>
<tr>
<td>Single-step machining: Advance counterspindle and pull part</td>
<td>FREIFAHR</td>
</tr>
<tr>
<td></td>
<td>PARTRHSGS</td>
</tr>
<tr>
<td></td>
<td>GSFORWARD</td>
</tr>
<tr>
<td></td>
<td>GSDRPPULL</td>
</tr>
<tr>
<td></td>
<td>WKSNFAS</td>
</tr>
<tr>
<td></td>
<td>GSBACKWARD</td>
</tr>
<tr>
<td>Remove part form counterspindle with gantry</td>
<td>PTTPART</td>
</tr>
<tr>
<td>All bar machining operations with selection in automatic mode</td>
<td>BAR</td>
</tr>
<tr>
<td></td>
<td>CLAMP</td>
</tr>
<tr>
<td></td>
<td>BAR/CLAMP</td>
</tr>
<tr>
<td>All bar machining operations for single-step machining: Manual stop at stop with</td>
<td>BAR</td>
</tr>
<tr>
<td>gripper</td>
<td>BAR/CLAMP</td>
</tr>
<tr>
<td></td>
<td>BAR/CLAMP</td>
</tr>
<tr>
<td>Single-step machining: Cut-off and remove with receiving pan</td>
<td>RCP</td>
</tr>
</tbody>
</table>
### 16.3 Assignment between machining operations and blocks called

<table>
<thead>
<tr>
<th>Type of machining</th>
<th>Block called</th>
</tr>
</thead>
<tbody>
<tr>
<td>Single-step machining: Cut-off and removal with gantry</td>
<td>PTFW, PTBK</td>
</tr>
<tr>
<td>Single-step machining: Manual removal at counterspindle (only 4 axes)</td>
<td>PS</td>
</tr>
</tbody>
</table>
Error Messages

17.1 Introduction............................................................................................................... 17-304
17.2 Alphabetical listing of error messages ........................................................................ 17-305
17.1 Introduction

This chapter lists in alphabetical order the error messages that occur most frequently in the AutoTurn Graphic Programming System. The error messages are self-explanatory and make for easy fault location. Text marked with xxx indicates that the error message text is variable.
17.2 Alphabetical listing of error messages

**A**

- **A face recess can be added to the present contour**
  
  See above (apply analogously to the face recess!).

- **A radial recess can be added to the present contour**
  
  See above (applies analogously to radial recess!).

- **All tools are changed!**
  
  This message only means that all defined tools have been changed! Your toolholder may be full. Please check whether an existing tool has been deleted and can be replaced with a new one. The toolholder must not be full!

- **An undercut can be added to the present contour**
  
  It is normally not technically expedient to add an undercut at this point. If required, however, please use the contour definition to program the undercut.

- **Arc lies outside the blank**
  
  The circle data you have entered exceed the programmed dimensions of the blank. Perhaps the radius or end point you have selected is too large. You can see the incorrect value in the help display. Please check the drawing and enter the correct values.
**B**

Blank is smaller than finished part

The length of the blank entered is too small for your existing finished part. Please check your entries and correct the length of the blank or length of the finished part as appropriate. The blank must be at least as large as the finished part.

---

**C**

Circle exceeds the programmed lengths

The circle data you have entered exceed the programmed length of the finished part. Perhaps the radius or end point you have selected is too large.

Normally AutoTurn provides you with a help display in which you can see the incorrect value. Please check your drawing and enter the correct values.

---

**D**

D or L coordinates do not lie on specified circle

The circle data you have entered are not consistent.

The radius you have selected may be too large/small.

Normally, AutoTurn provides a help display in which you can see the incorrect value. Please check your drawing and enter the correct values.

Defective quadrant: only 3/4/8 possible

The tool you have selected may only take the quadrant values 3/4/8. Please check your entry.
### D

**Distance between center point/starting point not equal to CP/end point**  
When programming a circle, the distance between the starting point and the center point and from the end point to the center point (also the radius) must be constant. Please check your input against the drawing and enter the correct values.

**Distance between end point and center point > circle radius**  
See above!

**Distance between starting point and center point > circle radius**  
See above!

**Distance from start point to end point > double circle radius**  
An incorrect  
- radius,  
- start of circle or  
- end of circle  
has been entered while programming a circle. Please check your drawing and enter the correct values.

**Drill/mill diameter <=0 for tool xxx**  
Tool diameter zero was entered during drilling or milling.
Error during PREBORING xxx
during EXTERNAL PRE-TURNING
LONGITUDINAL xxx
during PREDRILLING xxx
during PRE-FACING xxx

Error encountered:
residual surface xxx 1

AutoTurn has detected an error in the automatic sequence at the type of machining stated or in single machining. Please check the relevant work step or replace the automatic work step with a single work step or a macro.

Error encountered: first external turning on finished part xxx

AutoTurn was unable to finish turning the outside contour of the workpiece entered. Examine the updated workpiece for residual material on the outside surface and machine the remaining part in a single machining step.

Error encountered: first internal turning on finished part xxx

See above (analogously for inside contour!).

Error encountered: thread xxx

The system was unable to machine the thread you have entered for the workpiece. Examine the updated workpiece for residual material on the outside surface and machine the thread in a single machining step.

Error in allowance

The allowance you have entered cannot be calculated for the existing contour. You may have exceeded the dimensions of the blank or the turning center. Please check the dimensions entered and correct if necessary.

Error in drilling pattern data

The system cannot produce a proper drilling pattern with the data entered. Please compare the data entered against the data in the drawing and correct any errors.
**H200 error xxx in contour calculation**

The contour calculation and the automatic sequence is based on a high-level language (H200 language). Unable to calculate the contour due to incorrect parameterization. Check the input values and try to describe the contour in a different way.

**Impermissible thread size**

AutoTurn is unable to process the thread size you have entered. Please correct your entry.

**Impermissible tolerance data**

AutoTurn is unable to process the tolerance you have entered. Please correct your entry.

**Incorrect length xxx**

The length you have entered produces a contour violation (for example: longer than the blank and finished part). Please check the drawing and enter the correct dimensions.

**Incorrect sign**

You have specified the wrong sign. Please check your entries and make the appropriate changes.

**Incorrect sign for depth xxx**

The sign for the depth specified is not correct. If a "-" is indicated in a help display shown, you must also enter it. Please check and correct your entries.

**Incorrect sign for length**

The sign is not correct for the length specified. Please take account of programming in the negative direction. If a "-" is indicated in a help display shown, you must also enter it. Please check your entries and make the appropriate changes.
17.2 Alphabetical listing of error messages

**I**

Incorrect tool width (negative or missing)
The selected tool width cannot be used to produce the workpiece. Please check the input value or select a different tool.

Input data incorrect or incomplete
The input data are inconsistent or the system required further information (inputs) in order to continue calculation of the work step. Please check whether you have the information or whether you have forgotten to enter some values.

Input outside blank/finished part limits
The entered contour or machining exceeds the scope of the raw material you have defined or the dimensions entered for the finished part. Machining is not possible. Please check your drawing and enter the correct values for the geometry or machining.

Input required
An input is required in this field for further calculation. If the required input values are not complete (as for "taper"), the contour definition must be programmed. Please check the drawing and enter the correct values.

**L**

Length of the finished part + cut-off allowance > turning length
The finished part entered (together with the cut-off dimension and cut-off depth) is longer than the turning length (normally the length of the blank). Please correct the length of the blank.
## Error Messages

### 17.2 Alphabetical listing of error messages

**M**

<table>
<thead>
<tr>
<th>Error Description</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Maximum NC block number exceeded</td>
<td>For NC block output, the block number exceeds the maximum block number specified in the machine adaptation settings. Restart numbering the NC blocks.</td>
</tr>
</tbody>
</table>

**N**

<table>
<thead>
<tr>
<th>Error Description</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>No attribute programmed</td>
<td>The system cannot find an attribute that can be changed in the contour you have programmed.</td>
</tr>
<tr>
<td>No contour programmed</td>
<td>AutoTurn cannot find a contour in the workpiece you have selected. Please define a finished part contour to be changed.</td>
</tr>
<tr>
<td>No drilling pattern defined</td>
<td>The system allows you to save drilling patterns. The selected drilling pattern has not been saved in the system however. Please check the name and save the relevant drilling pattern if necessary.</td>
</tr>
<tr>
<td>No inside contour available</td>
<td>See above (analogously for inside contour!).</td>
</tr>
<tr>
<td>No outside contour available</td>
<td>The workpiece you have described does not have an outside contour. Perhaps you have forgotten it or created it as an inside contour. Please check the geometry you have entered.</td>
</tr>
<tr>
<td>No radius/chamfer programmed</td>
<td>The contour you have programmed does not contain a radius or chamfer that can be changed.</td>
</tr>
<tr>
<td>No thread programmed</td>
<td>You have not programmed a thread that can be changed or machined in your contour.</td>
</tr>
</tbody>
</table>
17.2 Alphabetical listing of error messages

N

No tool in the selected group

Tools are defined in 30 groups in the technology database of AutoTurn. The user and the machine manufacturer are able to influence these data. There are currently no tools defined in the tool group you have selected. The definition of new tools is described in Chapter 2.

No workholder of the selected group in the workholder file

Workholders are defined in various groups in the technology database of AutoTurn. The user and the machine manufacturer are able to influence these data. It may be that the workholders in the group you have selected have been deleted. The definition is described in Chapter 3.

Not yet implemented

The function you have selected does not exist in this product version.

O

Only numbers input / calculation permitted

Only numbers may be entered in this input field. AutoTurn also allows you to calculate in the input field (calculator function). The calculated value (for example: 80/2 – 10 = 30), 30 in this case, is calculated directly. Please correct your entry.

Only whole number permitted

Only whole numbers may be used in this input field; i.e. no decimal places entered. Please correct your entry.

Output value set to minimum value

For NC block output, the output value is set to the minimum value of the relevant NC word (can be set in machine adaptation).

Output value set to maximum value

For NC block output, the output value is set to the maximum value of the relevant NC word (can be set in machine adaptation).
P

Processor error xxx

This messages can be encountered in the case of complex workpieces (calculations) or wrong parameters in the input masks. For example, processor error 607 with external pre-turning: Unable to cut segmentation. The allowances may be too large. Check the allowances in the machine adaptation settings.

Programmed tool xxx not on the turret

During the set up dialog, the relevant tool was not assigned to the turret. If possible, the tool is assigned to a free station in the turret. If there is no free station, a "−1" is entered in the NC blocks as the station and offset switch number.

R

Rounding radius is greater than defined circle (same direction of rotation)

The rounding radius you have entered is greater than the radius of the previous element. This would cause the programmed circle to be omitted. Please check your entries and correct the rounding radius or the previous element as appropriate.

Rounding radius/chamfer is not permitted/not possible here

It does not make technical sense or is not possible to enter a radius/chamfer at the point you have specified. You may program the contour definition to avoid this problem.
### 17.2 Alphabetical listing of error messages

#### S

**Starting point and end point are identical**

The programmed contour does not make sense. Contour elements (for example: circle), which define the end point simultaneously with your starting point cannot be produced. Please check your drawing and enter the correct values.

**Station is not assigned**

The selected turret station is not assigned for AutoTurn. Operations like "Replace" produce this error. Please select a different station.

**Straight line and circle do not intersect**

The consecutive elements straight line and circle do not have an intersection in the contour definition programmed. Since the end point of the last element is always the starting point of the next element, these elements cannot be chained. The contour does not make sense. Please check your drawing and enter the correct values.

#### T

**Tangent for chamfer too small**

Unable to calculate the input chamfer in the contour with the size specified. Please check your entry and change the size of the chamfer.

**Tangent for rounding radius too small**

See above (applicable for rounding radius!).

**Tangential arc for rounding radius too small**

Unable to calculate the entered circle radius in the contour with the size specified. Please check your entries and change the radius of the circle or rounding.

**Taper lies outside the blank**

The programmed taper exceeds the dimensions of the blank and therefore does not supply a geometry that makes any sense. Please check your drawing and enter the correct values.
### 17.2 Alphabetical listing of error messages

#### T

**The length of the finished part is greater than the length of the blank**

The length of the finished part entered exceeds the dimensions for the defined blank. Please check your drawing and enter the correct values for your blank or finished part.

The blank must always be greater than the finished part.

**The tool must not be changed**

The tool you have selected for changing may be required for machining the current workpiece. AutoTurn prevents you from changing it.

Please select a different tool or another workpiece.

**Tool automatically assigned to turret station xxx**

During the set up dialog, the relevant tool was not assigned to the turret. Unable to assign a free station on the turret to the tool.

**Turning length is greater than the length of the blank**

The defined machining is greater than the length of the blank.

Please enter the correct values for turning or extend the size of the blank.

**Turning length is smaller than length of the finished part**

The defined machining is smaller than the length of the finished part.

Please check your drawing and enter the correct values.
Unable to add a contour element

It is not possible to add contour elements and contour definitions at this point.
Please delete the last element of your contour and enter a description for this part of the workpiece.
Fast contour entry makes for effective repogramming!

Unable to add a radius/chamfer

AutoTurn is unable to find a suitable point in the contour you have programmed for adding a radius or chamfer. If a radius or chamfer is absolutely necessary, however, you must program it in the contour definition.

Unable to add an attribute

The programming system cannot find a suitable point for adding an attribute in the contour you have programmed.

Undercut/recess not possible

It does not make technical sense to add an undercut or a recess at this point. If absolutely necessary, however, use the contour definition to program the contour of the undercut or recess.

Unknown tool type:
Tool type 500 is entered in the TOA file

During the set up dialog, the relevant tool was not assigned to the turret. So it is not possible to assign the relevant tool type to the tool in the TOA file.
W

Wrong key

You have pressed the wrong key by mistake. AutoTurn attempts to counteract all operating errors so that the user is always guided through the system. Please check the key you have selected.
17.2 Alphabetical listing of error messages

MMC 103  PCU 50
Appendix

A References .................................................................................................................... A-320

B Index ......................................................................................................................... A-335
A References

General Documentation

/BU/
SINUMERIK 840D/840Di/810D/802S, C, D
Ordering Information
Catalog NC 60
Order No.: E86060-K4460-A101-A8-7600

/ST7/
SIMATIC
SIMATIC S7 Programmable Logic Controllers
Catalog ST 70
Order No.: E86060-K4670-A111-A3-7600

/Z/
SINUMERIK, SIROTEC, SIMODRIVE
Accessories and Equipment for Special-Purpose Machines
Catalog NC Z
Order No.: E86060-K4490-A001-A7-7600

Electronic Documentation

/CD1/
The SINUMERIK System
DOC ON CD
(with all SINUMERIK 840D/840Di/810D and SIMODRIVE publications)
Order No.: 6FC5 298-6CA00-0BG3
User Documentation

/AUK/  SINUMERIK 840D/810D
Short Guide AutoTurn Operation  (09.01 Edition)
Order No.: 6FC5 298-4AA30-0BP3

/AUP/  SINUMERIK 840D/810D
AutoTurn Graphic Programming System  (02.02 Edition)
Operator's Guide Programming/Setup
Order No.: 6FC5 298-4AA40-0BP3

/BA/  SINUMERIK 840D/810D
Operator's Guide MMC  (10.00 Edition)
Order No.: 6FC5 298-6AA00-0BP0

/BAA/  SINUMERIK 840D/840Di/810D
Operator's Guide @Event  (01.02 Edition)
Order No.: 6AU1900-0CL20-0BA0

/BAD/  SINUMERIK 840D/840Di/810D
Order No.: 6FC5 298-6AF00-0BP1

/BEM/  SINUMERIK 840D/810D
Operator's Guide HMI Embedded  (09.01 Edition)
Order No.: 6FC5 298-6AC00-0BP1

/BAE/  SINUMERIK 840D/810D
Order No.: 6FC5 298-3AA60-0BP1

/BAH/  SINUMERIK 840D/840Di/810D
Operator's Guide HAT 6 (HPU new)  (09.01 Edition)
Order No.: 6FC5 298-0AD60-0BP2

/BAK/  SINUMERIK 840D/840Di/810D
Short Operating Guide  (02.01 Edition)
Order No.: 6FC5 298-6AA10-0BP0

/BAM/  SINUMERIK 810D/840D
Operator's Guide ManualTurn  (08.00 Edition)
Order No.: 6FC5 298-6AD00-0BP1
<table>
<thead>
<tr>
<th>Code</th>
<th>Title</th>
<th>Edition</th>
<th>Order No.</th>
</tr>
</thead>
<tbody>
<tr>
<td>/BAS/</td>
<td>SINUMERIK 840D/810D Operator's Guide ShopMill</td>
<td>(10.01)</td>
<td>6FC5 298-6AD10-0BP0</td>
</tr>
<tr>
<td>/BAT/</td>
<td>SINUMERIK 840D/810D Operator's Guide ShopTurn</td>
<td>(01.02)</td>
<td>6FC5 298-6AD50-0BP1</td>
</tr>
<tr>
<td>/BAP/</td>
<td>SINUMERIK 840D/840Di/810D Operator's Guide Handheld Programming Unit</td>
<td>(04.00)</td>
<td>6FC5 298-5AD20-0BP1</td>
</tr>
<tr>
<td>/BNM/</td>
<td>SINUMERIK 840D/840Di/810D User's Guide Measuring Cycles</td>
<td>(09.01)</td>
<td>6FC5 298-6AA70-0BP1</td>
</tr>
<tr>
<td>/DA/</td>
<td>SINUMERIK 840D/840Di/810D Diagnostics Guide</td>
<td>(09.01)</td>
<td>6FC5 298-6AA20-0BP1</td>
</tr>
<tr>
<td>/KAM/</td>
<td>SINUMERIK 840D/810D Short Guide ManualTurn</td>
<td>(04.01)</td>
<td>6FC5 298-5AD40-0BP0</td>
</tr>
<tr>
<td>/KAS/</td>
<td>SINUMERIK 840D/810D Short Guide ShopMill</td>
<td>(04.01)</td>
<td>6FC5 298-5AD30-0BP0</td>
</tr>
<tr>
<td>/KAT/</td>
<td>SINUMERIK 840D/810D Short Guide ShopTurn</td>
<td>(07.01)</td>
<td>6FC5 298-6AF20-0BP0</td>
</tr>
<tr>
<td>/PG/</td>
<td>SINUMERIK 840D/840Di/810D Programming Guide Fundamentals</td>
<td>(09.01)</td>
<td>6FC5 298-6AB00-0BP1</td>
</tr>
<tr>
<td>/PGA/</td>
<td>SINUMERIK 840D/840Di/810D Programming Guide Advanced</td>
<td>(09.01)</td>
<td>6FC5 298-6AB10-0BP1</td>
</tr>
<tr>
<td>/PGK/</td>
<td>SINUMERIK 840D/840Di/810D Short Guide Programming</td>
<td>(02.01)</td>
<td>6FC5 298-6AB30-0BP1</td>
</tr>
</tbody>
</table>
/PGM/
SINUMERIK 840D/840Di/810D
Programming Guide ISO Milling  
Order No.: 6FC5 298-6AC20-0BP1
(10.01 Edition)

/PGT/
SINUMERIK 840D/840Di/810D
Programming Guide ISO Turning  
Order No.: 6FC5 298-6AC10-0BP1
(10.01 Edition)

/PGZ/
SINUMERIK 840D/840Di/810D
Programming Guide Cycles  
Order No.: 6FC5 298-6AB40-0BP1
(09.01 Edition)

/PI/
PCIN 4.4
Software for Data Transfer to/from MMC Module  
Order No.: 6FX2 060-4AA00-4XB0 (Ger., Engl., Fr.)  
Order from: WK Fürth

/SYI/
SINUMERIK 840Di
System Overview  
Order No.: 6FC5 298-6AE40-0BP0
(02.01 Edition)

Manufacturer/Service Documentation

a) Lists
/LIS/
SINUMERIK 840D/840Di/810D
SIMODRIVE 611D
Lists  
Order No.: 6FC5 297-6AB70-0BP1
(09.01 Edition)

b) Hardware
/BH/
SINUMERIK 840D/840Di/810D
Operator Components Manual (HW)  
Order No.: 6FC5 297-6AA50-0BP1
(09.01 Edition)

/BHA/
SIMODRIVE Sensor
Absolute Encoder with Profibus DP  
User's Guide (HW)  
Order No.: 6SN1 197-0AB10-0YP1
(02.99 Edition)
References

/EMV/ SINUMERIK, SIROTEC, SIMODRIVE
EMC Installation Guideline (06.99 Edition)
Planning Guide (HW)
Order No.: 6FC5 297-0AD30-0BP1

/PHC/ SINUMERIK 810D
Configuring Manual (HW) (10.01 Edition)
Order No.: 6FC5 297-4AD10-0BP1

/PHD/ SINUMERIK 840D
NCU 561.2-573.2 Configuring Manual (HW) (09.01 Edition)
Order No.: 6FC5 297-6AC10-0BP1

/PHF/ SINUMERIK FM-NC
NCU 570 Configuring Manual (HW) (04.96 Edition)
Order No.: 6FC5 297-3AC00-0BP0

/PMH/ SIMODRIVE Sensor
Measuring System for Main Spindle Drives
Order No.: 6SN1197-0AB30-0BP0
c) Software /FB1/

SINUMERIK 840D/840Di/810D

Description of Functions Basic Machine (Part 1) (09.01 Edition)

(the various sections are listed below)

Order No.: 6FC5 297-6AC20-0BP1

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 (Part 2) (09.01 Edition)

including FM-NC: Turning, Stepping Motor

(the various sections are listed below)

Order No.: 6FC5 297-6AC30-0BP1

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
A-326 SINUMERIK 840D/840Di/810D AutoTurn Programming/Setup (AUP) – 02.02 Edition

Appendix 02.02

References

S5 Synchronized Actions (up to SW 3/thereafter /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) (09.01 Edition)
(the various sections are listed below)
Order No.: 6FC5 297-6AC80-0BP1

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 Speed/Torque Coupling Master-Slave
TE4 Transformation Package Handling
TE5 Setpoint Exchange
TE6 MCS Coupling
TE7 Retrace Support
TE8 Clock-Independent Path-Synchronous Output of Switching Signal
V2 Preprocessing
W5 3D Tool Radius Compensation

/FBA/

SIMODRIVE 611D/SINUMERIK 840D/810D
Description of Functions Drive Functions (09.01 Edition)
(the various sections are listed below)
Order No.: 6SN1 197-0AA80-0BP7

DB1 Operational Messages/Alarm Reactions
DD1 Diagnostic Functions
DD2 Speed Control Loop
DE1 Extended Drive Functions
DF1 Enable Commands
DG1 Encoder Parameterization
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/ IT Solutions
NC Data Management Server (DNC NT-2000)
Description of Functions (05.00 Edition)
Order No.: 6FC5 297-5AE50-0BP1

/FBDT/ IT Solutions
NC Data Transfer (SinDNC)
Description of Functions (03.01 Edition)
Order No.: 6FC5 297-1AE70-0BP0

/FBFA/ SINUMERIK 840D/840Di/810D
Description of Functions (09.01 Edition)
ISO Dialects for SINUMERIK
Order No.: 6FC5 297-6AE10-0BP1

/FBFE/ SINUMERIK 840D/810D
Description of Functions Remote Diagnosis (11.01 Edition)
Order No.: 6FC5 297-0AF00-0BP1
FE1 Remote Diagnosis
FE2 Alarm-Controlled Notification per Email: @Event

/FBH/ SINUMERIK 840D/840Di/810D
HMI Programming Package (10.01 Edition)
Order No.: (supplied with 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
References

/FBMA/  SINUMERIK 840D/810D  
Description of Functions **ManualTurn**  (08.00 Edition)  
Order No.: 6FC5 297-6AD50-0BP1

/FBO/  SINUMERIK 840D/810D  
Description of Functions  
**Configuring OP 030 Operator Interface**  (09.01 Edition)  
(the various 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  
PS  Online only: Configuration Syntax (Configuring Package):  
is supplied with software and available as PDF file.

/FBP/  SINUMERIK 840D  
Description of Functions **C-PLC Programming**  (03.96 Edition)  
Order No.: 6FC5 297-3AB60-0BP0

/FBR/  SINUMERIK 840D/810D  
IT Solutions  
**Computer Link (SinCOM)**  (04.00 Edition)  
Description of Functions  
Order No.: 6FC5 297-5AD60-0BP0  
NFL  Host Computer Interface  
NPL  PLC/NCK Interface

/FBSI/  SINUMERIK 840D/SIMODRIVE  
Description of Functions  
**SINUMERIK Safety Integrated**  (03.01 Edition)  
Order No.: 6FC5 297-6AB80-0BP0

/FBSP/  SINUMERIK 840D/810D  
Description of Functions **ShopMill**  (10.01 Edition)  
Order No.: 6FC5 297-6AD80-0BP1

/FBST/  **SIMATIC**  
**FM STEPDRIVE/SIMOSTEP**  (01.01 Edition)  
Description of Functions  
Order No.: 6SN1 197-0AA70-0YP4
<table>
<thead>
<tr>
<th>Code</th>
<th>Title</th>
<th>Edition</th>
</tr>
</thead>
<tbody>
<tr>
<td>/FBSY/</td>
<td>SINUMERIK 840D/810D Description of Functions <strong>Synchronized Actions</strong> for Wood, Glass, Ceramics and Presses</td>
<td>(09.01 Edition)</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>/FBT/</td>
<td>SINUMERIK 840D/810D Description of Functions <strong>ShopTurn</strong></td>
<td>(03.01 Edition)</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>/FBTC/</td>
<td>SINUMERIK 840D/810D IT Solutions <strong>SINUMERIK Tool Data Communication SinTDC</strong></td>
<td>(01.02 Edition)</td>
</tr>
<tr>
<td></td>
<td>Description of Functions</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>/FBTD/</td>
<td>SINUMERIK 840D/810D IT Solutions <strong>Tool Information (SinTDI) with Online Help</strong></td>
<td>(03.01 Edition)</td>
</tr>
<tr>
<td></td>
<td>Description of Functions</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>/FBU/</td>
<td><strong>SIMODRIVE 611 universal</strong> Description of Functions</td>
<td>(08.01 Edition)</td>
</tr>
<tr>
<td></td>
<td>Closed-Loop Control Component for Speed Control and Positioning</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>/FBW/</td>
<td>SINUMERIK 840D/810D Description of Functions <strong>Tool Management</strong></td>
<td>(10.01 Edition)</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>/HBI/</td>
<td>SINUMERIK 840Di Manual</td>
<td>(09.01 Edition)</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
Appendix 02.02 References

/MMC 103 PCU 50/

/SINUMERIK 840D/840Di/810D
SINUMERIK Start-Up Tool SinuCOM NC (02.02 Edition)
System Description
Order No.: (included in online help for Start-Up Tool)

/KBU/
SIMODRIVE 611 universal
Short Description (05.00 Edition)
Closed-Loop Control Component for Speed Control
Order No.: 6SN1 197-0AB40-0BP3

/PFK/
SIMODRIVE
Planning Guide 1FT5/1FT6/1FK6 Motors (12.01 Edition)
Three-Phase AC Servo Motors for Feed and
Main Spindle Drives
Order No.: 6SN1197-0AC20-0BP0

/PJE/
SINUMERIK 840D/810D
Configuring Package HMI Embedded (08.01 Edition)
Description of Functions: Software Update, Configuration,
Installation
Order No.: 6FC5 297-6EA10-0BP0
(The publication "Configuring Syntax" is supplied with
software and is available as PDF file)

/PJFE/
SIMODRIVE
Planning Guide Synchronous Motors 1FE1 (09.01 Edition)
Three-Phase AC Motors for Main Spindle Drives
Order No.: 6SN1 197-0AC00-0BP1

/PJLM/
SIMODRIVE
Planning Guide Linear Motors (09.01 Edition)
(on request)
ALL General Information about Linear Motors
1FN1 Three-Phase AC Linear Motor 1FN1
1FN3 Three-Phase AC Linear Motor 1FN3
CON Connections
Order No.: 6SN1 197-0AB70-0BP2

/PJM/
SIMODRIVE
Planning Guide Motors (09.00 Edition)
AC Motors for Feed and Main Spindle Drives
Order No.: 6SN1 197-0AA20-0BP4
<table>
<thead>
<tr>
<th>Code</th>
<th>Description</th>
<th>Edition</th>
</tr>
</thead>
<tbody>
<tr>
<td>/PJU/</td>
<td>SIMODRIVE 611 Planning Guide Inverters</td>
<td>(05.01 Edition)</td>
</tr>
<tr>
<td></td>
<td>Order No.: 6SN1 197-0AA00-0BP5</td>
<td></td>
</tr>
<tr>
<td>/POS1/</td>
<td>SIMODRIVE POSMO A User's Guide Distributed Positioning Motor on PROFIBUS DP</td>
<td>(04.01 Edition)</td>
</tr>
<tr>
<td></td>
<td>Order No.: 6SN2 197-0AA00-0BP2</td>
<td></td>
</tr>
<tr>
<td>/POS2/</td>
<td>SIMODRIVE POSMO A Installation Instructions (enclosed with POSMO A)</td>
<td>(12.98 Edition)</td>
</tr>
<tr>
<td></td>
<td>Order No.: 462 008 0815 00</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Order No.: 6SN2 197-0AA20-0BP1</td>
<td></td>
</tr>
<tr>
<td>/PPH/</td>
<td>SIMODRIVE Planning Guide 1PH2/1PH4/1PH7 Motors AC Asynchronous Motors for Main Spindle Drives</td>
<td>(12.01 Edition)</td>
</tr>
<tr>
<td></td>
<td>Order No.: 6SN1197-0AC60-0BP0</td>
<td></td>
</tr>
<tr>
<td>/PPM/</td>
<td>SIMODRIVE Planning Guide Hollow-Shaft Motors Hollow-Shaft Motors for Main Spindle Drives 1PM4 and 1PM6</td>
<td>(10.01 Edition)</td>
</tr>
<tr>
<td></td>
<td>Order No.: 6SN1197-0AD03-0BP0</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Order No.: 6ES7 398-8AA03-8AA0</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Order No.: 6ES7 810-4CA02-8AA0</td>
<td></td>
</tr>
<tr>
<td>/S7HR/</td>
<td>SIMATIC S7-300 Manual STEP 7, Reference Manuals, V. 3.1</td>
<td>(03.97 Edition)</td>
</tr>
<tr>
<td></td>
<td>Order No.: 6ES7 810-4CA02-8AR0</td>
<td></td>
</tr>
<tr>
<td>/S7S/</td>
<td>SIMATIC S7-300 FM 353 Positioning Module for Stepper Drives</td>
<td>(04.97 Edition)</td>
</tr>
<tr>
<td></td>
<td>Order in conjunction with Configuring Package</td>
<td></td>
</tr>
</tbody>
</table>
Appendix 02.02

References

/S7L/ SIMATIC S7-300
FM 354 Positioning Module for Servo Drives
Order in conjunction with Configuring Package

/S7M/ SIMATIC S7-300
FM 357.2 Multi-Axis Module for Servo and Stepper Drives
Order in conjunction with Configuring Package

/SHM/ SIMODRIVE 611
Manual
Single-Axis Positioning for MCU 172A
Order No.: 6SN 1197-4MA00-0BP0

/SP/ SIMODRIVE 611-A/611-D
SimoPro 3.1
Program for Configuring Machine-Tool Drives
Order No.: 6SC6 111-6PC00-0BA, Order from: WK Fürth
d) Installation and Start-up

/IAA/ SIMODRIVE 611A
Installation and Start-Up Guide (10.00 Edition)
(incl. description of SIMODRIVE 611D start-up software)
Order No.: 6SN 1197-0AA60-0BP6

/IAC/ SINUMERIK 810D
Installation and Start-Up Guide (10.01 Edition)
(incl. description of SIMODRIVE 611D start-up software)
Order No.: 6FC5 297-4AD20-0BP1

/IAD/ SINUMERIK 840D/SIMODRIVE 611D
Installation and Start-Up Guide (09.01 Edition)
(incl. description of SIMODRIVE 611D start-up software)
Order No.: 6FC5 297-6AB10-0BP1

/IAF/ SINUMERIK FM-NC
Installation and Start-Up Guide (07.00 Edition)
Order No.: 6FC5 297-3AB00-0BP1

/IAM/ SINUMERIK 840D/840Di/810D
HMI/MMC Installation and Start-Up Guide (11.01 Edition)
Order No.: 6FC5 297-6AE20-0BP1

AE1 Updates/Options
BE1 Extend the Operator Interface
HE1 Online Help
IM2 HMI Embedded Start-Up
IM4 HMI Advanced Start-Up (PCU 50)
TX1 Creating Foreign-Language Texts
Appendix
References

MMC 103  PCU 50
B Index

A
Acceleration rates 14-270
Adaptation to the machine 14-254
Adaptation to the machine, operation 14-255
Adaptation to the machine, Setting 14-256
Address letter 14-262
Allowance 5-82
Allowance finished part 7-146
Allowance, equidistant 7-149
Allowances 5-106, 14-258
Arc 5-92, 5-93, 5-95
Automatic 8-162
Automatic run 8-163

B
Blank 5-75, 5-78
Blank definition 7-131
Blank dimensions 5-78
Blank material 5-79
Blank stock 7-131
Blank stocks 5-78
Blank stocks, extended 5-79
Block numbering 14-257
Boring rods 2-39

C
Center drills 2-42
Centered positioning 5-81
Chamfer 5-98, 7-136
Chamfer angle 5-98, 7-140
Chamfer right 7-140
Change 5-106
Changing material data 4-64
Chasing passes 4-62
Chuck height 14-258
Circle of holes 9-188
CIRCULAR-SAW MILL 2-46
CIRCULAR-SAW MILLER 2-45

D
Defining your own dimensions 5-82
Definition, free 7-146
Delete 5-103
Delete material 4-68
Delete program 5-73, 5-76
Deleting a tool 2-50
Depth of cut 4-62
Diameter programming 5-95
Dimension 5-107
Dimensions 5-80
Distance to go 7-142
Downtimes 14-269
DRILLED GROOVE MILL 2-46
Drilled holes 13-234
Drilling tools 2-42
DRILLS 2-42
DXF boundary conditions 12-222

G
G functions 14-261
Geometry description 7-134
Geometry help 10-192, 10-195
Grooving tools 2-40

H
Handling single-step non-cutting operation 9-174
Hexagon 5-78
Hollow cylinder 5-78

I
ID number 2-37
Insert 5-104
INSERT DRILLS 2-43
Inside contour 5-85
Inside jaws 3-56
Installation 1-26
Installation on PC 1-26
Interactive programming 9-169
Internal contour 5-85, 7-140
INTERNAL GROOVING TOOLS 2-40
INTERNAL THREAD CHASERS 2-41

K
Knurling tool 2-45

L
Left-hand thread 5-102
Left-justified positioning 5-81
LIMIT STOP 2-43
Link 7-138
Link to CAD systems 12-222
LONGIT. BORING BARS 2-39
Longitudinal axis 7-133
LONGITUDINAL TURNING TOOLS 2-38

M
Machining limit modification 5-116
Machining limits 5-111, 5-115, 7-143, 7-145
Machining method 4-60
Machining operations 8-164
<table>
<thead>
<tr>
<th>Topic</th>
<th>Page(s)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Machining plan</td>
<td>8-164</td>
</tr>
<tr>
<td>Machining position</td>
<td>5-108</td>
</tr>
<tr>
<td>Machining side</td>
<td>5-110</td>
</tr>
<tr>
<td>Machining times</td>
<td>14-269</td>
</tr>
<tr>
<td>Macro drilling</td>
<td>9-179</td>
</tr>
<tr>
<td>Macro Grooving</td>
<td>9-178</td>
</tr>
<tr>
<td>Macro programming example</td>
<td>9-185</td>
</tr>
<tr>
<td>Macro Turning</td>
<td>9-178</td>
</tr>
<tr>
<td>Macros</td>
<td>9-178, 9-181</td>
</tr>
<tr>
<td>Main block</td>
<td>14-265</td>
</tr>
<tr>
<td>Main menu</td>
<td>5-71, 7-130, 13-225</td>
</tr>
<tr>
<td>Marking operations</td>
<td>8-165</td>
</tr>
<tr>
<td>Material</td>
<td>4-61, 7-132</td>
</tr>
<tr>
<td>Materials</td>
<td>4-60</td>
</tr>
<tr>
<td>Metric thread</td>
<td>5-100</td>
</tr>
<tr>
<td>Milling</td>
<td>9-183</td>
</tr>
<tr>
<td>Milling macros</td>
<td>9-187</td>
</tr>
<tr>
<td>Milling menus</td>
<td>9-180</td>
</tr>
<tr>
<td>Milling tools</td>
<td>2-44</td>
</tr>
<tr>
<td>Mirror workpiece</td>
<td>5-108</td>
</tr>
<tr>
<td>Modal functions</td>
<td>14-261</td>
</tr>
<tr>
<td>Modality</td>
<td>14-261</td>
</tr>
<tr>
<td>Modifying tool data</td>
<td>2-47</td>
</tr>
<tr>
<td>Monitor</td>
<td>14-265</td>
</tr>
<tr>
<td>Multiple thread</td>
<td>5-102</td>
</tr>
<tr>
<td>NC blocks</td>
<td>14-265, 16-283</td>
</tr>
<tr>
<td>NC blocks, C axis</td>
<td>16-292</td>
</tr>
<tr>
<td>NC blocks, Counterspindle</td>
<td>16-295</td>
</tr>
<tr>
<td>NC blocks, general</td>
<td>16-285</td>
</tr>
<tr>
<td>NC blocks, operation</td>
<td>14-267</td>
</tr>
<tr>
<td>NC blocks, tool change</td>
<td>16-286</td>
</tr>
<tr>
<td>NC blocks, turning</td>
<td>16-289</td>
</tr>
<tr>
<td>NC data</td>
<td>8-168</td>
</tr>
<tr>
<td>NC plain text</td>
<td>14-267</td>
</tr>
<tr>
<td>NC words</td>
<td>14-260, 15-273, 15-274</td>
</tr>
<tr>
<td>NC words, C-axis machining</td>
<td>15-281</td>
</tr>
<tr>
<td>NC words, counterspindle</td>
<td>15-282</td>
</tr>
<tr>
<td>NC words, feedrate</td>
<td>15-278</td>
</tr>
<tr>
<td>NC words, insertion texts</td>
<td>15-280</td>
</tr>
<tr>
<td>NC words, operation</td>
<td>14-263</td>
</tr>
<tr>
<td>NC words, spindle</td>
<td>15-278</td>
</tr>
<tr>
<td>NC words, thread</td>
<td>15-280</td>
</tr>
<tr>
<td>NC words, tool change</td>
<td>15-277</td>
</tr>
<tr>
<td>NC words, turning</td>
<td>15-276</td>
</tr>
<tr>
<td>Neutral view</td>
<td>2-35</td>
</tr>
<tr>
<td>New program</td>
<td>5-71</td>
</tr>
<tr>
<td>Non-cutting handling macro</td>
<td>9-180</td>
</tr>
<tr>
<td>Operating elements</td>
<td>1-19</td>
</tr>
<tr>
<td>Operating instructions</td>
<td>1-28</td>
</tr>
<tr>
<td>Operation</td>
<td>7-130</td>
</tr>
<tr>
<td>Operator action</td>
<td>8-162</td>
</tr>
<tr>
<td>Operator keyboard</td>
<td>1-19</td>
</tr>
<tr>
<td>Output resolution</td>
<td>14-256</td>
</tr>
<tr>
<td>Output system</td>
<td>14-256</td>
</tr>
<tr>
<td>Outside contour</td>
<td>5-85</td>
</tr>
<tr>
<td>Outside jaw</td>
<td>3-56</td>
</tr>
<tr>
<td>Outside jaws</td>
<td>7-144</td>
</tr>
<tr>
<td>Parameters</td>
<td>7-129</td>
</tr>
<tr>
<td>Part program list</td>
<td>8-168</td>
</tr>
<tr>
<td>PC keyboard</td>
<td>1-20, 1-23</td>
</tr>
<tr>
<td>PC settings</td>
<td>1-28</td>
</tr>
<tr>
<td>Peripheral surface</td>
<td>5-86</td>
</tr>
<tr>
<td>Peripheral surface machining</td>
<td>7-159</td>
</tr>
<tr>
<td>Pitch</td>
<td>4-62</td>
</tr>
<tr>
<td>Pocket</td>
<td>13-236</td>
</tr>
<tr>
<td>Pocket contour</td>
<td>7-151</td>
</tr>
<tr>
<td>Position finished part</td>
<td>7-134</td>
</tr>
<tr>
<td>Positioning</td>
<td>5-80</td>
</tr>
<tr>
<td>Positioning the finished part</td>
<td>7-133</td>
</tr>
<tr>
<td>Prereturning longitudinal</td>
<td>13-249, 13-250, 13-251</td>
</tr>
<tr>
<td>Priorities</td>
<td>14-271</td>
</tr>
<tr>
<td>Processing mode</td>
<td>5-72</td>
</tr>
<tr>
<td>Program generation</td>
<td>8-162</td>
</tr>
<tr>
<td>Program handling</td>
<td>5-73, 5-74, 5-75, 8-168</td>
</tr>
<tr>
<td>Program handling list</td>
<td>8-168</td>
</tr>
<tr>
<td>Program management</td>
<td>5-73</td>
</tr>
<tr>
<td>Program memory</td>
<td>5-74</td>
</tr>
<tr>
<td>Program name</td>
<td>5-71</td>
</tr>
<tr>
<td>Program name</td>
<td>5-74</td>
</tr>
<tr>
<td>Program selection</td>
<td>5-74, 7-128</td>
</tr>
</tbody>
</table>
Programming 5-71
Programming functions 5-70
Programming sequence 1-22

R
Rapid traverse rates 14-270
REAMERS 2-43
Relief-cut elements 5-115
Right-hand thread 5-102
Rough face turning 13-243, 13-247
Rough turning longitudinal 13-245
Roughing feedrate 4-62
Roughness height 5-99, 7-138
Rounding 5-99

S
Sample program sprocket shaft 13-223
Screen layout 1-23
Section 5-83
Setpoints 14-258
Set-up 6-120
Setup dialog 8-167
Setup functions 5-70
Setup sheet 14-257
Short Guide AutoTurn 0-12
Simulation 8-166
Single grooving operation 9-172, 9-173
Single milling operation 9-174
Single thread operation 9-174
Single-step machining 9-170, 9-175
Single-step machining Operation 9-175
Single-step machining Turning 9-171
SINUMERIK 840D-ASCII editor 5-77
Speed limitations 14-258
Speed limits 8-162
SPOT DRILLS 2-42
Square 5-78
Start conditions 7-128
Start-up SINUMERIK controls 1-25
STEP DRILLS 2-43
Straight line 5-94, 7-141
System 14-271
System settings 14-271

T
Tailstock points 3-57
Tailstock types 7-144
Taper 5-92, 7-138
Taper, angle 5-92
Texts 14-265
Thread 5-100, 7-136
Thread chasers 2-41
Thread cutting tools 2-46
Thread lead 5-100
THREAD TAPS 2-43
Thread undercut 7-137
Thread undercuts 5-101
Time calculation 14-257, 14-269
Tips and tricks 1-28
TOA data 14-257
Tool change 14-258
Tool data 2-37
Tool data dialog 2-47
Tool groups 2-32
Tool holder 8-167
Tool master data 2-35
Tool paths 8-166
Tool search 14-258
Tool, delete 6-120
Tool, exchange 6-120
Tool, zoom in 6-120
Toolholder setting 6-118
Tools 2-32
Tools, machine view 2-35
Tools, neutral views 2-35
Tools, powered 14-265
Turret indexing 14-271
Turret stations 14-271

U
UN threads 5-102
Undefined contour elements 10-191
Undefined contour elements, programming example 10-193
Undercuts 5-97
Use of AutoTurn 1-18
V
Value calculation  14-261

W
Whitworth thread  5-101
Workholder  7-143
Workholder data dialog  5-110
Workholder dialog  3-58
Workholder groups  3-54
Workholder master data  3-56

Workholder selection  5-110
Workholders  3-53, 5-110
Working area  14-258
Working zone limitations  14-258
Workpiece display  5-83
Workpiece zero  5-112

Z
Zero offset  5-112
Zero point  5-84
Appendix

Index

MMC 103  PCU 50
To
SIEMENS AG
A&D MC BMS
P.O. Box 3180
D-91050 Erlangen, Germany
Tel. ++49 – 180 / 5050 222 [Hotline]
Fax ++49 – 9131 / 98 – 2176 [Documentation]
email: motioncontrol.docu@erlf.siemens.de

Suggestions
Corrections
For Publication/Manual:
SINUMERIK 840D/840Di/810D
AutoTurn
Graphic Programming System
User Documentation

From
Name
Company/Dept.
Address:
Telephone: /
Fax: /

Suggestions and/or corrections

Programming/Setup
Order No.: 6FC5298-4AA40-0BP3
Edition: 02.02

Should you come across any printing errors when reading this publication, please notify us on this sheet. Suggestions for improvement are also welcome.
# Overview of SINUMERIK 840D/840Di/810D Documentation (02.2002)

## General Documentation
- SINUMERIK 840D/810D
- SINUMERIK 840D/840Di/810D
- SINUMERIK SIMODRIVE SIMODRIVE

## User Documentation
- SINUMERIK 840D/840Di/810D
- SINUMERIK SIMODRIVE SIMODRIVE
- SINUMERIK SIROTEC SIROTEC

## Manufacturer/Service Documentation
- SINUMERIK 840D/840Di/810D
- SINUMERIK SIMODRIVE SIMODRIVE
- SINUMERIK SIMODRIVE SIROTEC

## Electronic Documentation
- DOC ON CD

---

### User Documentation
- **Description of Functions**
  - Basic Machine
  - Advanced
  - Cycles
  - Measuring Cycles
  - ISO Turning/Milling
- **Drive Functions**
  - 810D
  - 840D
  - HMI

### Manufacturer/Service Documentation
- **Description of Functions**
  - Drive Functions
  - Drive Functions (HW)
  - Drive Functions (HW) Advanced
  - Drive Functions (FW)
- **Configuring Kit**
  - HMI Embedded
  - HMI Advanced

### Electronic Documentation
- DOC ON CD

---

*) These documents are a minimum requirement