# SINUMERIK

## SINUMERIK 840D sl Grinding

Operating Manual

## Valid for:
SINUMERIK 840D sl / 840DE sl

## Software version
- CNC system software for 840D sl / 840DE sl V4.92
- SINUMERIK Operate for PCU/PC V4.92

## 06/2019
A5E44903521B AB

## Table of Contents

<table>
<thead>
<tr>
<th>Chapter</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Preface</td>
<td>1</td>
</tr>
<tr>
<td>Fundamental safety instructions</td>
<td>2</td>
</tr>
<tr>
<td>Introduction</td>
<td>3</td>
</tr>
<tr>
<td>Multitouch operation with SINUMERIK Operate</td>
<td>4</td>
</tr>
<tr>
<td>Setting up the machine</td>
<td>5</td>
</tr>
<tr>
<td>Working in the manual mode</td>
<td>6</td>
</tr>
<tr>
<td>Machining the workpiece</td>
<td>7</td>
</tr>
<tr>
<td>Simulating machining</td>
<td>8</td>
</tr>
<tr>
<td>Generating a G-code program</td>
<td>9</td>
</tr>
<tr>
<td>Programming technology functions</td>
<td>10</td>
</tr>
<tr>
<td>Grinding with a B axis (only for cylindrical grinding machines)</td>
<td>11</td>
</tr>
<tr>
<td>Collision avoidance</td>
<td>12</td>
</tr>
<tr>
<td>Multi-channel view</td>
<td>13</td>
</tr>
<tr>
<td>Manage tools</td>
<td>14</td>
</tr>
<tr>
<td>Managing programs</td>
<td>15</td>
</tr>
<tr>
<td>Alarm, fault, and system messages</td>
<td>16</td>
</tr>
<tr>
<td>Teaching in a program</td>
<td>17</td>
</tr>
<tr>
<td>Ctrl-Energy</td>
<td>18</td>
</tr>
<tr>
<td>Easy XML</td>
<td>19</td>
</tr>
<tr>
<td>HT 8 (840D sl only)</td>
<td>A</td>
</tr>
<tr>
<td>Appendix</td>
<td></td>
</tr>
</tbody>
</table>
Legal information

Warning notice system

This manual contains notices you have to observe in order to ensure your personal safety, as well as to prevent damage to property. The notices referring to your personal safety are highlighted in the manual by a safety alert symbol, notices referring only to property damage have no safety alert symbol. These notices shown below are graded according to the degree of danger.

⚠️ DANGER
indicates that death or severe personal injury will result if proper precautions are not taken.

⚠️ WARNING
indicates that death or severe personal injury may result if proper precautions are not taken.

⚠️ CAUTION
indicates that minor personal injury can result if proper precautions are not taken.

⚠️ NOTICE
indicates that property damage can result if proper precautions are not taken.

If more than one degree of danger is present, the warning notice representing the highest degree of danger will be used. A notice warning of injury to persons with a safety alert symbol may also include a warning relating to property damage.

Qualified Personnel

The product/system described in this documentation may be operated only by personnel qualified for the specific task in accordance with the relevant documentation, in particular its warning notices and safety instructions. Qualified personnel are those who, based on their training and experience, are capable of identifying risks and avoiding potential hazards when working with these products/systems.

Proper use of Siemens products

Note the following:

⚠️ WARNING
Siemens products may only be used for the applications described in the catalog and in the relevant technical documentation. If products and components from other manufacturers are used, these must be recommended or approved by Siemens. Proper transport, storage, installation, assembly, commissioning, operation and maintenance are required to ensure that the products operate safely and without any problems. The permissible ambient conditions must be complied with. The information in the relevant documentation must be observed.

Trademarks

All names identified by ® are registered trademarks of Siemens AG. The remaining trademarks in this publication may be trademarks whose use by third parties for their own purposes could violate the rights of the owner.

Disclaimer of Liability

We have reviewed the contents of this publication to ensure consistency with the hardware and software described. Since variance cannot be precluded entirely, we cannot guarantee full consistency. However, the information in this publication is reviewed regularly and any necessary corrections are included in subsequent editions.
Preface

SINUMERIK documentation

The SINUMERIK documentation is organized into the following categories:

- General documentation/catalogs
- User documentation
- Manufacturer/service documentation

Additional information

You can find information on the following topics at the following address (https://support.industry.siemens.com/cs/de/en/view/108464614):

- Ordering documentation/overview of documentation
- Additional links to download documents
- Using documentation online (find and search in manuals/information)

If you have any questions regarding the technical documentation (e.g. suggestions, corrections), please send an e-mail to the following address (mailto:docu.motioncontrol@siemens.com).

mySupport/Documentation

At the following address (https://support.industry.siemens.com/My/ww/en/documentation), you can find information on how to create your own individual documentation based on Siemens' content, and adapt it for your own machine documentation.

Training

At the following address (http://www.siemens.com/sitrain), you can find information about SITRAIN (Siemens training on products, systems and solutions for automation and drives).

FAQs


SINUMERIK

You can find information about SINUMERIK at the following address (http://www.siemens.com/sinumerik).
Target group

This documentation is intended for users of cylindrical grinding machines and grinding machines running the SINUMERIK Operate software.

Benefits

The operating manual helps users familiarize themselves with the control elements and commands. Guided by the manual, users are capable of responding to problems and taking corrective action.

Standard scope

This documentation describes the functionality of the standard scope. Extensions or changes made by the machine manufacturer are documented by the machine manufacturer.

Other functions not described in this documentation might be executable in the control. However, no claim can be made regarding the availability of these functions when the equipment is first supplied or in the event of servicing.

Furthermore, for the sake of clarity, this documentation does not contain all detailed information about all types of the product and cannot cover every conceivable case of installation, operation or maintenance.

Note regarding the General Data Protection Regulation

Siemens observes standard data protection principles, in particular the principle of privacy by design. That means that

this product does not process / store any personal data, only technical functional data (e.g. time stamps). If a user links this data with other data (e.g. a shift schedule) or stores personal data on the same storage medium (e.g. hard drive) and thus establishes a link to a person or persons, then the user is responsible for ensuring compliance with the relevant data protection regulations.

Technical Support

Country-specific telephone numbers for technical support are provided in the Internet at the following address (https://support.industry.siemens.com/sc/ww/en/sc/2090) in the "Contact" area.
# Table of contents

Preface ................................................................................................................................................. 3

1 Fundamental safety instructions ........................................................................................................ 15
  1.1 General safety instructions ............................................................................................................. 15
  1.2 Warranty and liability for application examples ............................................................................ 16
  1.3 Industrial security .......................................................................................................................... 17

2 Introduction .......................................................................................................................................... 19
  2.1 Product overview ............................................................................................................................ 19
  2.2 Operator panel fronts ...................................................................................................................... 20
    2.2.1 Overview .................................................................................................................................. 20
    2.2.2 Keys of the operator panel ....................................................................................................... 22
  2.3 Machine control panels .................................................................................................................. 30
    2.3.1 Overview .................................................................................................................................. 30
    2.3.2 Controls on the machine control panel ..................................................................................... 30
  2.4 User interface .................................................................................................................................. 34
    2.4.1 Screen layout ............................................................................................................................ 34
    2.4.2 Status display ............................................................................................................................ 35
    2.4.3 Actual value window .................................................................................................................. 37
    2.4.4 T,F,S window ............................................................................................................................ 39
    2.4.5 Current block display ................................................................................................................. 40
    2.4.6 Operation via softkeys and buttons ........................................................................................... 42
    2.4.7 Entering or selecting parameters .............................................................................................. 43
    2.4.8 Pocket calculator ....................................................................................................................... 45
    2.4.9 Pocket calculator functions ....................................................................................................... 46
    2.4.10 Context menu ........................................................................................................................... 48
    2.4.11 Changing the user interface language ...................................................................................... 48
    2.4.12 Entering Chinese characters .................................................................................................... 49
      2.4.12.1 Entering Asian characters .................................................................................................. 51
      2.4.12.2 Editing the dictionary ....................................................................................................... 52
    2.4.13 Entering Korean characters ...................................................................................................... 53
    2.4.14 Protection levels ....................................................................................................................... 56
    2.4.15 Cleaning mode .......................................................................................................................... 57
    2.4.16 Display live image from a camera ............................................................................................. 58
    2.4.17 Online help in SINUMERIK Operate ....................................................................................... 59

3 Multitouch operation with SINUMERIK Operate ............................................................................ 63
  3.1 Multitouch panels ............................................................................................................................ 63
  3.2 Touch-sensitive user interface .......................................................................................................... 64
  3.3 Finger gestures ................................................................................................................................. 65
  3.4 Multitouch user interface .................................................................................................................. 68
    3.4.1 Screen layout ............................................................................................................................. 68
4 Setting up the machine ........................................................................................................85

4.1 Switching on and switching off .......................................................................................85

4.2 Approaching a reference point .......................................................................................86

4.2.1 Referencing axes .........................................................................................................86

4.2.2 User agreement ..........................................................................................................87

4.3 Operating modes ............................................................................................................89

4.3.1 Operating modes .........................................................................................................89

4.3.2 Modes groups and channels .......................................................................................91

4.3.3 Channel switchover .................................................................................................91

4.4 Settings for the machine ...............................................................................................93

4.4.1 Switching over the coordinate system (MCS/WCS) ......................................................93

4.4.2 Switching the unit of measurement ..........................................................................93

4.4.3 Setting the zero offset ...............................................................................................95

4.5 Zero offsets ....................................................................................................................97

4.5.1 Zero offsets ...............................................................................................................97

4.5.2 Display active zero offset ..........................................................................................98

4.5.3 Displaying the zero offset “overview” .....................................................................99

4.5.4 Displaying and editing base zero offset ................................................................100

4.5.5 Displaying and editing settable zero offset ..............................................................101

4.5.6 Displaying and editing seat-related fine offset .........................................................101

4.5.7 Displaying and editing details of the zero offsets .....................................................102

4.5.8 Deleting a zero offset ...............................................................................................104

4.5.9 Deleting seat-related fine offsets ............................................................................104

4.6 Measuring a tool ............................................................................................................106
6.2 Selecting a program ................................................................. 145
6.3 Executing a trial program run ................................................ 146
6.4 Displaying the current program block .................................... 148
6.4.1 Current block display ......................................................... 148
6.4.2 Displaying a basic block ..................................................... 149
6.4.3 Display program level ......................................................... 150
6.5 Correcting a program ............................................................... 152
6.6 Repositioning axes ................................................................. 153
6.7 Starting execution at a specific point ....................................... 154
6.7.1 Use block search ............................................................... 154
6.7.2 Continuing program from search target .............................. 156
6.7.3 Simple search target definition .......................................... 156
6.7.4 Defining an interruption point as search target .................. 157
6.7.5 Parameters for block search in the search pointer ............... 157
6.7.6 Block search mode ............................................................ 158
6.8 Controlling the program run .................................................. 160
6.8.1 Program control ............................................................... 160
6.8.2 Skip blocks ...................................................................... 161
6.9 Overstore .............................................................................. 163
6.10 Editing a program ................................................................. 165
6.10.1 Processing a program (editor) ........................................... 165
6.10.2 Searching in programs ..................................................... 165
6.10.3 Replacing program text .................................................... 167
6.10.4 Copying/pasting/deleting program blocks ....................... 168
6.10.5 Renumbering a program .................................................. 169
6.10.6 Creating a program block ................................................ 170
6.10.7 Editor settings ............................................................... 172
6.11 Working with DXF files ........................................................ 176
6.11.1 Overview ......................................................................... 176
6.11.2 Displaying CAD drawings ............................................... 177
6.11.2.1 Open a DXF file ......................................................... 177
6.11.2.2 Cleaning a DXF file ................................................... 177
6.11.2.3 Enlarging or reducing the CAD drawing ....................... 178
6.11.2.4 Changing the section ................................................... 179
6.11.2.5 Rotating the view ......................................................... 179
6.11.2.6 Displaying/editing information for the geometric data ... 180
6.11.3 Importing and editing a DXF file ....................................... 181
6.11.3.1 General procedure ..................................................... 181
6.11.3.2 Setting the tolerance ................................................... 181
6.11.3.3 Assigning the machining plane ..................................... 181
6.11.3.4 Selecting the machining range / deleting the range and element .................................................. 182
6.11.3.5 Saving the DXF file ...................................................... 183
6.11.3.6 Specifying a reference point ....................................... 184
6.11.3.7 Accepting contours ..................................................... 184
6.12 Display and edit user variables ............................................. 187
6.12.1 Overview ........................................................................ 187
6.12.2 Global R parameters ........................................................ 188
7 Simulating machining .................................................................................................................. 209
  7.1 Overview ............................................................................................................................. 209
  7.2 Simulation before machining of the workpiece ................................................................. 213
    7.2.1 Simulation before machining of the workpiece......................................................... 213
    7.2.2 Start simulation .......................................................................................................... 213
  7.3 Simultaneous recording before machining of the workpiece........................................... 215
    7.3.1 Overview .................................................................................................................... 215
    7.3.2 Starting simultaneous recording ................................................................................ 215
  7.4 Simultaneous recording during machining of the workpiece ............................................ 216
  7.5 Program control during simulation .................................................................................... 217
    7.5.1 Changing the feedrate ............................................................................................... 217
    7.5.2 Simulating the program block by block .................................................................... 218
  7.6 Different views of a workpiece .......................................................................................... 219
    7.6.1 Overview .................................................................................................................... 219
    7.6.2 Side view ..................................................................................................................... 219
    7.6.3 Dressing view ............................................................................................................. 219
    7.6.4 3D view ....................................................................................................................... 219
    7.6.5 Face view - for cylindrical grinding ............................................................................ 220
    7.6.6 Further side views - for surface grinding .................................................................. 220
    7.6.7 Machine space (with "collision avoidance" option) .................................................. 221
  7.7 Editing the simulation display ............................................................................................. 222
  7.7.1 Deleting the tool path .................................................................................................... 222
  7.8 Editing and adapting a simulation graphic ......................................................................... 223
    7.8.1 Enlarging or reducing the graphical representation ................................................ 223
    7.8.2 Panning a graphical representation ........................................................................... 224
    7.8.3 Rotating the graphical representation ....................................................................... 224
    7.8.4 Modifying the viewport ............................................................................................ 225
  8 Generating a G-code program ............................................................................................... 227
  8.1 Graphical programming ..................................................................................................... 227
<table>
<thead>
<tr>
<th>Section</th>
<th>Title</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>8.2</td>
<td>Program views</td>
<td>228</td>
</tr>
<tr>
<td>8.3</td>
<td>Program structure</td>
<td>230</td>
</tr>
<tr>
<td>8.4</td>
<td>Basic principles</td>
<td>231</td>
</tr>
<tr>
<td>8.4.1</td>
<td>Machining planes</td>
<td>231</td>
</tr>
<tr>
<td>8.4.2</td>
<td>Programming a tool (T)</td>
<td>231</td>
</tr>
<tr>
<td>8.5</td>
<td>Generating a G code program</td>
<td>233</td>
</tr>
<tr>
<td>8.6</td>
<td>Selection of the cycles via softkey</td>
<td>234</td>
</tr>
<tr>
<td>9</td>
<td>Programming technology functions</td>
<td>235</td>
</tr>
<tr>
<td>9.1</td>
<td>Programming contours</td>
<td>235</td>
</tr>
<tr>
<td>9.1.1</td>
<td>Representation of the contour</td>
<td>235</td>
</tr>
<tr>
<td>9.1.2</td>
<td>Creating a new contour</td>
<td>236</td>
</tr>
<tr>
<td>9.1.3</td>
<td>Creating contour elements</td>
<td>238</td>
</tr>
<tr>
<td>9.1.3.1</td>
<td>Entering a contour element</td>
<td>240</td>
</tr>
<tr>
<td>9.1.3.2</td>
<td>Cylindrical grinding</td>
<td>241</td>
</tr>
<tr>
<td>9.1.3.3</td>
<td>Surface grinding</td>
<td>242</td>
</tr>
<tr>
<td>9.1.4</td>
<td>Changing the contour</td>
<td>245</td>
</tr>
<tr>
<td>9.1.4.1</td>
<td>Overview</td>
<td>245</td>
</tr>
<tr>
<td>9.1.4.2</td>
<td>Modifying a contour element</td>
<td>245</td>
</tr>
<tr>
<td>9.1.5</td>
<td>Contour call (CYCLE62)</td>
<td>246</td>
</tr>
<tr>
<td>9.1.5.1</td>
<td>Function</td>
<td>246</td>
</tr>
<tr>
<td>9.1.5.2</td>
<td>Calling the cycle</td>
<td>247</td>
</tr>
<tr>
<td>9.1.5.3</td>
<td>Parameter</td>
<td>247</td>
</tr>
<tr>
<td>9.2</td>
<td>Set dresser coordinate system - CYCLE435</td>
<td>248</td>
</tr>
<tr>
<td>9.2.1</td>
<td>Note regarding the dressing tool position</td>
<td>248</td>
</tr>
<tr>
<td>9.2.2</td>
<td>Function</td>
<td>248</td>
</tr>
<tr>
<td>9.3</td>
<td>Form-truing (CYCLE495)</td>
<td>250</td>
</tr>
<tr>
<td>9.4</td>
<td>Oscillating cycles (CYCLE4071 ... CYCLE4079)</td>
<td>254</td>
</tr>
<tr>
<td>9.4.1</td>
<td>Note regarding oscillating cycles</td>
<td>254</td>
</tr>
<tr>
<td>9.4.2</td>
<td>CYCLE4071 - longitudinal grinding with infeed at the reversal point</td>
<td>254</td>
</tr>
<tr>
<td>9.4.3</td>
<td>CYCLE4072 - longitudinal grinding with infeed at the reversal point and cancel signal</td>
<td>256</td>
</tr>
<tr>
<td>9.4.4</td>
<td>CYCLE4073 - longitudinal grinding with continuous infeed</td>
<td>260</td>
</tr>
<tr>
<td>9.4.5</td>
<td>CYCLE4074 - longitudinal grinding with continuous infeed and cancel signal</td>
<td>261</td>
</tr>
<tr>
<td>9.4.6</td>
<td>CYCLE4075 - surface grinding with infeed at the reversal point</td>
<td>264</td>
</tr>
<tr>
<td>9.4.7</td>
<td>CYCLE4077 - surface grinding with infeed at the reversal point and cancel signal</td>
<td>267</td>
</tr>
<tr>
<td>9.4.8</td>
<td>CYCLE4078 - surface grinding with continuous infeed</td>
<td>271</td>
</tr>
<tr>
<td>9.4.9</td>
<td>CYCLE4079 - surface grinding with intermittent infeed</td>
<td>273</td>
</tr>
<tr>
<td>9.5</td>
<td>Aligning a grinding wheel (CYCLE400)</td>
<td>277</td>
</tr>
<tr>
<td>9.5.1</td>
<td>Function</td>
<td>277</td>
</tr>
<tr>
<td>9.5.2</td>
<td>Calling the cycle</td>
<td>277</td>
</tr>
<tr>
<td>10</td>
<td>Grinding with a B axis (only for cylindrical grinding machines)</td>
<td>279</td>
</tr>
<tr>
<td>10.1</td>
<td>Overview</td>
<td>279</td>
</tr>
<tr>
<td>10.2</td>
<td>T, S, M window for set-up B axis</td>
<td>281</td>
</tr>
<tr>
<td>10.3</td>
<td>Measuring in JOG</td>
<td>283</td>
</tr>
<tr>
<td>10.3.1</td>
<td>Grinding wheel alignment for grinding</td>
<td>283</td>
</tr>
<tr>
<td>10.3.2</td>
<td>Manually measuring the grinding tool (with B axis)</td>
<td>283</td>
</tr>
</tbody>
</table>
10.3.3 Manually measuring the dresser (with B axis) ................................................................. 284
10.3.4 Calibrating the swivel axis ............................................................................................ 286

11 Collision avoidance ............................................................................................................ 289
  11.1 Activating collision avoidance ......................................................................................... 291
  11.2 Set collision avoidance .................................................................................................. 292

12 Multi-channel view ............................................................................................................ 295
  12.1 Multi-channel view ........................................................................................................ 295
  12.2 Multi-channel view in the "Machine" operating area ..................................................... 296
  12.3 Multi-channel view for large operator panels .............................................................. 299
  12.4 Setting the multi-channel view ...................................................................................... 301

13 Manage tools ...................................................................................................................... 303
  13.1 Lists for the tool management ...................................................................................... 303
  13.2 Magazine management ................................................................................................. 305
  13.3 Tool types .................................................................................................................... 306
  13.4 Tool dimensioning ....................................................................................................... 308
  13.5 Tool list ........................................................................................................................ 310
  13.5.1 Tool list ................................................................................................................... 310
  13.5.2 Additional data ........................................................................................................ 312
  13.5.3 Creating a new tool .................................................................................................. 313
  13.5.4 Measuring a tool - tool list ..................................................................................... 315
  13.5.5 Managing several cutting edges ............................................................................. 315
  13.5.6 Deleting a tool ......................................................................................................... 316
  13.5.7 Loading and unloading tools ..................................................................................... 317
  13.5.8 Selecting a magazine ............................................................................................... 318
  13.5.9 Code carrier connection (only 840D sl) ................................................................. 319
  13.5.9.1 Overview ........................................................................................................... 319
  13.5.9.2 Managing tools on a code carrier ...................................................................... 320
  13.5.10 Managing a tool in a file ...................................................................................... 322
  13.6 Tool wear ..................................................................................................................... 324
  13.6.1 Tool wear ............................................................................................................... 324
  13.6.2 Reactivating a tool .................................................................................................. 327
  13.7 Tool data OEM .............................................................................................................. 329
  13.8 Magazine ...................................................................................................................... 331
  13.8.1 Positioning a magazine .......................................................................................... 332
  13.8.2 Relocating a tool .................................................................................................... 333
  13.8.3 Deleting/unloading/loading/relocating all tools ...................................................... 334
  13.9 Sorting tool management lists ..................................................................................... 335
  13.10 Filtering the tool management lists ........................................................................... 336
  13.11 Specific search in the tool management lists ............................................................ 338
  13.12 Tool details ............................................................................................................... 340
  13.12.1 Displaying tool details ......................................................................................... 340
  13.12.2 Tool data ............................................................................................................ 340

Grinding
Operating Manual, 06/2019, A5E44903521B AB
## Table of contents

### Managing programs

<table>
<thead>
<tr>
<th>Section</th>
<th>Description</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>14.1</td>
<td>Overview</td>
<td>355</td>
</tr>
<tr>
<td>14.1.1</td>
<td>NC memory</td>
<td>358</td>
</tr>
<tr>
<td>14.1.2</td>
<td>Local drive</td>
<td>358</td>
</tr>
<tr>
<td>14.1.3</td>
<td>Creating an NC directory on the local drive</td>
<td>358</td>
</tr>
<tr>
<td>14.1.4</td>
<td>USB drives</td>
<td>359</td>
</tr>
<tr>
<td>14.1.5</td>
<td>FTP drive</td>
<td>360</td>
</tr>
<tr>
<td>14.2</td>
<td>Opening and closing the program</td>
<td>362</td>
</tr>
<tr>
<td>14.3</td>
<td>Executing a program</td>
<td>364</td>
</tr>
<tr>
<td>14.4</td>
<td>Creating a directory / program / job list</td>
<td>366</td>
</tr>
<tr>
<td>14.4.1</td>
<td>File and directory names</td>
<td>366</td>
</tr>
<tr>
<td>14.4.2</td>
<td>Creating a new directory</td>
<td>366</td>
</tr>
<tr>
<td>14.4.3</td>
<td>Creating a new workpiece</td>
<td>367</td>
</tr>
<tr>
<td>14.4.4</td>
<td>Creating a new G code program</td>
<td>368</td>
</tr>
<tr>
<td>14.4.5</td>
<td>Creating a new dressing program</td>
<td>368</td>
</tr>
<tr>
<td>14.4.6</td>
<td>Storing any new file</td>
<td>369</td>
</tr>
<tr>
<td>14.4.7</td>
<td>Creating a job list</td>
<td>370</td>
</tr>
<tr>
<td>14.4.8</td>
<td>Creating a program list</td>
<td>372</td>
</tr>
<tr>
<td>14.5</td>
<td>Creating templates</td>
<td>373</td>
</tr>
<tr>
<td>14.6</td>
<td>Searching directories and files</td>
<td>374</td>
</tr>
<tr>
<td>14.7</td>
<td>Displaying the program in the Preview</td>
<td>376</td>
</tr>
<tr>
<td>14.8</td>
<td>Selecting several directories/programs</td>
<td>377</td>
</tr>
<tr>
<td>14.9</td>
<td>Copying and pasting a directory/program</td>
<td>379</td>
</tr>
<tr>
<td>14.10</td>
<td>Deleting a program/directory</td>
<td>381</td>
</tr>
<tr>
<td>14.11</td>
<td>Changing file and directory properties</td>
<td>382</td>
</tr>
<tr>
<td>14.12</td>
<td>Creating drives</td>
<td>384</td>
</tr>
<tr>
<td>14.12.1</td>
<td>Overview</td>
<td>384</td>
</tr>
<tr>
<td>14.12.2</td>
<td>Setting up drives</td>
<td>384</td>
</tr>
<tr>
<td>14.13</td>
<td>EXTCLASS</td>
<td>391</td>
</tr>
</tbody>
</table>
14.14 Execution from external storage (EES) ................................................................. 393
14.15 Backing up data .................................................................................................... 394
14.15.1 Generating an archive in the Program Manager ............................................. 394
14.15.2 Generating an archive via the system data ..................................................... 395
14.15.3 Reading in an archive in the Program Manager .............................................. 397
14.15.4 Read in archive from system data ..................................................................... 398
14.16 Setup data ........................................................................................................... 400
14.16.1 Backing up setup data .................................................................................... 400
14.16.2 Reading-in set-up data .................................................................................. 401
14.17 Backing up parameters ..................................................................................... 404
14.18 V24 .................................................................................................................... 407
14.18.1 Reading-in and reading-out archives via a serial interface ......................... 407
14.18.2 Setting V24 in the program manager ............................................................. 409

15 Alarm, fault, and system messages ........................................................................... 411
15.1 Displaying messages ........................................................................................... 411
15.2 Displaying alarms ............................................................................................... 412
15.3 Displaying an alarm log ..................................................................................... 415
15.4 Sorting, alarms, faults and messages .................................................................. 416
15.5 Creating screenshots ......................................................................................... 417
15.6 PLC and NC variables ....................................................................................... 418
15.6.1 Displaying and editing PLC and NC variables ............................................... 418
15.6.2 Saving and loading screen forms ................................................................. 422
15.7 Version ................................................................................................................ 423
15.7.1 Displaying version data ................................................................................ 423
15.7.2 Save information ........................................................................................... 424
15.8 Logbook .............................................................................................................. 425
15.8.1 Overview ....................................................................................................... 425
15.8.2 Displaying and editing the logbook .................................................................. 425
15.8.3 Making a logbook entry ................................................................................ 426
15.9 Remote diagnostics ............................................................................................ 428
15.9.1 Setting remote access .................................................................................. 428
15.9.2 Permit modem .............................................................................................. 429
15.9.3 Request remote diagnostics ......................................................................... 430
15.9.4 Exit remote diagnostics ............................................................................... 431

16 Teaching in a program ............................................................................................. 433
16.1 Overview ........................................................................................................... 433
16.2 Select teach in mode .......................................................................................... 435
16.3 Processing a program ........................................................................................ 436
16.3.1 Inserting a block .......................................................................................... 436
16.3.2 Editing a block .............................................................................................. 436
16.3.3 Selecting a block .......................................................................................... 437
16.3.4 Deleting a block ........................................................................................... 437
## Table of contents

16.4 Teach sets ........................................................................................................................... 439
16.4.1 Input parameters for teach-in blocks ........................................................................... 440
16.5 Settings for teach-in ........................................................................................................... 442

17 Ctrl-Energy ........................................................................................................................................... 443
17.1 Functions .......................................................................................................................... 443
17.2 Ctrl-E analysis ................................................................................................................... 444
17.2.1 Displaying energy consumption ................................................................................. 444
17.2.2 Displaying the energy analyses ................................................................................... 445
17.2.3 Measuring and saving the energy consumption ....................................................... 446
17.2.4 Tracking measurements ............................................................................................. 447
17.2.5 Tracking usage values ............................................................................................... 448
17.2.6 Comparing usage values ............................................................................................ 448
17.2.7 Long-term measurement of the energy consumption .............................................. 449
17.3 Ctrl-E profiles ................................................................................................................... 451
17.3.1 Using the energy-saving profile ................................................................................ 451

18 Easy XML ............................................................................................................................................ 453
18.1 Easy XML .......................................................................................................................... 453
18.2 Easy Extend ....................................................................................................................... 455
18.2.1 Overview ..................................................................................................................... 455
18.2.2 Enabling a device ....................................................................................................... 455
18.2.3 Activating and deactivating a device ......................................................................... 456
18.2.4 Initial commissioning of additional devices ............................................................... 457
18.2.5 Commissioning Easy Extend .................................................................................... 457
18.3 SINUMERIK Integrate Run MyScreens ........................................................................ 458

19 HT 8 (840D sl only)............................................................................................................................ 461
19.1 Overview .......................................................................................................................... 461
19.2 Traversing keys ............................................................................................................... 464
19.3 Machine control panel menu .......................................................................................... 465
19.4 Virtual keyboard .............................................................................................................. 467
19.5 Calibrating the touch panel ............................................................................................ 469

A Appendix............................................................................................................................................. 471
A.1 840D sl / 828D documentation overview ........................................................................ 471

Index......................................................................................................................................................... 473
1.1 General safety instructions

**WARNING**

Danger to life if the safety instructions and residual risks are not observed

If the safety instructions and residual risks in the associated hardware documentation are not observed, accidents involving severe injuries or death can occur.

- Observe the safety instructions given in the hardware documentation.
- Consider the residual risks for the risk evaluation.

**WARNING**

Malfunctions of the machine as a result of incorrect or changed parameter settings

As a result of incorrect or changed parameterization, machines can malfunction, which in turn can lead to injuries or death.

- Protect the parameterization against unauthorized access.
- Handle possible malfunctions by taking suitable measures, e.g. emergency stop or emergency off.
1.2 Warranty and liability for application examples

Application examples are not binding and do not claim to be complete regarding configuration, equipment or any eventuality which may arise. Application examples do not represent specific customer solutions, but are only intended to provide support for typical tasks.

As the user you yourself are responsible for ensuring that the products described are operated correctly. Application examples do not relieve you of your responsibility for safe handling when using, installing, operating and maintaining the equipment.
1.3 Industrial security

Note
Industrial security

Siemens provides products and solutions with industrial security functions that support the secure operation of plants, systems, machines and networks.

In order to protect plants, systems, machines and networks against cyber threats, it is necessary to implement – and continuously maintain – a holistic, state-of-the-art industrial security concept. Products and solutions from Siemens constitute one element of such a concept.

Customers are responsible for preventing unauthorized access to their plants, systems, machines and networks. Such systems, machines and components should only be connected to an enterprise network or the Internet if and to the extent such a connection is necessary and only when appropriate security measures (e.g. using firewalls and/or network segmentation) are in place.

For additional information on industrial security measures that can be implemented, please visit:
Industrial security (https://www.siemens.com/industrialsecurity)

Siemens’ products and solutions undergo continuous development to make them more secure. Siemens strongly recommends that product updates are applied as soon as they become available, and that only the latest product versions are used. Use of product versions that are no longer supported, and failure to apply the latest updates may increase customer’s exposure to cyber threats.

To stay informed about product updates, subscribe to the Siemens Industrial Security RSS Feed at:
Industrial security (https://www.siemens.com/industrialsecurity)

Further information is provided on the Internet:
## WARNING

**Unsafe operating states resulting from software manipulation**

Software manipulations, e.g. viruses, Trojans, or worms, can cause unsafe operating states in your system that may lead to death, serious injury, and property damage.

- Keep the software up to date.
- Incorporate the automation and drive components into a holistic, state-of-the-art industrial security concept for the installation or machine.
- Make sure that you include all installed products into the holistic industrial security concept.
- Protect files stored on exchangeable storage media from malicious software by with suitable protection measures, e.g. virus scanners.
- On completion of commissioning, check all security-related settings.
- Protect the drive against unauthorized changes by activating the "Know-how protection" converter function.
Introduction

2.1 Product overview

The SINUMERIK control system is a CNC (Computerized Numerical Control) for machine tools. You can use the CNC to implement the following basic functions in conjunction with a machine tool:

- Create can adapt part programs
- Execute part programs
- Manual control
- Access internal and external data media
- Edit data for programs
- Manage tools, zero points and further user data required in programs
- Diagnose control system and machine

Operating areas

The basic functions are grouped in the following operating areas in the control:

- Execute part program, manual control
- Editing data for programs/tool management
- Creating and adapting part programs
- Access to internal and external data storage media
- Alarm display, service display
- Adapting the NC data to the machine, system setting

Operating areas:

- MACHINE
- PARAMETER
- PROGRAM
- PROGRAM MANAGER
- DIAGNOSTICS
- COMMISSIONING
2.2 Operator panel fronts

2.2.1 Overview

The display (screen) and operation (e.g. hardkeys and softkeys) of the SINUMERIK Operate user interface are via the operator panel front.
Operator controls and indicators

In this example, the OP 010 operator panel front is used to illustrate the components that are available for operating the controller and machine tool.

1. Alphabetic key group
   With the <Shift> key pressed, you activate the special characters on keys with double assignments, and write in the uppercase.
   Note: Depending on the particular configuration of your control system, uppercase letters are always written.

2. Numerical key group
   With the <Shift> key pressed, you activate the special characters on keys with double assignments.

3. Control key group
4. Hotkey group
5. Cursor key group
6. USB interface
7. Menu select key
8. Menu forward button
9. Machine area button
10. Menu back key
11. Softkeys
Further information

Further information about the OP 010 and other usable operator panel fronts can be found at:


2.2.2 Keys of the operator panel

The following keys and key combinations are available for operation of the control and the machine tool.

Keys and key combinations

<table>
<thead>
<tr>
<th>Key</th>
<th>Function</th>
</tr>
</thead>
<tbody>
<tr>
<td>&lt;ALARM CANCEL&gt;</td>
<td>Cancels alarms and messages that are marked with this icon.</td>
</tr>
<tr>
<td>&lt;CHANNEL&gt;</td>
<td>Advances if several channels exist.</td>
</tr>
<tr>
<td>&lt;HELP&gt;</td>
<td>Calls the context-sensitive online help for the selected window.</td>
</tr>
<tr>
<td>&lt;NEXT WINDOW&gt; *</td>
<td>Toggles between the windows.</td>
</tr>
<tr>
<td></td>
<td>For a multi-channel view or for a multi-channel functionality, switches within a channel gap between the upper and lower window.</td>
</tr>
<tr>
<td></td>
<td>Selects the first entry in selection lists and in selection boxes.</td>
</tr>
<tr>
<td></td>
<td>Moves the cursor to the beginning of a text.</td>
</tr>
<tr>
<td>* on USB keyboards use the &lt;Home&gt; or &lt;Pos 1&gt; key</td>
<td></td>
</tr>
</tbody>
</table>
2.2 Operator panel fronts

- **<NEXT WINDOW> + <SHIFT>**
  - Selects the first entry in selection lists and in selection boxes.
  - Moves the cursor to the beginning of a text.
  - Selects a contiguous selection from the current cursor position up to the target position.
  - Selects a contiguous selection from the current cursor position up to the beginning of a program block.

- **<NEXT WINDOW> + <ALT>**
  - Moves the cursor to the first object.
  - Moves the cursor in the first column of a table row.
  - Moves the cursor to the beginning of a program block.

- **<NEXT WINDOW> + <CTRL>**
  - Moves the cursor to the beginning of a program.
  - Moves the cursor in the first row of the current column.

- **<NEXT WINDOW> + <CTRL> + <SHIFT>**
  - Moves the cursor to the beginning of a program.
  - Moves the cursor in the first row of the current column.
  - Selects a contiguous selection from the current cursor position up to the target position.
  - Selects a contiguous selection from the current cursor position up to the beginning of the program.

- **<PAGE UP>**
  Scrolls upwards by one page in a window.

- **<PAGE UP> + <SHIFT>**
  In the program manager and in the program editor from the cursor position, selects directories or program blocks up to the beginning of the window.

- **<PAGE UP> + <CTRL>**
  Positions the cursor to the topmost line of a window.

- **<PAGE DOWN>**
  Scrolls downwards by one page in a window.

- **<PAGE DOWN> + <SHIFT>**
  In the program manager and in the program editor, from the cursor position, selects directories or program blocks up to the end of the window.

- **<PAGE DOWN> + <CTRL>**
  Positions the cursor to the lowest line of a window.

- **<Cursor right>**
  - Editing box
    - Opens a directory or program (e.g. cycle) in the editor.
  - Navigation
    - Moves the cursor further to the right by one character.
2.2 Operator panel fronts

<Cursor right> + <CTRL>
- Editing box
  Moves the cursor further to the right by one word.
- Navigation
  Moves the cursor in a table to the next cell to the right.

<Cursor left>
- Editing box
  Closes a directory or program (e.g. cycle) in the program editor.
  If you have made changes, then these are accepted.
- Navigation
  Moves the cursor further to the left by one character.

<Cursor left> + <CTRL>
- Editing box
  Moves the cursor further to the left by one word.
- Navigation
  Moves the cursor in a table to the next cell to the left.

<Cursor up>
- Editing box
  Moves the cursor into the next upper field.
- Navigation
  - Moves the cursor in a table to the next cell upwards.
  - Moves the cursor upwards in a menu screen.

<Cursor up> + <CTRL>
- Moves the cursor in a table to the beginning of the table.
- Moves the cursor to the beginning of a window.

<Cursor up> + <SHIFT>
In the program manager and in the program editor, selects a contiguous selection of directories and program blocks.

<Cursor down>
- Editing box
  Moves the cursor downwards.
- Navigation
  - Moves the cursor in a table to the next cell downwards.
  - Moves the cursor in a window downwards.

<Cursor down> + <CTRL>
- Navigation
  - Moves the cursor in a table to the end of the table.
  - Moves the cursor to the end of a window.
- Simulation
  Reduces the override.

<Cursor down> + <SHIFT>
In the program manager and in the program editor, selects a contiguous selection of directories and program blocks.
**<SELECT>**
Switches between several specified options in selection drop-down list boxes and in selection boxes.
Activates checkboxes.
In the program editor and in the program manager, selects a program block or a program.

**<SELECT> + <CTRL>**
When selecting table rows, switches between selected and not selected.

**<SELECT> + <SHIFT>**
Selects in selection lists and in selection boxes the previous entry or the last entry.

**<END>**
Moves the cursor to the last entry field in a window, to the end of a table or a program block.
Selects the last entry in selection lists and in selection boxes.

**<END> + <SHIFT>**
Moves the cursor to the last entry.
Selects a contiguous selection from the cursor position up to the end of a program block.

**<END> + <CTRL>**
Moves the cursor to the last entry in the last line of the current column or to the end of a program.

**<END> + <CTRL> + <SHIFT>**
Moves the cursor to the last entry in the last line of the current column or to the end of a program.
Selects a contiguous selection from the cursor position up to the end of a program block.

**<BACKSPACE>**
- **Editing box**
  Deletes a character selected to the left of the cursor.
- **Navigation**
  Deletes all of the selected characters to the left of the cursor.

**<BACKSPACE> + <CTRL>**
- **Editing window**
  Deletes a word selected to the left of the cursor.
- **Navigation**
  Deletes all of the selected characters to the left of the cursor.

**<TAB>**
- **Indents the cursor by one character in the program editor.**
- **Moves the cursor to the next entry to the right in the program manager.**
2.2 Operator panel fronts

- **<TAB> + <SHIFT>**
  - Indents the cursor by one character in the program editor.
  - Moves the cursor to the next entry to the left in the program manager.

- **<TAB> + <CTRL>**
  - Indents the cursor by one character in the program editor.
  - Moves the cursor to the next entry to the right in the program manager.

- **<TAB> + <SHIFT> + <CTRL>**
  - Indents the cursor by one character in the program editor.
  - Moves the cursor to the next entry to the left in the program manager.

- **<CTRL> + <A>**
  - Selects all entries (only in the program editor and program manager) in the current window.

- **<CTRL> + <C>**
  - Copies the selected content.

- **<CTRL> + <E>**
  - Calls the “Ctrl Energy” function.

- **<CTRL> + <F>**
  - Opens the search dialog in the machine data and setting data lists, when loading and saving in the MDI editor as well as in the program manager and in the system data.

- **<CTRL> + <G>**
  - Switches in the parameter screen between the help display and the graphic view.

- **<CTRL> + <L>**
  - Switches the current user interface successively through all installed languages.

- **<CTRL> + <SHIFT> + <L>**
  - Switches the current user interface through all installed languages in the inverse sequence.

- **<CTRL> + <P>**
  - Generates a screenshot from the current user interface and saves it as a file.

- **<CTRL> + <S>**
  - Switches the single block in or out in the simulation.

- **<CTRL> + <V>**
  - Pastes text from the clipboard at the current cursor position.
  - Pastes text from the clipboard at the position of a selected text.

- **<CTRL> + <X>**
  - Cuts out the selected text. The text is contained in the clipboard.
<CTRL> + <Y>
Reactivates changes that were undone (only in the program editor).

<CTRL> + <Z>
Undoes the last action (only in the program editor).

<CTRL> + <ALT> + <C>
Creates a complete standard archive (.ARC) on an external data carrier (USB-FlashDrive) (for 840D sl / 828D).

Note:
Please refer to the machine manufacturer's specifications.

<CTRL> + <ALT> + <S>
Creates a complete standard archive (.ARC) on an external data carrier (USB-FlashDrive) (for 840D sl).
Creates a complete Easy Archive (.ARD) on an external data carrier (USB-FlashDrive) (for 828D).

Note:
Please refer to the machine manufacturer's specifications.

<CTRL> + <ALT> + <D>
Backs up the log files on the USB-FlashDrive. If a USB-FlashDrive is not inserted, then the files are backed-up in the manufacturer's area of the CF card.

<SHIFT> + <ALT> + <D>
Backs up the log files on the USB-FlashDrive. If a USB-FlashDrive is not inserted, then the files are backed-up in the manufacturer's area of the CF card.

<SHIFT> + <ALT> + <T>
Starts "HMI Trace".

<SHIFT> + <ALT> + <T>
Exits "HMI Trace".

<ALT> + <S>
Opens the editor to enter Asian characters.

<ALT> + <Cursor up>
Moves the block start or block end up in the editor.

<ALT> + <Cursor down>
Moves the block start or block end down in the editor.

<DEL>
- Editing box
  Deletes the first character to the right of the cursor.
- Navigation
  Deletes all characters.
Introduction

2.2 Operator panel fronts

<DEL> + <CTRL>
- Editing box
  Deletes the first word to the right of the cursor.
- Navigation
  Deletes all characters.

<Spacebar>
- Editing window
  Inserts a space.
- Switches between several specified options in selection drop-down list boxes and in selection boxes.

<Plus>
- Opens a directory which contains the element.
- Increases the size of the graphic view for simulation and traces.

<Minus>
- Closes a directory which contains the element.
- Reduces the size of the graphic view for simulation and traces.

<Equals>
Opens the calculator in the entry fields.

<Asterisk>
Opens a directory with all of the subdirectories.

<Tilde>
Changes the sign of a number between plus and minus.

<INSERT>
- Opens an editing window in the insert mode. Pressing the key again, exits the window and the entries are undone.
- Opens a selection box and shows the selection possibilities.
- In the machining step program, enters an empty line for G code.
- Changes into the double editor or into the multi-channel view from the edit mode into the operating mode. You can return to the edit mode by pressing the key again.

<INSERT> + <SHIFT>
For G code programming, for a cycle call, activates or deactivates the edit mode.
<INPUT>
- Completes the input of a value in the entry field.
- Opens a directory or a program.
- Inserts an empty program block if the cursor is positioned at the end of a program block.
- Inserts a character to select a new line, and the program block is split up into two parts.
- Inserts a new line after the program block in the G code.
- Inserts a new line for G code in the machining step program.
- Changes into the double editor or into the multi-channel view from the edit mode into the operating mode. You can return to the edit mode by pressing the key again.

<ALARM> - only OP 010 and OP 010C
Calls the "Diagnosis" operating area.

<PROGRAM> - only OP 010 and OP 010C
Calls the "Program Manager" operating area.

<OFFSET> - only OP 010 and OP 010C
Calls the "Parameter" operating area.

<PROGRAM MANAGER> - only OP 010 and OP 010C
Calls the "Program Manager" operating area.

Menu forward key
Advances in the extended horizontal softkey bar.

Menu back key
Returns to the higher-level menu.

<MACHINE>
Calls the "Machine" operating area.

<MENU SELECT>
Calls the main menu to select the operating area.
2.3 Machine control panels

2.3.1 Overview

The machine tool can be equipped with a machine control panel by Siemens or with a specific machine control panel from the machine manufacturer.

You use the machine control panel to initiate actions on the machine tool such as traversing an axis or starting the machining of a workpiece.

2.3.2 Controls on the machine control panel

In this example, the MCP 483C IE machine control panel is used to illustrate the operator controls and displays of a Siemens machine control panel.

Overview

![Diagram of the machine control panel with labels](image)

1. EMERGENCY STOP button
2. Installation locations for control devices (d = 16 mm)
3. RESET
4. Program control
5. Operating modes, machine functions
6. User keys T1 to T15
7. Traversing axes with rapid traverse override and coordinate switchover
8. Spindle control with override switch
9. Feed control with override switch
10. Keyswitch (four positions)
Operator controls

**EMERGENCY STOP button**

Press the button in situations where:
- life is at risk.
- there is the danger of a machine or workpiece being damaged.

All drives will be stopped with the greatest possible braking torque.

**Machine manufacturer**

For additional responses to pressing the EMERGENCY STOP button, please refer to the machine manufacturer's instructions.

**RESET**

- Stop processing the current programs. The NCK control remains synchronized with the machine. It is in its initial state and ready for a new program run.
- Cancel alarm.

**Program control**

**<SINGLE BLOCK>**

Single block mode on/off.

**<CYCLE START>**

The key is also referred to as NC Start. Execution of a program is started.

**<CYCLE STOP>**

The key is also referred to as NC Stop. Execution of a program is stopped.

**Operating modes, machine functions**

**<JOG>**

Select "JOG" mode.

**<TEACH IN>**

Selecting the "Teach In" function

**<MDI>**

Select "MDI" mode.

**<AUTO>**

Select "AUTO" mode.

**<REPOS>**

Repositions, re-approaches the contour.
<REF POINT>
Approach reference point.

Inc <VAR> (Incremental Feed Variable)
Incremental mode with variable increment size.

Inc (incremental feed)
Incremental mode with predefined increment size of 1, ..., 10000 increments.

Machine manufacturer
A machine data code defines how the increment value is interpreted.

Traversing axes with rapid traverse override and coordinate switchover

Axis keys
Selects an axis.

Direction keys
Select the traversing direction.

<RAPID>
Traverse axis in rapid traverse while pressing the direction key.

<WCS MCS>
Switches between the workpiece coordinate system (WCS) and machine coordinate system (MCS).

Spindle control with override switch

<SPINDLE STOP>
Stop spindle.

<SPINDLE START>
Spindle is enabled.
Feed control with override switch

<FEED STOP>
Stops execution of the running program and shuts down axis drives.

<FEED START>
Enable for program execution in the current block and enable for ramp-up to the feedrate value specified by the program.
2.4 User interface

2.4.1 Screen layout

Overview

Active operating area and mode
Alarm/message line
Channel operational messages
Display for
- Active tool T
- Current feedrate F
- Active spindle with current state (S)
- Spindle utilization rate in percent
Vertical softkey bar
Display active G functions, all G functions, H functions and input window for different functions (e.g. skip blocks, program control)
Horizontal softkey bar
2.4.2 Status display

The status display includes the most important information about the current machine status and the status of the NCK. It also shows alarms as well as NC and PLC messages.

Depending on your operating area, the status display is made up of several lines:

- **Large status display**
  The status display is made up of three lines in the "Machine" operating area.

- **Small status display**
  In the "Parameter", "Program", "Program Manager", "Diagnosis" and "Start-up" operating areas, the status display consists of the first line from the large display.

### Status display of "Machine" operating area

**First line**

**Ctrl-Energy - power display**

<table>
<thead>
<tr>
<th>Display</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td>![icon]</td>
<td>The machine is not productive.</td>
</tr>
<tr>
<td>![icon]</td>
<td>The machine is productive and energy is being consumed.</td>
</tr>
<tr>
<td>![icon]</td>
<td>The machine is feeding energy back into the supply system.</td>
</tr>
</tbody>
</table>

The power display must be switched on in the status line.

**Note**

Information about configuration is available in the following reference:


### Active operating area

<table>
<thead>
<tr>
<th>Display</th>
<th>Meaning</th>
</tr>
</thead>
</table>
| ![icon] | "Machine" operating area  
With touch operation, you can change the operating area here. |
| ![icon] | "Parameter" operating area |
| ![icon] | "Program" operating area |
| ![icon] | "Program manager" operating area |
### Display | Meaning
--- | ---
![Diagnostics] | "Diagnostics" operating area
![Startup] | "Startup" operating area

#### Active mode or function

| Display | Meaning |
--- | ---|
![Jog] | "Jog" mode |
![MDA] | "MDA" mode |
![AUTO] | "AUTO" mode |
![Teach In] | "TEACH IN" function |
![REPOS] | "REPOS" function |
![REF POINT] | "REF POINT" function |

#### Alarms and messages

| Display | Meaning |
--- | ---|
![Alarm] | Alarm display. The alarm numbers are displayed in white lettering on a red background. The associated alarm text is shown in red lettering. An arrow indicates that several alarms are active. An acknowledgment symbol indicates that the alarm can be acknowledged or canceled. |
![NC or PLC message] | NC or PLC message. Message numbers and texts are shown in black lettering. An arrow indicates that several messages are active. |
![READY TO START] | Messages from NC programs do not have numbers and appear in green lettering. |

#### Second line

| Display | Meaning |
--- | ---|
[TEST_TEACHEN] | Program path and program name |

The displays in the second line can be configured.

**Machine manufacturer**

Please observe the information provided by the machine manufacturer.
### 2.4 User interface

<table>
<thead>
<tr>
<th>Display</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td>![CHAN1 RESET]</td>
<td>Channel status display. If the machine has several channels, then the channel name is also displayed. If there is only one channel, then only &quot;Reset&quot; is displayed as channel status. With touch operation, you can change the channel here.</td>
</tr>
<tr>
<td>![Dry Run]</td>
<td>Channel status display: The program was canceled with &quot;Reset&quot;. The program is executed. The program was interrupted with &quot;Stop&quot;.</td>
</tr>
<tr>
<td>![Dry Prt]</td>
<td>Display of active program controls: PRT no axis motion DRY dry run feedrate RG0: reduced rapid traverse M01: programmed stop 1 M101: programmed stop 2 (the designation is variable) SB1: Single block, coarse (program stops only after blocks that perform a machine function) SB2: Calculation block (program stops after each block) SB3: Single block, fine (program also only stops after blocks which perform a machine function in cycles) CST: configured stop (program stops at stop-relevant locations, which you defined before the program starts)</td>
</tr>
<tr>
<td>![Faulty NC block / user alarm]</td>
<td>Channel operational messages: Stop: An operator action is usually required. Wait: No operator action is required.</td>
</tr>
</tbody>
</table>

The machine manufacturer settings determine which program controls are displayed.

**Machine manufacturer**

Please observe the information provided by the machine manufacturer.

### 2.4.3 Actual value window

The actual values of the axes and their positions are displayed.

**Work/Machine**

The displayed coordinates are based on either the machine coordinate system or the workpiece coordinate system. The machine coordinate system (Machine), in contrast to the workpiece coordinate system (Work), does not take any work offsets into consideration.

You can use the "Machine actual values" softkey to toggle between the machine coordinate system and the workpiece coordinate system.
The actual value display of the positions can also refer to the SZS coordinate system (settable zero system). However the positions are still output in the Work.

The SZS coordinate system corresponds to the Work coordinate system, reduced by certain components ($P_TRAFRAME, $P_PFRAME, $P_ISO4FRAME, $P_CYCFRAME), which are set by the system when machining and are then reset again. By using the SZS coordinate system, jumps into the actual value display are avoided that would otherwise be caused by the additional components.

Machine manufacturer
Please observe the information provided by the machine manufacturer.

Maximize display
Press the “>>” and “Zoom act. val.” softkeys.

Display overview

<table>
<thead>
<tr>
<th>Display</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td>Header columns</td>
<td></td>
</tr>
<tr>
<td>Work/Machine</td>
<td>Display of axes in selected coordinate system.</td>
</tr>
<tr>
<td>Position</td>
<td>Position of displayed axes.</td>
</tr>
<tr>
<td>Display of distance-to-go</td>
<td>The distance-to-go for the current NC block is displayed while the program is running.</td>
</tr>
<tr>
<td>Feed/override</td>
<td>The feed acting on the axes, as well as the override, are displayed in the full-screen version.</td>
</tr>
<tr>
<td>REPOS offset</td>
<td>The distances traversed in manual mode are displayed. This information is only displayed when you are in the &quot;REPOS&quot; function.</td>
</tr>
<tr>
<td>Collision avoidance</td>
<td>Collision avoidance is activated for JOG, MDI and AUTO modes.</td>
</tr>
<tr>
<td></td>
<td>Collision avoidance is deactivated for JOG, MDI and AUTO modes.</td>
</tr>
<tr>
<td>Footer</td>
<td>Display of active work offsets and transformations. The T, F, S values are also displayed in the full-screen version.</td>
</tr>
</tbody>
</table>

See also
Set collision avoidance (Page 292)
2.4.4 T,F,S window

The most important data concerning the current tool, the feedrate (path feed or axis feed in JOG) and the spindle is displayed in the "T, F, S" window.

The "T, S, F" window displays several spindles with a maximum of two utilization indicators. The grinding power display is integrated in the spindle speed display. The power bar is located in the Z plane behind the speed value.

The following applies to the spindle display:

- The master spindle is always displayed
- The PLC specifies which tool spindle should be displayed
- The spindle number entered in the tool data is also the active tool spindle if the value is not zero

### Tool data

<table>
<thead>
<tr>
<th>Display</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td>T</td>
<td>Tool name Name of the current tool. Location Location number of the current tool.</td>
</tr>
<tr>
<td>D</td>
<td>Cutting edge of the current tool. The tool is displayed with the associated tool type icon corresponding to the actual coordinate system in the selected cutting edge position. If the tool is swiveled, then this is taken into account in the display of the cutting edge position. In DIN-ISO mode, the H number is displayed instead of the cutting edge number.</td>
</tr>
<tr>
<td>H</td>
<td>H number (tool offset data record for DIN-ISO mode). If there is a valid D number, this is also displayed.</td>
</tr>
<tr>
<td>Ø</td>
<td>Diameter of the current tool.</td>
</tr>
<tr>
<td>R</td>
<td>Radius of the current tool.</td>
</tr>
<tr>
<td>Z</td>
<td>Z value of the current tool.</td>
</tr>
<tr>
<td>X</td>
<td>X value of the current tool.</td>
</tr>
</tbody>
</table>

### Feed data

<table>
<thead>
<tr>
<th>Display</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td>F</td>
<td>Feed disable</td>
</tr>
</tbody>
</table>
| M       | Actual feed value If several axes traverse, it is displayed for:  
  - "JOG" mode: Axis feed for the traversing axis  
  - "MDA" and "AUTO" mode: Programmed axis feed |
| Rapid traverse | G0 is active |
### Spindle data

<table>
<thead>
<tr>
<th>Display</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.000</td>
<td>No feed is active</td>
</tr>
<tr>
<td>Override</td>
<td>Display as a percentage</td>
</tr>
</tbody>
</table>

#### Display of logical spindles

If the spindle converter is active, logical spindles are displayed in the workpiece coordinate system. When switching over to the machine coordinate system, the physical spindles are displayed.

**Note**

**Display of logical spindles**

If the spindle converter is active, logical spindles are displayed in the workpiece coordinate system. When switching over to the machine coordinate system, the physical spindles are displayed.

---

**Machine manufacturer**

Please refer to the machine manufacturer's specifications.

### 2.4.5 Current block display

The window of the current block display shows the program blocks currently being executed.

#### Display of current program

The following information is displayed in the running program:

- The workpiece name or program name is entered in the header line.
- The program block which is just being processed appears colored.
Display of the machining times

If you set that the machining times are to be recorded in the settings for automatic mode, the measured times are shown at the end of the line as follows:

<table>
<thead>
<tr>
<th>Display</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td>Light green background</td>
<td>Measured machining time of the program block (automatic mode)</td>
</tr>
<tr>
<td><img src="17.18" alt="Image" /></td>
<td></td>
</tr>
<tr>
<td>Green background</td>
<td>Measured machining time of the program group (automatic mode)</td>
</tr>
<tr>
<td><img src="19.47" alt="Image" /></td>
<td></td>
</tr>
<tr>
<td>Light blue background</td>
<td>Estimated machining time of the program block (simulation)</td>
</tr>
<tr>
<td><img src="17.31" alt="Image" /></td>
<td></td>
</tr>
<tr>
<td>Blue background</td>
<td>Estimated machining time of the program group (simulation)</td>
</tr>
<tr>
<td><img src="19.57" alt="Image" /></td>
<td></td>
</tr>
<tr>
<td>Yellow background</td>
<td>Wait time (automatic mode or simulation)</td>
</tr>
<tr>
<td><img src="4.53" alt="Image" /></td>
<td></td>
</tr>
</tbody>
</table>

Highlighting of selected G code commands or keywords

In the program editor settings, you can specify whether selected G code commands are to be highlighted in color. The following colors are used as standard:

<table>
<thead>
<tr>
<th>Display</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td>Blue font</td>
<td>D, S, F, T, M and H functions</td>
</tr>
<tr>
<td><img src="M30" alt="Image" /></td>
<td></td>
</tr>
<tr>
<td>Red font</td>
<td>&quot;G0&quot; motion command</td>
</tr>
<tr>
<td><img src="G0" alt="Image" /></td>
<td></td>
</tr>
<tr>
<td>Green font</td>
<td>&quot;G1&quot; motion command</td>
</tr>
<tr>
<td><img src="G1" alt="Image" /></td>
<td></td>
</tr>
<tr>
<td>Blue-green font</td>
<td>&quot;G2&quot; or &quot;G3&quot; motion command</td>
</tr>
<tr>
<td><img src="G3" alt="Image" /></td>
<td></td>
</tr>
<tr>
<td>Gray font</td>
<td>Comment</td>
</tr>
<tr>
<td>![Image](; Kommentar)</td>
<td></td>
</tr>
</tbody>
</table>

Machine manufacturer

You can define further highlight colors in the "sleditorwidget.ini" configuration file. Please refer to the machine manufacturer's instructions.
2.4 User interface

2.4.6 Operation via softkeys and buttons

Operating areas/operating modes

The user interface consists of different windows featuring eight horizontal and eight vertical softkeys.

You operate the softkeys with the keys next to the softkey bars.

You can display a new window or execute functions using the softkeys.

The operating software is sub-divided into six operating areas (machine, parameter, program, program manager, diagnosis, startup), three operating modes and four functions (JOG, MDI, AUTO, TEACH IN, REF. POINT, REPOS, single block).

Changing the operating area

Press the <MENU SELECT> key and select the desired operating area using the horizontal softkey bar.

You can call the "Machine" operating area directly using the key on the operator panel.

Changing the operating mode

You can select a mode or function directly with the keys on the machine control panel or the vertical softkeys in the main menu.

See also

Settings for the automatic mode (Page 207)
General keys and softkeys

When the symbol appears to the right of the dialog line on the user interface, you can change the horizontal softkey bar within an operating area. To do so, press the menu forward key.

The symbol indicates that you are in the expanded softkey bar.

Pressing the key again will take you back to the original horizontal softkey bar.

Use the ">>" softkey to open a new vertical softkey bar.

Use the "<<" softkey to return to the previous vertical softkey bar.

Use the "Return" softkey to close an open window.

Use the "Cancel" softkey to exit a window without accepting the entered values and return to the next highest window.

When you have entered all the necessary parameters in the parameter screen form correctly, you can close the window and save the parameters using the "Accept" softkey. The values you entered are applied to a program.

Use the "OK" softkey to initiate an action immediately, e.g. to rename or delete a program.

2.4.7 Entering or selecting parameters

When setting up the machine and during programming, you must enter various parameter values in the entry fields. The background color of the fields provides information on the status of the entry field.

Orange background The input field is selected
Light orange background The input field is in edit mode
Pink background The entered value is incorrect

Selecting parameters

Some parameters require you to select from a number of options in the input field. Fields of this type do not allow you to type in a value.

The selection symbol is displayed in the tooltip: 

Associated selection fields

There are selection fields for various parameters:

- Selection of units
- Changeover between absolute and incremental dimensions
Procedure

1. Keep pressing the <SELECT> key until the required setting or unit is selected.

   The <SELECT> key only works if there are several selection options available.

   - OR -

   Press the <INSERT> key.

   The selection options are displayed in a list.

2. Select the required setting using the <Cursor down> and <Cursor up> keys.

3. If required, enter a value in the associated input field.

4. Press the <INPUT> key to complete the parameter input.

Changing or calculating parameters

If you only want to change individual characters in an input field rather than overwriting the entire entry, switch to insertion mode.

In this mode, you can also enter simple calculation expressions, without having to explicitly call the calculator.

Note

Functions of the calculator

Function calls of the calculator are not available in the parameter screens of the cycles and functions in the "Program" operating area.
Enter the value or the calculation. Close the value entry using the <INPUT> key and the result is transferred into the field.

Accepting parameters
When you have correctly entered all necessary parameters, you can close the window and save your settings.
You cannot accept the parameters if they are incomplete or obviously erroneous. In this case, you can see from the dialog line which parameters are missing or were entered incorrectly.

Press the "OK" softkey.
- OR -
Press the "Accept" softkey.

2.4.8 Pocket calculator
The calculator allows you to calculate values for entry fields. It is possible to choose between a simple standard calculator and the extended view with mathematical functions.

Using the calculator
- You can simply use the calculator at the touch panel.
- Without a touch panel, you can use the calculator using the mouse.

Procedure

1. Position the cursor on the desired entry field.
2. Press the <=> key.
The calculator is displayed.
3. Press the <min> key if you would like to work with the standard calculator.
- OR -
Press the <extend> key to switch to the extended view.
4. Input the arithmetic statement.
You can use functions, arithmetic symbols, numbers, and commas.
5. Press the equals symbol on the calculator.
- OR -
Press the "Calculate" softkey.
6. Press the "Accept" softkey.
The calculated value is accepted and displayed in the entry field of the window.

2.4.9 Pocket calculator functions

The called operations continue to be displayed in the entry field of the calculator until the value is calculated. This allows you to subsequently modify entries and to nest functions.

The following save and delete functions are provided for modifications:

<table>
<thead>
<tr>
<th>Key</th>
<th>Function</th>
</tr>
</thead>
<tbody>
<tr>
<td>MS</td>
<td>Buffer value (Memory Save)</td>
</tr>
<tr>
<td>MR</td>
<td>Retrieve from buffer memory (Memory Recall)</td>
</tr>
<tr>
<td>MC</td>
<td>Delete buffer memory contents (Memory Clear)</td>
</tr>
<tr>
<td>←</td>
<td>Delete individual character (Backspace)</td>
</tr>
<tr>
<td>CE</td>
<td>Delete expression (Clear Element)</td>
</tr>
<tr>
<td>C</td>
<td>Delete all entries (Clear)</td>
</tr>
</tbody>
</table>

Nesting functions
Various possibilities are available for the nesting of functions as follows:

- Position the cursor within the bracket of the function call and supplement the argument with an additional function.
- Highlight the expression which is to be used as an argument in the entry line and then press the desired function key.

Percentage calculation
The calculator supports the calculation of a percentage, as well as changing of a basic value by a percentage. Press the following keys in this regard:

Example: Percentage
4  ^  50  %  =  2
Example: Change by percentage
4  +  50  %  =  6

Calculating trigonometric functions

1. Check whether the angles are specified in radians "RAD" or in degrees "DEG".
2. Press the "RAD" key to calculate the trigonometric functions in degrees "DEG".
The designation of the key changes to "DEG".
   - OR -
3. Press the "DEG" key to calculate the trigonometric functions in radian.
The designation of the key changes to "RAD".
4. Press the key for the desired trigonometric function, e.g. "SIN".

Further mathematical functions

Press the keys in the specified order:

Square number
\[ x^2 \] Number

Square root
\[ \sqrt{x} \] Number

Exponential function
Base number \[ \exp \] Exponent

Residue class calculation
Number \[ \mod \] Divider

Absolute value
\[ \text{ABS} \] Number

Integer component
\[ \text{INT} \] Number
Conversion between millimeters and inches

1. Enter the numerical value.
2. Press the "MM" key to convert inches to millimeters. The key is highlighted in blue.
   - OR -
3. Press the "INCH" key to convert millimeters to inches. The button is highlighted in blue.

3. Press the "=" key on the calculator. The calculated value is displayed in the entry field. The key for the unit is highlighted in gray once again.

2.4.10 Context menu

When you right-click, the context menu opens and provides the following functions:

- Cut
  Cut Ctrl+X

- Copy
  Copy Ctrl+C

- Paste
  Paste Ctrl+V

Program editor

Additional functions are available in the editor

- Undo the last change
  Undo Ctrl+Z

- Redo the changes that were undone
  Redo Ctrl+Y

Up to 50 changes can be undone.

2.4.11 Changing the user interface language

Procedure

1. Select the "Start-up" operating area.
2. Press the "Change language" softkey. The "Language selection" window opens. The language set last is selected.
3. Position the cursor on the desired language.
4. Press the "OK" softkey.

- OR -

Press the <INPUT> key.

The user interface changes to the selected language.

Note
Changing the language directly on the input screens
You can switch between the user interface languages available on the controller directly on the user interface by pressing the key combination <CTRL + L>.

2.4.12 Entering Chinese characters

Using the input editor IME (Input Method Editor), you can select Asian characters on classic panels (without touch operation) where you enter the phonetic notation. These characters are transferred into the user interface.

Note
Call the input editor with <Alt + S>
The input editor can only be called there where it is permissible to enter Asian characters.

The editor is available for the following Asian languages:
- Simplified Chinese
- Traditional Chinese

Input types

<table>
<thead>
<tr>
<th>Input type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Pinyin input</td>
<td>Latin letters are combined phonetically to denote the sound of the character. The editor lists all of the characters from the dictionary that can be selected.</td>
</tr>
<tr>
<td>Zhuyin input (only traditional Chinese)</td>
<td>Non-Latin letters are combined phonetically to denote the sound of the character. The editor lists all of the characters from the dictionary that can be selected.</td>
</tr>
<tr>
<td>Entering Latin letters</td>
<td>The characters that are entered are directly transferred into the input field, from where the editor was called.</td>
</tr>
</tbody>
</table>
Structure of the editor

1. Phonetic sound selection from the dictionary
2. Learning function of the dictionary
3. Listed characters
4. Phonetic sound input
5. Function selection

Figure 2-1  Example: Pinyin input

Figure 2-2  Example: Zhuyin input

Functions

- Pinyin input
- Entering Latin letters
- Editing the dictionary

Dictionaries

The simplified Chinese and traditional Chinese dictionaries that are supplied can be expanded:

- If you enter new phonetic notations, the editor creates a new line. The entered phonetic notation is broken down into known phonetic notations. Select the associated character for each component. The compiled characters are displayed in the additional line. Accept the new word into the dictionary and into the input field by pressing the <Input> key.

- Using any Unicode editor, you can enter new phonetic notations into a text file. These phonetic notations are imported into the dictionary the next time that the input editor is started.
2.4.12.1 Entering Asian characters

Precondition
The control has been switched over to Chinese.

Procedure

Editing characters using the Pinyin method
1. Open the screen form and position the cursor on the input field. Press the <Alt +S> keys. The editor is displayed.

2. Enter the desired phonetic notation using Latin letters. Use the upper input field for traditional Chinese.

3. Press the <Cursor down> key to reach the dictionary.

4. Keeping the <Cursor down> key pressed, displays all the entered phonetic notations and the associated selection characters.

5. Press the <BACKSPACE> softkey to delete entered phonetic notations.

6. Press the number key to insert the associated character. When a character is selected, the editor records the frequency with which it is selected for a specific phonetic notation and offers this character at the top of the list when the editor is next opened.

Editing characters using the Zhuyin method (only traditional Chinese)
1. Open the screen form and position the cursor on the input field. Press the <Alt +S> keys. The editor is displayed.

2. Enter the desired phonetic notation using the numerical block. Each number is assigned a certain number of letters that can be selected by pressing the numeric key one or several times.

3. Press the <Cursor down> key to reach the dictionary.

4. Keeping the <Cursor down> key pressed, displays all the entered phonetic notations and the associated selection characters.
5. Press the <BACKSPACE> softkey to delete entered phonetic notations.

6. To select the associated character, press the <cursor right> or <cursor left> keys.

7. Press the <input> key to enter the character.

2.4.12.2 Editing the dictionary

Learning function of the input editor

Requirement:
The control has been switched over to Chinese.
An unknown phonetic notation has been entered into the input editor.

1. The editor provides a further line in which the combined characters and phonetic notations are displayed.
The first part of the phonetic notation is displayed in the field for selecting the phonetic notation from the dictionary. Various characters are listed for this particular phonetic notation.

2. Press the number key to insert the associated character into the additional line.
The next part of the phonetic notation is displayed in the field for selecting the phonetic notation from the dictionary.

3. Repeat step 2 until the complete phonetic notation has been compiled.
Press the <TAB> key to toggle between the compiled phonetic notation field and the phonetic notation input.
Compiled characters are deleted using the <BACKSPACE> key.

4. Press the <input> key to transfer the compiled phonetic notation to the dictionary and the input field.

Importing a dictionary
A dictionary can now be generated using any Unicode editor by attaching the corresponding Chinese characters to the pinyin phonetic spelling. If the phonetic spelling contains several Chinese characters, then the line must not contain any additional match. If there are several matches for one phonetic spelling, then these must be specified in the dictionary line by line. Otherwise, several characters can be specified for each line.
The generated file should be saved in the UTF8 format under the name dictchs.txt (simplified Chinese) or dictcht.txt (traditional Chinese).

Line structure:
Pinyin phonetic spelling <TAB> Chinese characters <LF>
OR
Pinyin phonetic spelling <TAB> Chinese character1<TAB> Chinese character2 <TAB> … <LF>
<TAB> - tab key
<LF> - line break

Store the created dictionary in one of the following paths:
../user/sinumerik/hmi/ime/
../oem/sinumerik/hmi/ime/

When the Chinese editor is called the next time, it enters the content of the dictionary into the system dictionary.

Example:

<table>
<thead>
<tr>
<th>Pinyin</th>
<th>Chinese</th>
</tr>
</thead>
<tbody>
<tr>
<td>ai</td>
<td>哀 哀 哀 哀 哀</td>
</tr>
<tr>
<td>caise</td>
<td>彩色</td>
</tr>
<tr>
<td>hongse</td>
<td>紅色</td>
</tr>
<tr>
<td>huise</td>
<td>灰色</td>
</tr>
<tr>
<td>heli</td>
<td>河裏</td>
</tr>
<tr>
<td>zuishaowan</td>
<td>最好玩</td>
</tr>
</tbody>
</table>

2.4.13 Entering Korean characters

You can enter Korean characters in the input fields on classic panels (without touch operation) using the input editor IME (Input Method Editor).

Note
You require a special keyboard to enter Korean characters. If this is not available, then you can enter the characters using a matrix.
Korean keyboard

To enter Korean characters, you will need a keyboard with the keyboard assignment shown below. In terms of key layout, this keyboard is the equivalent of an English QWERTY keyboard and individual events must be grouped together to form syllables.

<table>
<thead>
<tr>
<th>1</th>
<th>2</th>
<th>3</th>
<th>4</th>
<th>5</th>
<th>6</th>
<th>7</th>
<th>8</th>
<th>9</th>
<th>0</th>
</tr>
</thead>
<tbody>
<tr>
<td>Tab</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>Backspace</td>
</tr>
<tr>
<td>Enter</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Ops</td>
<td>ASDFGHJKL</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Latic</td>
<td></td>
<td>ZXCVBNM</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Shift</td>
<td></td>
<td>Alt</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Structure of the editor

Functions

- **Matrix**: Editing characters using a matrix
- **Booktalk 2**: Editing characters using the keyboard
- **한글**: Entering Korean characters
- **A**: Entering Latin letters

Precondition

The control has been switched over to Korean.
Procedure

**Editing characters using the keyboard**

1. Open the screen form and position the cursor on the input field. Press the <Alt +S> keys. The editor is displayed.

2. Switch to the "Keyboard - Matrix" selection box.

3. Select the keyboard.

4. Switch to the function selection box.

5. Select Korean character input.

6. Enter the required characters.

7. Press the <input> key to enter the character into the input field.

**Editing characters using a matrix**

1. Open the screen form and position the cursor on the input field. Press the <Alt +S> keys. The editor is displayed.

2. Switch to the "Keyboard - Matrix" selection box.

3. Select the "matrix".

4. Switch to the function selection box.

5. Select Korean character input.

6. Enter the number of the line in which the required character is located. The line is highlighted in color.

7. Enter the number of the column in which the required character is located. The character will be briefly highlighted in color and then transferred to the Character field.
8. Press the <input> key to enter the character into the input field.

2.4.14 Protection levels

The input and modification of data in the control system is protected by passwords at sensitive places.

Access protection via protection levels

The input or modification of data for the following functions depends on the protection level setting:

- Tool offsets
- Work offsets
- Setting data
- Program creation / program editing

Note

Configuring access levels for softkeys

You have the option of providing softkeys with protection levels or completely hiding them.

References

For additional information, please refer to the following documentation:

SINUMERIK Operate Commissioning Manual

Softkeys

<table>
<thead>
<tr>
<th>Machine operating area</th>
<th>Protection level</th>
</tr>
</thead>
<tbody>
<tr>
<td>Synch. action</td>
<td>End user</td>
</tr>
<tr>
<td></td>
<td>(protection level 3)</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Parameters operating area</th>
<th>Protection level</th>
</tr>
</thead>
<tbody>
<tr>
<td>Tool management lists</td>
<td>Keyswitch 3</td>
</tr>
<tr>
<td></td>
<td>(protection level 4)</td>
</tr>
</tbody>
</table>
2.4.15 Cleaning mode

In cleaning mode, you can clean the user interface of the panel without inadvertently initiating touch functions.

When you activate cleaning mode, the system does not respond when you touch the screen. Switching over to another panel and entering data at the keyboard are deactivated. The display is dimmed. The progress bar shows the remaining time in seconds.

<table>
<thead>
<tr>
<th>Diagnostics operating area</th>
<th>Protection level</th>
</tr>
</thead>
<tbody>
<tr>
<td>Logbook</td>
<td>Keyswitch 3 (protection level 4)</td>
</tr>
<tr>
<td>Change</td>
<td>User (protection level 3)</td>
</tr>
<tr>
<td>New entry</td>
<td>User (protection level 3)</td>
</tr>
<tr>
<td>Startup complete</td>
<td>Manufacturer (protection level 1)</td>
</tr>
<tr>
<td>Machine installed</td>
<td>User (protection level 3)</td>
</tr>
<tr>
<td>Add HW comp</td>
<td>Service (protection level 2)</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Start-up operating area</th>
<th>Protection levels</th>
</tr>
</thead>
<tbody>
<tr>
<td>System data</td>
<td>End user (protection level 3)</td>
</tr>
<tr>
<td>Setup archive</td>
<td>Keyswitch 3 (protection level 4)</td>
</tr>
<tr>
<td>General M0</td>
<td>Keyswitch 3 (protection level 4)</td>
</tr>
<tr>
<td>Control Unit parameter</td>
<td>Keyswitch 3 (protection level 4)</td>
</tr>
<tr>
<td>Licenses</td>
<td>Keyswitch 3 (protection level 4)</td>
</tr>
<tr>
<td>Set M0 active (cf)</td>
<td>Keyswitch 3 (protection level 4)</td>
</tr>
<tr>
<td>Reset (po)</td>
<td>End user (protection level 3)</td>
</tr>
<tr>
<td>Change password</td>
<td>End user (protection level 3)</td>
</tr>
<tr>
<td>Delete password</td>
<td>End user (protection level 3)</td>
</tr>
</tbody>
</table>
Depending on the setting, cleaning mode lasts between 10 seconds and 1 minute. You can work as usual once this time has expired.

**Note**

Use a suitable cleaning agent to clean the screen.

**Procedure**

1. Select the "Start-up" operating area.

2. Press the "Cleaning mode for panel" softkey.
   The system switches into cleaning mode.

**2.4.16 Display live image from a camera**

In SINUMERIK Operate, you can display a live image from a camera.

- The camera image allows you to track remote processes and monitor difficult-to-access areas.
- You can also document and store machine states.
- You can create and configure up to two cameras.

**Machine manufacturer**

Please observe the information provided by the machine manufacturer.

You configure the cameras in the "Camera configuration" window, which you call with the "Camera" softkey in the "Setup" operating area.

**Requirement**

- The cameras used meet the required criteria for maximum resolution, frame rate, compression rate and network capability.
- You have configured the cameras used.

**Call live streaming**

You have the following options for calling live streaming:

- By clicking the "Camera" softkey in the "Machine" operating area
- By clicking the "Camera" softkey in the "Set up" operating area
• Via the "Camera 1" and "Camera 2" widgets in the sidescreen
• Via the "Camera 1" and "Camera 2" widgets in the Display Manager (840D sl only)

The live streaming is displayed in the first-called window.

To switch between the calling windows, live streaming must be restarted.

---

**Note**

As long as the "Camera configuration" window is open, live streaming can only be started from this window.

---

### 2.4.17 Online help in SINUMERIK Operate

A comprehensive context-sensitive online help is stored in the control system.

- A brief description is provided for each window and, if required, step-by-step instructions for the operating sequences.
- A detailed help is provided in the editor for every entered G code. You can also display all G functions and take over a selected command directly from the help into the editor.
- A help page with all parameters is provided on the input screen in the cycle programming.
- Lists of the machine data
- Lists of the setting data
- Lists of the drive parameters
- List of all alarms

---

### Procedure

**Calling context-sensitive online help**

1. You are in an arbitrary window of an operating area.
2. Press the <HELP> key or on an MF2 keyboard, the <F12> key.
   
   The help page of the currently selected window is opened in a subscreen.

3. Press the "Fullscreen" softkey to use the entire user interface for the display of the online help.
   
   Press the "Full screen" softkey again to return to the subscreen.

4. If further help is offered for the function or associated topics, position the cursor on the desired link and press the "Follow reference" softkey.
   
   The selected help page is displayed.

5. Press the "Back to reference" softkey to jump back to the previous help.

---

**Calling a topic in the table of contents**
1. Press the "Table of contents" softkey.
   Depending on which technology you are using, the Operating Manuals
   "Operator control Milling", "Operator control Turning" or "Operator con‐
   trol Universal" as well as the "Programming" Programming Manual are
   displayed.

2. Select the desired manual with the <Cursor down> and <Cursor up>
   keys.

3. Press the <Cursor right> or <INPUT> key or double-click to open the
   book and the section.

4. Navigate to the desired topic with the "Cursor down" key.

5. Press the <Follow reference> softkey or the <INPUT> key to display the
   help page for the selected topic.

6. Press the "Current topic" softkey to return to the original help.

Searching for a topic

1. Press the "Search" softkey.
   The "Search in Help for: " window appears.

2. Activate the "Full text " checkbox to search in all help pages.
   If the checkbox is not activated, a search is performed in the table of
   contents and in the index.

3. Enter the desired keyword in the "Text" field and press the "OK" softkey.
   If you enter the search term on the operator panel, replace an umlaut
   (accented character) by an asterisk (*) as dummy.
   All entered terms and sentences are sought with an AND operation. In
   this way, only documents and entries that satisfy all the search criteria
   are displayed.

4. Press the "Keyword index" softkey if you only want to display the index of
   the operating and programming manual.

Displaying alarm descriptions and machine data

1. If messages or alarms are pending in the "Alarms", "Messages" or "Alarm
   Log" window, position the cursor at the appropriate display and press
   <HELP> or key <F12>
   The associated alarm description is displayed.
2. If you are in the "Start-up" operating area in the windows for the display of the machine, setting and drive data, position the cursor on the desired machine data or drive parameter and press the <HELP> key or <F12> key.
   The associated data description is displayed.

### Displaying and inserting a G code command in the editor

1. A program is opened in the editor.
   Position the cursor on the desired G code command and press the <HELP> or the <F12> key.
   The associated G code description is displayed.

2. Press the "Display all G functions" softkey.

3. With the aid of the search function, select, for example, the desired G code command.

4. Press the "Transfer to editor" softkey.
   The selected G function is taken into the program at the cursor position.

5. Press the "Exit help" softkey again to close the help.
2.4 User interface
3.1 Multitouch panels

The "SINUMERIK Operate Generation 2" user interface has been optimized for multitouch operation. You can execute all actions by touch and finger gestures. Using SINUMERIK Operate is much quicker with touch operation and finger gestures.

Machine manufacturer
Please observe the information provided by the machine manufacturer.

The following operator panel fronts, handheld devices and SINUMERIK control systems can be operated with the "SINUMERIK Operate Generation 2" user interface:

- OP 015 black
- OP 019 black
- PPU 290.3
- HT 8
- SIMATIC ITC V3
- SIMATIC IFP
- SIMATIC panel IPC

Additional information
You can find more information on the topic of "user interface" at:

- SINUMERIK Operate Commissioning Manual (IM9), 840D sl
- SINUMERIK Operate Commissioning Manual (IH9), 828D

You can find additional information on multitouch panels at:

- Operating components - handheld devices, SINUMERIK 840 D sl (OP 015 black / 019 black)
- PPU and Components Manual, SINUMERIK 828D (PPU 290.3)
3.2 Touch-sensitive user interface

When using touch panels, wear thin gloves made of cotton or gloves for touch-sensitive glass user interfaces with capacitive touch function.

If you are using somewhat thicker gloves, then exert somewhat more pressure when using the touch panel.

Compatible gloves

You will operate the touch-sensitive glass user interface on the Operator panel optimally with the following gloves.

- Dermatril L
- Camatril Velours type 730
- Uvex Profas Profi ENB 20A
- Camapur Comfort Antistatic type 625
- Carex type 1505 / k (leather)
- Reusable gloves, medium, white, cotton: BM Polyco (RS order number 562-952)

Thicker work gloves

- Thermoplus KCL type 955
- KCL Men at Work type 301
- Camapur Comfort type 619
- Comasec PU (4342)
3.3 Finger gestures

Finger gestures

Tap
- Select window
- Select object (e.g. NC set)
- Activate entry field
  - Enter or overwrite value
  - Tap again to change the value

Tap with 2 fingers
- Call the shortcut menu (e.g. copy, paste)

Flick vertically with one finger
- Scroll in lists (e.g. programs, tools, zero points)
- Scroll in files (e.g. NC program)

Flick vertically with two fingers
- Page-scroll in lists (e.g. ZO)
- Page-scroll in files (e.g. NC programs)

Flick vertically with three fingers
- Scroll to the start or end of lists
- Scroll to the start or end of files
3.3 Finger gestures

- **Flick horizontally with one finger**
  - Scroll in lists with many columns

- **Spread**
  - Zoom in on graphic contents (e.g. simulation, mold making view)

- **Pinch**
  - Zoom out from graphic contents (e.g. simulation, mold making view)

- **Pan with one finger**
  - Move graphic contents (e.g. simulation, mold making view)
  - Move list contents

- **Pan with two fingers**
  - Rotate graphic contents (e.g. simulation, mold making view)

- **Tap and hold**
  - Open input fields to change
  - Activate or deactivate edit mode (e.g. current block display)
Tap and hold using 2 fingers

- Open cycles line by line to change (without input screen form)

Tapping with 2 index fingers – only for 840D sl

- Tap with two fingers simultaneously in the lower right- and left-hand corners to open the TCU menu. The menu has to be opened for service purposes.

Note

Flicking gestures with several fingers

The gestures only function reliably if you hold your fingers sufficiently far apart. The fingers should be at least 1 cm apart.
3.4 Multitouch user interface

3.4.1 Screen layout

Touch and gesture operator controls for SINUMERIK Operate with the "SINUMERIK Operate Generation 2" user interface.

1. Changing the channel
2. Cancel alarms
3. Function key block
4. Display next vertical softkey bar
5. Virtual keyboard

3.4.2 Function key block

<table>
<thead>
<tr>
<th>Operator control</th>
<th>Function</th>
</tr>
</thead>
<tbody>
<tr>
<td>Switch operating area</td>
<td>Tap the current operating area, and select the desired operating area from the operating area bar.</td>
</tr>
<tr>
<td>Switch operating mode</td>
<td>The operating mode is only displayed. To switch the operating mode, tap the operating area and select the operating area from the vertical softkey bar.</td>
</tr>
<tr>
<td>Operator control</td>
<td>Function</td>
</tr>
<tr>
<td>------------------</td>
<td>----------</td>
</tr>
<tr>
<td><img src="image" alt="Undo" /></td>
<td>Undo</td>
</tr>
<tr>
<td></td>
<td>Multiple changes are undone one by one. As soon as a change has been completed in an input field, this function is no longer available.</td>
</tr>
<tr>
<td><img src="image" alt="Restoring" /></td>
<td>Restoring</td>
</tr>
<tr>
<td></td>
<td>Multiple changes are restored one by one. As soon as a change has been completed in an input field, this function is no longer available.</td>
</tr>
<tr>
<td><img src="image" alt="Virtual keyboard" /></td>
<td>Virtual keyboard</td>
</tr>
<tr>
<td></td>
<td>Activates the virtual keyboard.</td>
</tr>
<tr>
<td><img src="image" alt="Calculator" /></td>
<td>Calculator</td>
</tr>
<tr>
<td></td>
<td>Displays a calculator.</td>
</tr>
<tr>
<td><img src="image" alt="Online help" /></td>
<td>Online help</td>
</tr>
<tr>
<td></td>
<td>Opens the online help.</td>
</tr>
<tr>
<td><img src="image" alt="Camera" /></td>
<td>Camera</td>
</tr>
<tr>
<td></td>
<td>Generates a screenshot.</td>
</tr>
</tbody>
</table>

### 3.4.3 Further operator touch controls

<table>
<thead>
<tr>
<th>Operator control</th>
<th>Function</th>
</tr>
</thead>
<tbody>
<tr>
<td><img src="image" alt="Next horizontal softkey bar" /></td>
<td>Advances to the next horizontal softkey bar. When page 2 of the menu is called, the arrow appears on the right.</td>
</tr>
<tr>
<td><img src="image" alt="Higher-level menu" /></td>
<td>Advances to the higher-level menu.</td>
</tr>
<tr>
<td><img src="image" alt="Next vertical softkey bar" /></td>
<td>Advances to the next vertical softkey bar.</td>
</tr>
<tr>
<td><img src="image" alt="Cancel alarm" /></td>
<td>Tapping the Cancel alarm symbol clears all queued cancel alarms.</td>
</tr>
<tr>
<td><img src="image" alt="Channel display" /></td>
<td>If a channel menu has been configured, it is displayed. Tapping the channel display in the status display switches you to the next channel.</td>
</tr>
</tbody>
</table>
3.4.4 Virtual keyboard

If you called the virtual keyboard using the function key block, then you have the option of adapting the key assignment using the shift keys.

| 1 | Shift key for uppercase and lowercase letters |
| 2 | Shift key for letters and special characters |
| 3 | Shift key for country-specific keyboard assignment |
| 4 | Shift key for full keyboard and numerical key block |

Hardware keyboard

If a real keyboard is connected, the icon of a minimized keyboard appears in place of the virtual keyboard.

Use the icon to open the virtual keyboard again.

3.4.5 Special "tilde" character

If the shift key for letters and special characters is pressed, the keyboard assignment changes to the special characters.

| 1 | <Tilde> |

In the Editor or in alphanumeric input fields, the special character <Tilde> is entered with the <Tilde> key. In numerical input fields, the <Tilde> key changes the sign of a number between plus and minus.
3.5 Expansion with side screen

3.5.1 Overview

Panels in widescreen format provide the possibility of using the extra area to display additional elements. In addition to the SINUMERIK Operate screen, displays and virtual keys are shown to provide faster information and operation.

This sidescreen must be activated. To do this, a navigation bar is displayed.

You can display the following elements above the navigation bar:

- Displaying (widgets)
- Virtual keys (pages)
  - ABC keyboard
  - MCP keys

Machine manufacturer
Please observe the information provided by the machine manufacturer.

Requirements

- A widescreen format multitouch panel (e.g. OP 015 black) is required to display widgets and pages.
- It is only possible to activate and configure a sidescreen when using the "SINUMERIK Operate Generation 2" user interface.

References

For information on activating the sidescreen and to configure the virtual keys, refer to the following literature:

- SINUMERIK Operate (IM9) / SINUMERIK 840D sl Commissioning Manual

3.5.2 Sidescreen with standard windows

When the sidescreen is activated, a navigation bar is shown on the left-hand side of the user interface.

This navigation bar can be used to switch directly to the desired operating area, and to show and hide the sidescreen.
### Navigation bar

<table>
<thead>
<tr>
<th>Operator control</th>
<th>Function</th>
</tr>
</thead>
<tbody>
<tr>
<td><img src="image1.png" alt="Image" /></td>
<td>Opens the &quot;Machinery&quot; operating area.</td>
</tr>
<tr>
<td><img src="image2.png" alt="Image" /></td>
<td>Opens the tool list in the &quot;Parameter&quot; operating area.</td>
</tr>
<tr>
<td><img src="image3.png" alt="Image" /></td>
<td>Opens the &quot;Work offset&quot; window in the &quot;Parameter&quot; operating area.</td>
</tr>
<tr>
<td><img src="image4.png" alt="Image" /></td>
<td>Opens the &quot;Program&quot; operating area.</td>
</tr>
<tr>
<td><img src="image5.png" alt="Image" /></td>
<td>Opens the &quot;Program manager&quot; operating area.</td>
</tr>
<tr>
<td><img src="image6.png" alt="Image" /></td>
<td>Opens the &quot;Diagnostics&quot; operating area.</td>
</tr>
<tr>
<td><img src="image7.png" alt="Image" /></td>
<td>Opens the &quot;Commissioning&quot; operating area.</td>
</tr>
<tr>
<td><img src="image8.png" alt="Image" /></td>
<td>Hides the sidescreen.</td>
</tr>
<tr>
<td><img src="image9.png" alt="Image" /></td>
<td>Shows the sidescreen.</td>
</tr>
</tbody>
</table>
3.5.3 Standard widgets

Open sidescreen

- Tap the arrow on the navigation bar to show the sidescreen.
  The standard widgets are displayed in minimized form as the header line.

Navigating in sidescreen

- To scroll through the list of widgets, swipe vertically with 1 finger.
  - OR -
- To return to the end or to the beginning of the list of widgets, swipe vertically with 3 fingers.

Open widgets

- To open a widget, tap the header line of the widget.

3.5.4 "Actual value" widget

The widget contains the position of the axes in the displayed coordinate system.
The distance-to-go for the current NC block is displayed while a program is running.
3.5.5 "Zero point" widget

The widget includes values of the active work offset for all configured axes. The approximate and detailed offset, as well as rotation, scaling and mirroring are displayed for each axis.

3.5.6 "Alarms" widget

The widget contains all the messages and alarms in the alarm list. The alarm number and description are displayed for every alarm. An acknowledgment symbol indicates how the alarm is acknowledged or canceled. Vertical scrolling is possible if multiple alarms are pending. Wipe horizontally to switch between alarms and messages.

3.5.7 "NC/PLC variables" widget

The "NC/PLC variables" widget displays the NC and PLC variables. The variable name, data type and value are shown for each variable.
Only those variables that are currently displayed in the "NC/PLC variables" screen in the "Diagnostics" operating area are shown. To update the list in the "NC/PLC variables" widget following a change in the "NC/PLC variables" screen in the "Diagnostics" operating area, collapse and expand the widget again.

Vertical scrolling is possible.

### 3.5.8 "Axle load" widget

The widget shows the load on all axles in a bar chart.

Up to 6 axes are displayed. Vertical scrolling is possible if multiple axes are present.

### 3.5.9 "Tool" widget

The widget contains the geometry and wear data for the active tool.

The following information is additionally displayed depending on the machine configuration:

- EC: Active location-dependent offset - setting up offset
- SC: Active location-dependent offset - additive offset
- TOFF: Programmed tool length offset in WCS coordinates, and programmed tool radius offset
- Override: Value of the overridden movements that were made in the individual tool directions

Grinding
Operating Manual, 06/2019, A5E44903521B AB
3.5.10 "Service life" widget

The widget displays the tool monitoring in relation to the following values:

- Operating time of tool (standard time monitoring)
- Finished workpieces (quantity monitoring)
- Tool wear (wear monitoring)

**Note**

Multiple cutting edges

If a tool has multiple cutting edges, the values of the edge with the lowest residual service life, quantity and wear is displayed.

It is possible to alternate between views by scrolling horizontally.

3.5.11 "Program runtime" widget

The widget contains the following data:

- Total runtime of the program
- Time remaining to end of program

This data is estimated for the first program run.

Additionally, progress of the program is visualized in a bar chart as a percentage.

3.5.12 Widget "Camera 1" and "Camera 2"

You can create up to two cameras for tracking remote processes and monitoring difficult-to-access areas.

Widgets "Camera 1" and "Camera 2" are used to display camera images. There is a dedicated widget for each camera.
If the particular camera has been configured, start streaming by opening the widget.

**Additional information** on activating widgets "Camera 1" and "Camera 2" is provided in the SINUMERIK Operate Commissioning Manual.

**3.5.13 Sidescreen with pages for the ABC keyboard and/or machine control panel**

Not only standard widgets but also pages with ABC keyboards and machine control panels can be configured in the sidescreen of a multitouch panel.
Configure ABC keyboard and MCP

If you configured ABC keyboard and MCP keys, then the navigation bar is extended for the sidescreen:

<table>
<thead>
<tr>
<th>Operator control</th>
<th>Function</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Display of standard widgets in the sidescreen</td>
</tr>
<tr>
<td></td>
<td>Display of an ABC keyboard on the sidescreen</td>
</tr>
<tr>
<td></td>
<td>Display of a machine control panel on the sidescreen</td>
</tr>
</tbody>
</table>

3.5.14 Example 1: ABC keyboard in the sidescreen

1. ABC keyboard
2. Key to display the keyboard
3.5.15 Example 2: Machine control panel in the sidescreen

① Machine control panel
② Key to display the machine control panel
3.6 SINUMERIK Operate Display Manager (840D sl only)

3.6.1 Overview

With a panel with full HD resolution (1920x1080), you have the possibility to work with the Display Manager.

The Display Manager allows you to see a lot of information at a glance.

With the Display Manager, the screen area is divided into several display areas.

In addition to SINUMERIK Operate, widgets, keyboards, a machine control panel and various applications are provided in the various areas.

Software option

The option "P81 – SINUMERIK Operate Display Manager" is required for the "SINUMERIK Operate Display Manager" function.

Note

The standard configuration of the Display Manager only supports the landscape orientation of the screen.

Additional information

For further information on the activation and configuration of the Display Manager, please refer to the following reference:

- SINUMERIK Operate (IM9) / SINUMERIK 840D sl Commissioning Manual

For further information on Full HD Panels, please refer to:

- Operator panel fronts manual: TOP 1500, TOP 1900, TOP 2200 / SINUMERIK 840D sl
3.6.2 Screen layout

The standard supply of a SINUMERIK Operate Display Manager offers the option of choosing between 3-display areas and 4-display areas.

1. SINUMERIK Operate with navigation bar for switchover of the operating area
2. Display area for standard widgets
3. Display area for applications (e.g. PDF)

3.6.3 Operator controls

The Display Manager is activated.

<table>
<thead>
<tr>
<th>Operator control</th>
<th>Function</th>
</tr>
</thead>
<tbody>
<tr>
<td>![Menu icon]</td>
<td><strong>Menu</strong>&lt;br&gt;Tap on the menu to select the desired arrangement of the display areas.</td>
</tr>
<tr>
<td>![3-display areas icon]</td>
<td><strong>3-display areas</strong>&lt;br&gt;- SINUMERIK Operate (with function block)&lt;br&gt;- Widget area&lt;br&gt;- Applications area (PDF, virtual keyboard)</td>
</tr>
</tbody>
</table>
## Operator control

### Function

<table>
<thead>
<tr>
<th>4-display areas</th>
</tr>
</thead>
<tbody>
<tr>
<td>- SINUMERIK Operate (with function block)</td>
</tr>
<tr>
<td>- Widget area</td>
</tr>
<tr>
<td>- Applications area (PDF, virtual keyboard)</td>
</tr>
<tr>
<td>- Area with virtual keyboard</td>
</tr>
</tbody>
</table>

### Mirroring display areas

Mirrors the selected arrangement of the display areas.

### Navigating in SINUMERIK Operate

Tap on the corresponding icon to directly open the desired operating area.

### Widgets

The following widgets are available by default:

- Actual values (Page 73)
- Zero point (Page 74)
- Tool (Page 75)
- Axle load (Page 75)
- Alarms (Page 74)
- Program runtime (Page 76)
- Service life (Page 76)
- NC/PLC variables (Page 74)

### PDF

Opens the PDF stored here.

The following functions are available in the PDF display:

- Open (CTRL + O)
- Mark (CTRL + A)
- Copy (CTRL + C)
- Go to (CTRL + G)
- Search (CTRL + F)
- Display and hide bookmarks (CTRL + B)

Alternatively, use the PDF display via the animated toolbar at the top left.

You optimize the read view using the following finger gestures:

- Double tap: Adapt width:
- <CTRL> + double tap: Adapt height:
### Operator control

<table>
<thead>
<tr>
<th>Operator control</th>
<th>Function</th>
</tr>
</thead>
</table>
| ![Keyboard](image1) | **Virtual keyboard**  
Displays a QWERTY keyboard in the display area for applications as well as in the 4th display area below SINUMERIK Operate. If the virtual keyboard is selected while the display area is maximized, the keyboard opens as a pop-up. The keyboard can be moved on the display as required by means of touch operation. |
| ![Camera](image2) | **Camera**  
Live streaming of the configured camera:  
- 1 ● Live streaming camera 1  
- 2 ● Live streaming camera 2  
- 1-2 ● Live streaming camera 1 and camera 2  
If a camera has been configured, you can directly view the relevant streaming process. If a camera configuration is changed or an issue with connectivity occurs, reboot the system to activate the streaming process on the camera. |
| ![Maximize](image3) | **Maximizing the display area**  
Enlarges the area with SINUMERIK Operate and the area for the applications to the full dimensions of the panel. |
| ![Minimize](image4) | **Minimizing the display area**  
The area with SINUMERIK Operate and the area for the applications are reduced back to their original size. |
| ![MCP](image5) | **Machine control panel**  
Shows a machine control panel.  
**Note:** Please observe the information provided by the machine manufacturer. |

### See also

Widget “Camera 1” and “Camera 2” (Page 76)
4.1 Switching on and switching off

Startup

When the control starts up, the main screen opens according to the operating mode specified by the machine manufacturer. This is usually the main screen for the "REF POINT" function.

Machine manufacturer
Please observe the information provided by the machine manufacturer.
4.2 Approaching a reference point

4.2.1 Referencing axes

Your machine tool can be equipped with an absolute or incremental path measuring system. An axis with incremental path measuring system must be referenced after the controller has been switched on – however, an absolute path measuring system does not have to be referenced.

For the incremental path measuring system, all the machine axes must therefore first approach a reference point, the coordinates of which are known to be relative to the machine zero-point.

Sequence

Prior to the approach, the axes must be in a position from where they can approach the reference point without a collision.

The axes can also all approach the reference point simultaneously, depending on the manufacturer’s settings.

**Machine manufacturer**

Please refer to the machine manufacturer's specifications.

### NOTICE

**Risk of collision**

If the axes are not in a collision-free position, you must first traverse them to safe positions in "JOG" or "MDI" mode.

You must follow the axis motions directly on the machine!

Ignore the actual value display until the axes have been referenced!

The software limit switches are not active!

Procedure

1. Press the <JOG> key.
2. Press the <REF. POINT> key.
3. Select the axis to be traversed.
4. Press the <-> or <+> key.
   The selected axis moves to the reference point.
   If you have pressed the wrong direction key, the action is not accepted and the axes do not move.

   A symbol is shown next to the axis if it has been referenced.

   The axis is referenced as soon as the reference point is reached. The actual value display is set to the reference point value.
   From now on, path limits, such as software limit switches, are active.
   End the function via the machine control panel by selecting operating mode "AUTO" or "JOG".

4.2.2 User agreement

   If you are using Safety Integrated (SI) on your machine, you will need to confirm that the current displayed position of an axis corresponds to its actual position on the machine when you reference an axis. Your confirmation is the requirement for the availability of other Safety Integrated functions.

   You can only give your user agreement for an axis after it has approached the reference point.
   The displayed axis position always refers to the machine coordinate system (Machine).

   Option
   User agreement with Safety Integrated is only possible with a software option.

Procedure

1. Select the "Machine" operating area.

2. Press the <REF POINT> key.

3. Select the axis to be traversed.

4. Press the <-> or <+> key.
   The selected axis moves to the reference point and stops. The coordinate of the reference point is displayed.
   The axis is marked with 🔄.
5. Press the "User enable" softkey. The "User Agreement" window opens. It shows a list of all machine axes with their current position and SI position.

6. Position the cursor in the "Acknowledgement" field for the axis in question.

7. Activate the acknowledgement with the <SELECT> key.

The selected axis is marked with an "x" meaning "safely referenced" in the "Acknowledgement" column.

By pressing the <SELECT> key again, you deactivate the acknowledgement again.
4.3 Operating modes

4.3.1 Operating modes

You can work in three different operating modes.

"JOG" mode

"JOG" mode is used for the following preparatory actions:

- Approach reference point, i.e. the machine axis is referenced
- Preparing a machine for executing a program in automatic mode, i.e. measuring tools, measuring the workpiece and, if necessary, defining the work offsets used in the program
- Traverse axes, e.g. during a program interrupt
- Positioning axes

Select "JOG"

Press the <JOG> key.

The following functions are available in "JOG" mode:

- "REF POINT"
- "REPOS"

"REF POINT" function

The "REF POINT" function is used to synchronize the control and the machine. For this purpose, you approach the reference point in "JOG" mode.

Selecting "REF POINT"

Press the <REF POINT> key.

"REPOS" function

The "REPOS" function is used for repositioning to a defined position. After a program interrupt (e.g. to correct tool wear values), move the tool away from the contour in "JOG" mode.

The path differences traversed in "JOG" mode are displayed in the actual value window as the "REPOS" offset.

"REPOS" offsets can be displayed in the machine coordinate system (MCS) or workpiece coordinate system (WCS).
Select "REPOS"

Press the <REPOS> key.

"MDI" mode (Manual Data Input)

In "MDI" mode, you can enter and execute G code commands non-modally to set up the machine or to perform a single action.

Selecting "MDI"

Press the <MDI> key.

The "TEACH IN" function is available in "MDI" mode.

"TEACH IN" function

With the "TEACH IN" function, you can create, edit and execute part programs (main programs and subroutines) for motion sequences or simple workpieces by approaching and saving positions.

Selecting "Teach In"

Press the <TEACH IN> key.

"AUTO" mode

In automatic mode, you can execute a program completely or only partially.

Select "AUTO"

Press the <AUTO> key.

The "Single block" function is available in "AUTO" mode.

"Single block" function

You can execute a program block-by-block with the "Single block" function.

Select "Single block"

Press the <SINGLE BLOCK> key.
4.3.2 Modes groups and channels

Every channel behaves like an independent NC. A maximum of one part program can be processed per channel.

- Control with 1 channel
  One mode group exists.
- Control with several channels
  Channels can be grouped to form several "mode groups."

Example

Control with 4 channels, where machining is carried out in 2 channels and 2 other channels are used to control the transport of the new workpieces.

Mode group 1 channel 1 (machining)
Channel 2 (transport)
Mode group 2 channel 3 (machining)
Channel 4 (transport)

Mode groups (MGs)

Technologically-related channels can be combined to form a mode group.
Axes and spindles of the same mode group can be controlled by one or more channels.
An operating mode group is in one of "Automatic", "JOG" or "MDI" operating modes, i.e., several channels of an operating mode group can never assume different operating modes.

4.3.3 Channel switchover

It is possible to switch between channels when several are in use. Since individual channels may be assigned to different mode groups, a channel switchover command is also an implicit mode switchover command.

When a channel menu is available, all of the channels are displayed on softkeys and can be switched over.

Changing the channel

Press the <CHANNEL> key.

The channel changes over to the next channel.
- OR -
If the channel menu is available, a softkey bar is displayed. The active channel is highlighted.
Another channel can be selected by pressing one of the other softkeys.
References

SINUMERIK Operate Commissioning Manual
4.4 Settings for the machine

4.4.1 Switching over the coordinate system (MCS/WCS)

The coordinates in the actual value display are relative to either the machine coordinate system or the workpiece coordinate system.

By default, the workpiece coordinate system is set as a reference for the actual value display.

The machine coordinate system (MCS), in contrast to the workpiece coordinate system (WCS), does not take into account any zero offsets, tool offsets and coordinate rotation.

Procedure

1. Select the "Machine" operating area.
2. Press the <JOG> or <AUTO> key.

The machine coordinate system is selected.

The title of the actual value window changes in the MCS.

Machine manufacturer

The softkey to changeover the coordinate system can be hidden. Please refer to the machine manufacturer's specifications.

4.4.2 Switching the unit of measurement

You can set millimeters or inches as the unit of measurement for the machine. Switching the unit of measurement always applies to the entire machine. All required information is automatically converted to the new unit of measurement, for example:

- Positions
- Tool offsets
- Zero offsets
The following conditions must be met before you can switch between units of measurement:

- The corresponding machine data are set.
- All channels are in the reset state.
- The axes are not being traversed via "JOG", "DRF", and the "PLC".
- Constant grinding wheel peripheral speed (GWPS) is not active.

Machine manufacturer
Please observe the information provided by the machine manufacturer.

Further information
Further information about switching the measurement unit can be found in the Basic Functions function manual.

Procedure

1. Select the mode <JOG> or <AUTO> in the "Machine" operating area.

2. Press the menu forward key and the "Settings" softkey. A new vertical softkey bar appears.

3. Press the "Switch to inch" softkey. A prompt asks you whether you really want to switch over the unit of measurement.

4. Press the "OK" softkey. The softkey label changes to "Switch to metric". The unit of measurement applies to the entire machine.

5. Press the "Switch to metric" softkey to set the unit of measurement of the machine to metric again.

See also
Default settings for manual mode (Page 141)
4.4.3 Setting the zero offset

You can enter a new position value in the actual value display for individual axes when a settable zero offset is active.

The difference between the position value in the machine coordinate system MCS and the new position value in the workpiece coordinate system WCS is saved permanently in the currently active zero offset (e.g. G54).

Relative actual value

Further, you also have the possibility of entering position values in the relative coordinate system.

Note

The new actual value is only displayed. The relative actual value has no effect on the axis positions and the active zero offset.

Resetting the relative actual value

Press the "Delete REL" softkey.

The actual values are deleted.

The softkeys to set the zero point in the relative coordinate system are only available if the corresponding machine data is set.

Machine manufacturer

Please refer to the machine manufacturer's specifications.

Precondition

The controller is in the workpiece coordinate system.

The actual value is set in the reset state.

Note

Setting the ZO in the Stop state

If you enter the new actual value in the Stop state, the changes made are only visible and only take effect when the program is continued.
Setting up the machine

4.4 Settings for the machine

Procedure

1. Select the "JOG" mode in the "Machine" operating area.

2. Press the "Set ZO" softkey.
   - OR -
   Press the ">>, "REL act. vals" and "Set REL" softkeys to set position values in the relative coordinate system.

3. Enter the new required position value for X, Y or Z directly in the actual value display (you can toggle between the axes with the cursor keys) and press the "Input" key to confirm the entries.
   - OR -
   Press softkeys "X=0","Y=0" or "Z=0" to set the relevant position to zero.

   ... 

   - OR -
   Press softkey "X=Y=Z=0" to set all axis positions to zero simultaneously.

Resetting the actual value

Press the "Delete active ZO" softkey.
The offset is deleted permanently.

Note

Irreversible active zero offset
The current active zero offset is irreversibly deleted by this action.
4.5 Zero offsets

4.5.1 Zero offsets

After reference point approach, the actual value display for the axis coordinates is based on the machine zero (M) of the machine coordinate system (Machine). The program for machining the workpiece, however, is based on the workpiece zero (W) of the workpiece coordinate system (Work). The machine zero and workpiece zero are not necessarily identical. The distance between the machine zero and the workpiece zero depends on the workpiece type and how it is clamped. This zero offset is taken into account during execution of the program and can be a combination of different offsets.

After reference point approach, the actual value display for the axis coordinates is based on the machine zero of the machine coordinate system (Machine).

The actual value display of the positions can also refer to the SZS coordinate system (Settable Zero System). The position of the active tool relative to the workpiece zero is displayed.

When the machine zero is not identical to the workpiece zero, at least one offset (base offset or zero offset) exists in which the position of the workpiece zero is saved.
4.5 Zero offsets

**Base offset**
The base offset is a zero offset that is always active. If you have not defined a base offset, its value will be zero. The base offset is specified in the "Zero Offset - Base" window.

**Settable Zero System (SZS)**
The SZS (Settable Zero System) corresponds to the WCS transformed by the programmable frame (e.g. $P_PFFRAME, $PCYCFRAME, $P_TOOLFRAME and $P_WPFRAME).

**Basic zero system (BZS)**
The BZS (Basic Zero System) includes not only the frames of the SZS, but also the current settable frame ($P_IFRAME and $P_GFRAME).

**Coarse and fine offsets**
Every zero offset (G54 to G57, G505 to G599) consists of a coarse offset and a fine offset. You can call the zero offsets from any program (coarse and fine offsets are added together).

You can save the workpiece zero, for example, in the coarse offset, and then store the offset that occurs when a new workpiece is clamped between the old and the new workpiece zero in the fine offset.

---

**Note**

**Deselect fine offset (only 840D sl)**
You have the option of deselecting the fine offset using machine data MD18600
$MN_MM_FRAME_FINE_TRANS

---

4.5.2 Display active zero offset

The following work offsets are displayed in the "Zero Offset - Active" window:

- Work offsets, for which active offsets are included, or for which values are entered.
- Settable work offsets
- Seat-related fine offsets
- Total work offset

This window is generally used only for monitoring.
The availability of the offsets depends on the setting.

**Machine manufacturer**

Please refer to the machine manufacturer's specifications.
Procedure

1. Select the "Parameter" operating area.

2. Press the "Work offset" softkey.
   The "Work Offset - Active" window is opened.

Note

Further details on work offsets

If you would like to see further details about the specified offsets or if you would like to change values for the rotation, scaling or mirroring, press the "Details" softkey.

4.5.3 Displaying the zero offset "overview"

The active offsets or system offsets are displayed for all set-up axes in the "Work Offset - Overview" window.

In addition to the offset (course and fine), the rotation, scaling and mirroring defined using this are also displayed.

This window is generally used only for monitoring.

Display of active work offsets

<table>
<thead>
<tr>
<th>Work offsets</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>DRF</td>
<td>Displays the handwheel axis offset.</td>
</tr>
<tr>
<td>Rotary table reference</td>
<td>Displays the additional work offsets programmed with $P_PARTFRAME.</td>
</tr>
<tr>
<td>Basic reference</td>
<td>Displays the additional work offsets programmed with $P_SETFRAME. Access to the system offsets is protected via a keyswitch.</td>
</tr>
<tr>
<td>External WO frame</td>
<td>Displays the additional work offsets programmed with $P_EXTFRAME.</td>
</tr>
<tr>
<td>Total base WO</td>
<td>Displays all effective basis offsets.</td>
</tr>
<tr>
<td>G500</td>
<td>Displays the work offsets activated with G54 - G599. Under certain circumstances, you can change the data using &quot;Set WO&quot;, i.e. you can correct a zero point that has been set.</td>
</tr>
<tr>
<td>Tool reference</td>
<td>Displays the additional work offsets programmed with $P_TOOLFRAME.</td>
</tr>
<tr>
<td>Workpiece reference</td>
<td>Displays the additional work offsets programmed with $P_WPFRAME.</td>
</tr>
<tr>
<td>Programmed WO</td>
<td>Displays the additional work offsets programmed with $P_PFRAME.</td>
</tr>
<tr>
<td>Cycle reference</td>
<td>Displays the additional work offsets programmed with $P_CYCFRAME.</td>
</tr>
</tbody>
</table>
4.5 Zero offsets

<table>
<thead>
<tr>
<th>Work offsets</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>GFRAME0 (seat-related work offset)</td>
<td>Displays the additional work offsets programmed with $P_GFRAME.</td>
</tr>
<tr>
<td>Total WO</td>
<td>Displays the active work offset, resulting from the total of all work offsets.</td>
</tr>
</tbody>
</table>

**Procedure**

1. Select the "Parameter" operating area.

2. Press the "Work offset" and "Overview" softkeys. The "Work offset - Overview" window opens.

**4.5.4 Displaying and editing base zero offset**

The defined channel-specific and global base offsets, divided into coarse and fine offsets, are displayed for all set-up axes in the "Zero offset - Base" window.

**Machine manufacturer**

Please refer to the machine manufacturer's specifications.

**Procedure**

1. Select the "Parameter" operating area.

2. Press the "Zero offset" softkey.

3. Press the "Base" softkey. The "Zero Offset - Base" window is opened.

4. You can edit the values directly in the table.

**Note**

**Activate base offsets**

The offsets specified here are immediately active.
4.5.5 Displaying and editing settable zero offset

All settable offsets, divided into coarse and fine offsets, are displayed in the "Work offset - G54...G599" window.

Rotation, scaling and mirroring are displayed.

Procedure

1. Select the "Parameter" operating area.
2. Press the "Work offset" softkey.
3. Press the "G54 ... G599" softkey.
   The "Work offset - G54 ... G599 [mm]" window opens.

Note

The labeling of the softkeys for the settable work offsets varies, i.e. the settable work offsets configured on the machine are displayed (examples: G54 ... G57, G54 ... G505, G54 ... G599).

Please observe the information provided by the machine manufacturer.

4. You can edit the values directly in the table.

Note

Activate settable zero offsets

The settable zero offsets must first be selected in the program before they have an impact.

4.5.6 Displaying and editing seat-related fine offset

In the window "Work offset - GFrame1 ... GFrame..." all position-related offset values (seat offsets) are displayed.

Translatory and rotary offsets are displayed.

Procedure

1. Select the "Parameter" operating area.
2. Press the "Work offset" softkey.
3. Press softkey "GFrame1...GFrame2". Window "Work offset - GFrame1 ... GFrame2" is opened.

Note:
The labeling of the softkeys for the seat-related fine offsets varies, i.e. the seat-dependent work offsets configured at the machine are displayed (examples: GFRAME1…GFRAME1, GFRAME1…GFRAME2, GFRAME1…GFRAME100).
Here, please observe the information provided by the machine manufacturer.

4. You can edit the values directly in the table.

Note
Activating seat-related fine offsets
The seat-related work offsets must first be selected in the program before they are activated.

4.5.7 Displaying and editing details of the zero offsets
For each zero offset, you can display and edit all data for all axes. You can also delete zero offsets.

For every axis, values for the following data will be displayed:

- Coarse and fine offsets
- Rotation
- Scaling
- Mirroring

Machine manufacturer
Please refer to the machine manufacturer’s specifications.

Note
Settings for rotation, scaling and mirroring are specified here and can only be changed here.

Tool details
You can display the following details for the tool and wear data for tools:

- TC
- Adapter dimension
- Length / length wear
- EC setup correction
- SC sum correction
- Total length
- Radius / radius wear

You can also change the display of the tool correction values between the Machine Coordinate System and the Workpiece Coordinate System.

Machine manufacturer
Please refer to the machine manufacturer's specifications.

Procedure

1. Select the "Parameter" operating area.
2. Press the "Zero offset" softkey.
3. Press the "Active", "Base" or "G54…G599" softkey.
   The corresponding window opens.
4. Place the cursor on the desired zero offset to view its details.
5. Press the "Details" softkey.
   A window opens, depending on the selected zero offset, e.g. "Zero Offset - Details: G54 to G599".
6. You can edit the values directly in the table.
   - OR -
   Press the "Clear offset" softkey to reset all entered values.

Press the "ZO +" or "ZO -" softkey to select the next or previous offset, respectively, within the selected area ("Active", "Base", "G54 to G599") without first having to switch to the overview window.

If you have reached the end of the range (e.g. G599), you will switch automatically to the beginning of the range (e.g. G54).
4.5 Zero offsets

These value changes are available in the part program immediately or after "Reset".

Machine manufacturer

Please refer to the machine manufacturer's specifications.

Press the "Back" softkey to close the window.

4.5.8 Deleting a zero offset

You have the option of deleting work offsets. This resets the entered values.

Procedure

1. Select the "Parameter" operating area.
2. Press the "Work offset" softkey.
3. Press the "Overview", "Basis" or "G54...G599" softkey.
4. Press the "Details" softkey.
5. Position the cursor on the work offset you would like to delete.
6. Press the "Clear offset" softkey.
   A confirmation prompt is displayed as to whether you really want to delete the work offset.
7. Press the "OK" softkey to confirm that you wish to delete the work offset.

4.5.9 Deleting seat-related fine offsets

You have the option of deleting seat-dependent fine offsets after you have changed a tool. The values that have been entered are then set to zero.
Procedure

1. Select the "Parameter" operating area.

2. Press the "Work offset" softkey.

3. Press softkey "GFrame1...G-Frame2".
   Note:
   The labeling of the softkeys for the seat-related fine offsets varies, i.e. the seat-dependent work offsets configured at the machine are displayed (examples: GFRAME1...GFRAME1, GFRAME1...GFRAME2, GFRAME1...GFRAME100).
   Here, please observe the information provided by the machine manufacturer.

4. In the "Work offset - GFrame1 ... GFrame2" window in the table, set all values of the seat-related fine offsets to zero.
   - OR -
   Press the "Delete all" softkey.
   A confirmation prompt is displayed as to whether you really want to delete the seat-related fine offsets.
   Note:
   The "Delete all" softkey is only available if seat-related work offsets have been set up.

5. Press the "OK" softkey to confirm that you wish to delete the work offset.
4.6 Measuring a tool

4.6.1 Cylindrical grinding

4.6.1.1 Overview

The geometries of the machining tool must be taken into consideration when executing a part program. These are stored as tool offset data in the tool list. Each time the tool is called, the control considers the tool offset data.

When programming the part program, you only need to enter the workpiece dimensions from the production drawing. The control then independently calculates the individual tool path.

Measuring grinding wheels and dressers

Tool offset data, i.e. the lengths or positions of the tools, are determined by taking manual measurements (scratching).

When measuring manually, traverse the tool manually to a selected reference point in order to determine the tool dimensions and positions in the X and Z directions. The control then calculates the tool offset data from the position of the tool carrier reference point and the approached reference point.

Measuring the reference points for the grinding wheel

For manual measurements of a grinding tool, you have the option of selecting the following reference points:

- Workpiece (with zero offset)
- Dresser (with zero offset as dressing tool)

Machine manufacturer

Please refer to the machine manufacturer's specifications.

Measuring the reference points for the dresser

Use the grinding wheel as reference point for manually measuring a dresser.

4.6.1.2 Measure grinding tool manually with the work piece reference point

Reference point

The workpiece edge serves as reference point when measuring length X and length Z. You specify the position of the workpiece edge during the measurement.
Procedure

1. Select "JOG" mode in the "Machine" operating area.

2. Press the "Meas. tool" softkey.

3. Press the "Measure wheel" softkey.

4. Press the "Select tool" softkey.

The "Tool Selection" window opens.

5. In the "Tool Selection" window, select the grinding tool that you want to measure and press the "OK" softkey.

The cutting edge location must be entered in the tool list.

- OR -

Press the "Tool list" softkey, select in the tool list the grinding tool that you want to measure and press the "In manual" softkey.

The tool is transferred to the "Measure: "Grinding Wheel" window.

6. Select the "Workpiece" entry in the "Reference point" selection field.

7. Press the "X" or "Z" softkey, depending on which tool length you want to measure.

8. Enter the position of the workpiece edge in X0 and Z0.

If no value is entered for X0 or Z0, the value is taken from the actual value display.

9. Scratch the required edge using the tool.

10. Press the "Set length" softkey.

The tool length is calculated automatically and entered in the tool list. The cutting edge location is considered automatically.

Note

Active grinding tool

Tool measurement is possible only with an active grinding tool.
4.6.1.3 Measure grinding tool manually with the dresser reference point

Reference point

A dresser serves as reference point for the measurement of length X or Z. The reference point of the dresser can be represented by a zero offset or a dressing tool. This setting is permanently stored in the machine data and specified by the machine manufacturer.

Machine manufacturer

Please refer to the machine manufacturer’s specifications.

Procedure

1. Select “JOG” mode in the "Machine" operating area.
2. Press the "Meas. tool" softkey.
3. Press the "Measure wheel" softkey.
4. Press the "Select tool" softkey.
   The “Tool Selection” window opens.
5. In the "Tool Selection" window, select the grinding tool that you want to measure and press the "OK" softkey.
   The cutting edge location must be entered in the tool list.
   - OR -
   Press the "Tool list" softkey, select in the tool list the grinding tool that you want to measure and press the "In manual" softkey.
   The tool is transferred to the "Measure: "Grinding Wheel" window.
6. Select the "Dresser" entry in the "Reference point" selection field.
7. Position the cursor in the “TR” field, press the "Select dresser" softkey, select the dresser for measuring the tool length and press the "OK" softkey.
   - OR -
   Position the cursor in the "Zero offset" field and press the "Select ZO" softkey.
Select in the "Zero Offset - G54 ... G509" window the desired zero offset and press the "In manual" softkey.

8. Press the "X" or "Z" softkey, depending on which tool length you want to measure.

9. Scratch the dresser using the tool.

10. Press the "Set length" softkey.
    The tool length is calculated automatically and entered in the tool list.
    The cutting edge location is considered automatically.

Note
Active grinding tool
Tool measurement is possible only with an active grinding tool.

4.6.1.4 Measuring the dressing tool manually with the grinding tool reference point

Reference point
A grinding wheel serves as reference point for the measurement of length X or Z.

Procedure

1. Select "JOG" mode in the "Machine" operating area.

2. Press the "Meas. tool" softkey.

3. Press the "Measure dresser" softkey.

4. Press the "Select dresser" softkey.
   The "Tool Selection" window opens.

5. In the "Tool Selection" window, select the dressing tool that you want to measure and press the "OK" softkey.
   The cutting edge location must be entered in the tool list.
   - OR -
Press the "Tool list" softkey, select in the tool list the dressing tool that you want to measure and press the "In manual" softkey.

The tool is transferred to the "Measure: Dresser" window.

- OR -

Position the cursor to the "Zero offset" field and press the "Select ZO" softkey.

Select in the "Zero Offset - G54 ... G509" window the desired zero offset and press the "In manual" softkey.

6. Position the cursor in the "TR" field, press the "Select grinding wheel" softkey, select the grinding wheel to be used as reference point and press the "OK" softkey.

7. Press the "X" or "Z" softkey, depending on which tool length you want to measure.

8. Scratch the required edge using the tool.

9. Press the "Set length" softkey.

The tool length is calculated automatically and entered in the tool list. The cutting edge location is considered automatically.

4.6.2 Surface grinding

4.6.2.1 Overview

The geometries of the machining tool must be taken into consideration when executing a part program. These are stored as tool offset data in the tool list. Each time the tool is called, the control considers the tool offset data.

When programming the part program, you only need to enter the workpiece dimensions from the production drawing. The control then independently calculates the individual tool path.

Measuring grinding wheels and dressers

Tool offset data, i.e. the lengths or positions of the tools, are determined by taking manual measurements (scratching).

When measuring manually, traverse the tool manually to a defined reference point in order to determine the tool dimensions and positions in the Y and Z directions. The control then calculates the tool offset data from the position of the tool carrier reference point and the reference point.
Reference points for grinding wheels

For manual measurements of a grinding tool, you have the option of selecting the following reference points:

- Workpiece (with zero offset)
- Dresser (with zero offset as dressing tool)

Machine manufacturer

Please refer to the machine manufacturer's specifications.

Reference points for dressers

Use the grinding wheel as reference point for manually measuring a dresser.

4.6.2.2 Measure grinding tool manually with the work piece reference point

Reference point

The workpiece edge serves as reference point when measuring length X and length Z. You specify the position of the workpiece edge during the measurement.

Procedure

1. Select "JOG" mode in the "Machine" operating area.

2. Press the "Meas. tool" softkey.

3. Press the "Measure wheel" softkey.

4. Press the "Select tool" softkey.

   The "Tool Selection" window opens.

5. In the "Tool Selection" window, select the grinding tool that you want to measure and press the "OK" softkey. The cutting edge location must be entered in the tool list.

   - OR -

   Press the "Tool list" softkey, select in the tool list the grinding tool that you want to measure and press the "In manual" softkey.
The tool is transferred to the "Measure: Grinding Wheel" window.

6. Select the "Workpiece" entry in the "Reference point" selection field.

7. Press the "Y" or "Z" softkey, depending on which tool length you want to measure.

8. Enter the position of the workpiece edge in Y0 or Z0. If no value is entered for Y0 or Z0, the value is taken from the actual value display.

9. Scratch the required edge using the tool.

10. Press the "Set length" softkey. The tool length is calculated automatically and entered in the tool list. The cutting edge location is considered automatically.

---

Note

**Active grinding tool**

Tool measurement is possible only with an active grinding tool.

---

### 4.6.2.3 Measure grinding tool manually with the dresser reference point

**Reference point**

A dresser serves as reference point for the measurement of length Y or Z. The reference point of the dresser can be represented by a zero offset or a dressing tool. This setting is permanently stored in the machine data and specified by the machine manufacturer.

**Machine manufacturer**

Please refer to the machine manufacturer's specifications.

**Procedure**

1. Select "JOG" mode in the "Machine" operating area.

2. Press the "Meas. tool" softkey.

3. Press the "Measure wheel" softkey.
4. Press the "Select tool" softkey. The "Tool Selection" window opens.

5. In the "Tool Selection" window, select the grinding tool that you want to measure and press the "OK" softkey. The cutting edge location must be entered in the tool list.
   - OR -
   Press the "Tool list" softkey, select in the tool list the grinding tool that you want to measure and press the "In manual" softkey.

   The tool is transferred to the "Measure: "Grinding Wheel" window.

6. Select the "Dresser" entry in the "Reference point" selection field.

7. Position the cursor in the "TR" field, press the "Select dresser" softkey, select the dresser for measuring the tool length and press the "OK" softkey. The tool is transferred to the "Measure: Length Manual" window.
   - OR -
   Press the "Tool list" softkey, select in the tool list the dressing tool for measuring the tool length and press the "In manual" softkey.

   The tool is transferred to the "Measure: "Grinding Wheel" window.

8. Press the "Y" or "Z" softkey, depending on which tool length of the dresser tool to be measured.

9. Scratch the dresser using the tool.

10. Press the "Set length" softkey. The tool length is calculated automatically and entered in the tool list. The cutting edge location is considered automatically.

---

**Note**

**Active grinding tool**

Tool measurement is possible only with an active grinding tool.

---

### 4.6.2.4 Measuring the dressing tool manually with the grinding tool reference point

**Reference point**

A grinding wheel serves as reference point for the measurement of length X, Y or Z.
Procedure

1. Select "JOG" mode in the "Machine" operating area.

2. Press the "Meas. tool" softkey.

3. Press the "Measure dresser" softkey.

4. Press the "Select dresser" softkey.
   The "Tool Selection" window opens.

5. In the "Tool Selection" window, select the dressing tool to be measured and press the "OK" softkey.
   The cutting edge location must be entered in the tool list.
   - OR -

6. Press the "Tool list" softkey, select in the tool list the dressing tool to be measured and press the "In manual" softkey.
   The tool is transferred to the "Measure: Dresser" window.

7. Position the cursor in the "TR" field, press the "Select grinding wheel" softkey.

8. Select the grinding wheel for measuring the tool length and press the "OK" softkey.
   - OR -

9. Press the "Tool list" softkey, select in the tool list the grinding wheel for measuring the tool length and press the "In manual" softkey.
   The tool is transferred into the "Measure: Dresser" window.

10. Press the "X", "Y" or "Z" softkey, depending on which tool length of the dresser tool to be measured.

11. Scratch the dresser using the tool.

12. Press the "Set length" softkey.
   The tool length is calculated automatically and entered in the tool list.
   The cutting edge location is considered automatically.
Note

Active dressing tool

Tool measurement is possible only with an active dressing tool.
4.7 Measuring the workpiece zero

4.7.1 Cylindrical grinding

4.7.1.1 Measuring the workpiece zero

The reference point for programming a workpiece is always the workpiece zero. To determine this zero point, measure the length or the diameter of the workpiece and save it in a zero offset. This means that the position is stored in the coarse offset and existing values in the fine offset are deleted.

Calculation

When the workpiece zero / zero offset is calculated, the tool length is automatically taken into account.

Machine manufacturer

Please refer to the machine manufacturer's specifications.

Requirement

The requirement for measuring the workpiece is that a tool with known lengths is in the machining position.

Procedure

1. Select "JOG" mode in the "Machine" operating area.

2. Press the "Workpiece zero" softkey.

3. Select "Measuring only" if you only wish to display the measured values.

- OR -

Select in the "Zero offset" field the desired zero offset in which you want to store the zero point (e.g. basis reference).

- OR -
Press the "Select ZO" softkey and select the zero offset in which the zero point is to be saved in the "Zero offset – G54 ... G599" window and press the "In manual" softkey.

You return to the "Measure: Edge" window.

4. Traverse the tool in the X or Z direction and scratch the workpiece.
5. Enter the position setpoint of the workpiece edge X0 or Z0 and press the "Set ZO" softkey.

Note
Settable zero offsets
The labeling of the softkeys for the settable zero offsets varies, i.e. the settable zero offsets configured on the machine are displayed (examples: G54...G57, G54...G505, G54...G599).

Please refer to the machine manufacturer's specifications.

4.7.2 Surface grinding

4.7.2.1 Overview
The reference point for programming a workpiece is always the workpiece zero. The workpiece zero is determined on the workpiece edge.

Manual measurement
To measure the zero point manually, you need to traverse your tool manually up to the workpiece. Alternatively, you can also deploy a grinding tool with known length.

4.7.2.2 Setting the edge
The workpiece lies parallel to the coordinate system on the work table. You measure one reference point in one of the axes (X, Y, Z).

Requirement
A grinding tool is deployed for scratching when you measure the workpiece zero manually.
4.7 Measuring the workpiece zero

Procedure

1. Select the "Machine" operating area and press the <JOG> key.

2. Press the "Workpiece zero" and "Set edge" softkeys.
   The "Measure: Edge" window opens.

3. Select "Measuring only" if you only wish to display the measured values.
   - OR -

4. In the selection box, select the desired zero offset in which you want to store the zero point.
   - OR -
   Press the "Select ZO" softkey to select a settable zero offset.
   In the "Zero Offset – G54 ... G599" window, select a zero offset, in which the zero point should be saved.
   Press the "In manual" softkey.
   You return to the measurement window.

5. Use the softkeys to select in which axis direction you want to approach the workpiece first.

6. In X0, Y0, or Z0, specify the setpoint position of the workpiece edge.
   The setpoint position corresponds, e.g. to the dimension specifications of the workpiece edge from the workpiece drawing.

Note

Settable zero offsets

The labeling of the softkeys for the settable zero offsets varies, i.e. the settable zero offsets configured on the machine are displayed (examples: G54…G57, G54…G505, G54…G599).

Please refer to the machine manufacturer's specifications.
4.8 Monitoring axis and spindle data

4.8.1 Specify working area limitations

Using the "Working area limitation" function you can limit the range within which a tool should traverse in all channel axes. This function allows you to set up protection zones in the working area that are inhibited for tool motion.

In this way, you are able to restrict the traversing range of the axes in addition to the limit switches.

Requirements

You can only make changes in "AUTO" mode when in the RESET condition. These changes are then immediate.

You can make changes in "JOG" mode at any time. These changes, however, only become active at the start of a new motion.

Procedure

1. Select the "Parameter" operating area.

2. Press the "Setting data" softkey.

   The "Working Area Limitation" window appears.

3. Place the cursor in the required field and enter the new values via the numeric keyboard.

   The upper or lower limit of the protection zone changes according to your inputs.

4. Click the "active" checkbox to activate the protection zone.

Note

You will find all of the setting data in the "Start-up" operating area under "Machine data" via the menu forward key.

4.8.2 Editing spindle data

The speed limits set for the spindles that must not be under- or overshot are displayed in the "Spindles" window.
You can limit the spindle speeds in fields "Minimum" and "Maximum" within the limit values defined in the relevant machine data.

**Spindle speed limitation at constant cutting rate**

In field "Spindle speed limitation at G96", the programmed spindle speed limitation at constant cutting speed is displayed together with the permanently active limitations.

This speed limitation, for example, prevents the spindle from accelerating to the max. spindle speed of the current gear stage (G96) when performing tapping operations or machining very small diameters.

---

**Note**

The "Spindle data" softkey only appears if a spindle is configured.

---

**Procedure**

1. Select the "Parameter" operating area.
2. Press the "Setting data" and "Spindle data" softkeys.
   The "Spindles" window opens.
3. If you want to change the spindle speed, place the cursor on the "Maximum", "Minimum", or "Spindle speed limitation at G96" and enter a new value.

---

**4.8.3 Spindle chuck data**

**4.8.3.1 Defining spindle chuck data**

You store the chuck dimensions of the spindles at your machine in the "Spindle Chuck Data" window.

**Manually measuring a tool**

If you want to use the chuck of the main or counter-spindle as a reference point during manual measuring, specify the chuck dimension ZC.
Main spindle

Dimensioning, main spindle jaw type 1
1. Stop edge
2. Front edge

Dimensioning, main spindle jaw type 2

Counter-spindle

You can measure either the forward edge or stop edge of the counter-spindle. The forward edge or stop edge automatically serves as the valid reference point when traversing the counter-spindle. This is especially important when gripping the workpiece using the counter-spindle.

Dimensioning, counter-spindle jaw type 1
1. Stop edge
2. Front edge

Dimensioning, counter-spindle jaw type 2
### Tailstock

#### Dimensioning tailstock main spindle

#### Dimensioning tailstock counter-spindle

### Procedure

1. Select the "Parameter" operating area.

2. Press the "Setting data" and "Spindle chuck data" softkeys. The "Spindle Chuck Data" window opens.

3. Enter the desired parameter. The settings become active immediately.

### 4.8.3.2 Parameters, spindle chuck data

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Description</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Main spindle</strong></td>
<td>Dimensions of the forward edge or stop edge</td>
<td></td>
</tr>
<tr>
<td></td>
<td>• Jaw type 1</td>
<td></td>
</tr>
<tr>
<td></td>
<td>• Jaw type 2</td>
<td></td>
</tr>
<tr>
<td>ZC1</td>
<td>Main spindle chuck dimensions (inc)</td>
<td>mm</td>
</tr>
<tr>
<td>ZS1</td>
<td>Main spindle stop dimensions (inc)</td>
<td>mm</td>
</tr>
<tr>
<td>ZE1</td>
<td>Jaw dimension, main spindle (inc) - only for &quot;Jaw type 2&quot;</td>
<td>mm</td>
</tr>
<tr>
<td>XR</td>
<td>Tailstock diameter - only for tailstock that has been set-up</td>
<td>mm</td>
</tr>
<tr>
<td>ZR</td>
<td>Tailstock length - only for tailstock that has been set-up</td>
<td>mm</td>
</tr>
<tr>
<td><strong>Counter-spindle</strong></td>
<td>Dimensions of the forward edge or stop edge</td>
<td></td>
</tr>
<tr>
<td></td>
<td>• Jaw type 1</td>
<td></td>
</tr>
<tr>
<td></td>
<td>• Jaw type 2</td>
<td></td>
</tr>
<tr>
<td>ZC3</td>
<td>Chuck dimension, counter-spindle (inc) - only for a counter-spindle that has been set-up</td>
<td>mm</td>
</tr>
</tbody>
</table>
4.8.4 Entering cylinder error compensation (only rotary grinding machine)

The "Cylinder error compensation" function allows you to correct cylinder errors that have occurred when clamping the workpiece. The maximum compensation value is 1 mm (for 828D) or 10 mm (for 840D sl).

Machine manufacturer
Please observe the information provided by the machine manufacturer.

Software option
In order to use this function, you need the software option: "SINUMERIK Grinding Advanced".

Further information
You can find further information on cylinder error compensation in the Monitoring and Compensation Function Manual.

Precondition
- The option is set
- A compensation table for sag compensation (CEC) has been set up.

Machine manufacturer
Please observe the information provided by the machine manufacturer.
Procedure

1. Select the "Parameter" operating area.

2. Press the "Setting data" and "Cylinder error compensation" softkeys. The "Cylinder Error compensation" window opens.

3. Select the desired data set in the "Compensation" selection field.

4. For measuring points P1 and P2, enter the basis value (ZM) and compensation value (XM).
   The "Set comp." softkey can be used after entering the compensation values.

5. Press the "Set comp." softkey.
   The compensation is applied.
   - OR -

   Press the "Delete comp." softkey if you wish to delete the compensation values.
   The cylinder error compensation is deleted.
4.9 Displaying setting data lists

You can display lists with configured setting data.

**Machine manufacturer**

Please refer to the machine manufacturer's specifications.

**Procedure**

1. Select the "Parameter" operating area.

2. Press the "Setting data" and "Data lists" softkeys.
   The "Setting Data Lists" window opens.

3. Press the "Select data list" softkey and in the "View" list, select the required list with setting data.
4.10 Handwheel assignment

You can traverse the axes in the machine coordinate system (Machine) or in the workpiece coordinate system (Work) via the handwheel.

Software option

You require the “Extended operator functions” option for the handwheel offset (only for 828D).

All axes are provided in the following order for handwheel assignment:

- Geometry axes
  When traversing, the geometry axes take into account the actual machine status (e.g. rotations, transformations). All channel machine axes, which are currently assigned to the geometry axis, are in this case simultaneously traversed.

- Channel machine axes
  Channel machine axes are assigned to the particular channel. They can only be individually traversed, i.e. the actual machine state has no influence.
  The also applies to channel machine axes, that are declared as geometry axes.

Machine manufacturer

Please refer to the machine manufacturer's specifications.

Procedure

1. Select the "Machine" operating area.

2. Press the <JOG>, <AUTO> or <MDI> key.

3. Press the menu forward key and the "Handwheel" softkey.
   The "Handwheel" window appears.
   A field for axis assignment will be offered for every connected handwheel.

4. Position the cursor in the field next to the handwheel with which you wish to assign the axis (e.g. No. 1).

5. Press the corresponding softkey to select the desired axis (e.g. "X").
   - OR
Open the "Axis" selection box using the <INSERT> key, navigate to the desired axis, and press the <INPUT> key.
Selecting an axis also activates the handwheel (e.g., "X" is assigned to handwheel no. 1 and is activated immediately).

6. Press the "Handwheel" softkey again.

- OR -
Press the "Back" softkey.
The "Handwheel" window closes.

Deactivate handwheel

1. Position the cursor on the handwheel whose assignment you wish to cancel (e.g. No. 1).
2. Press the softkey for the assigned axis again (e.g. "X").

- OR -
Open the "Axis" selection box using the <INSERT> key, navigate to the empty field, and press the <INPUT> key.
Clearing an axis selection also clears the handwheel selection (e.g., "X" is cleared for handwheel no. 1 and is no longer active).
4.11 MDI

In "MDI" mode (Manual Data Input mode), you can enter G-code commands or standard cycles block-by-block and immediately execute them for setting up the machine.

You have the option of loading an MDI program or a standard program with the standard cycles directly into the MDI buffer from the program manager; you can subsequently then edit it.

**Machine manufacturer**
Please observe the information provided by the machine manufacturer.

You can save programs, generated or modified in the MDI working window, in the program manager, e.g. in a directory specifically created for the purpose.

**Software option**
You require the "Extended operator functions" option to load and save MDI programs (for 828D).

4.11.1 Loading an MDA program from the Program Manager

**Procedure**

1. Select the "Machine" operating area.
2. Press the <MDI> key.
   The MDI editor opens.
3. Press the "Load MDI" softkey.
   The "Load in MDI" window opens. It shows you a view of the program manager.
4. Position the cursor to the corresponding storage location, press the "Search" softkey and enter the required search term in the search dialog if you wish to search for a specific file.
   **Note:** The place holders "***" (replaces any character string) and "?" (replaces any character) make it easier for you to perform a search.
5. Select the program that you would like to edit or execute in the MDI window.
6. Press the "OK" softkey.
   The window closes and the program is ready for operation.
4.11.2 Saving an MDA program

Procedure

1. Select the "Machine" operating area.

2. Press the <MDI> key.

The MDI editor opens.

3. Create the MDI program by entering the G-code commands using the operator's keyboard.

4. Press the "Store MDI" softkey.

The "Save from MDI: Select storage location" window opens. It shows you a view of the program manager.

5. Select the drive to which you want to save the MDI program you created, and place the cursor on the directory in which the program is to be stored.

   - OR -

Position the cursor to the required storage location, press the "Search" softkey and enter the required search term in the search dialog if you wish to search for a specific directory or subdirectory.

   **Note:** The place holders "*" (replaces any character string) and "?" (replaces any character) make it easier for you to perform a search.

6. Press the "OK" softkey.

When you place the cursor on a folder, a window opens which prompts you to assign a name.

   - OR -

When you place the cursor on a program, you are asked whether the file should be overwritten.

7. Enter the name for the rendered program and press the "OK" softkey.

The program will be saved under the specified name in the selected directory.
4.11.3 Editing/executing a MDI program

Procedure

1. Select the "Machine" operating area.

2. Press the <MDI> key.
   The MDI editor opens.

3. Enter the desired G-code commands using the operator’s keyboard.
   - OR -
   Enter a standard cycle, e.g. CYCLE62 ().

4. Editing G-code commands/program blocks
   - OR -
   Select the required program block (e.g. CYCLE62) and press the <cursor right> key, enter the required value and press "OK".

   When editing a cycle, either the help screen or the graphic view can be displayed.

5. Press the <CYCLE START> key.

   The control executes the input blocks.

When executing G-code commands and standard cycles, you have the option of controlling the sequence as follows:

- Executing the program block-by-block
- Testing the program
  Settings under program control
- Setting the test-run feedrate
  Settings under program control

4.11.4 Deleting an MDA program

Precondition

The MDA editor contains a program that you created in the MDI window or loaded from the program manager.
Procedure

Press the "Delete blocks" softkey.

The program blocks displayed in the program window are deleted.
Setting up the machine

4.11 MDI
Working in the manual mode

5.1 Overview

Always use "JOG" mode when you want to set up the machine for the execution of a program or to carry out simple traversing movements on the machine:

- Synchronize the measuring system of the controller with the machine (reference point approach)
- Set up the machine, i.e. activate manually-controlled motions on the machine using the keys and handwheels provided on the machine control panel.
- You can activate manually controlled motions on the machine using the keys and handwheels provided on the machine control panel while a part program is interrupted.
5.2 Selecting a tool and spindle

5.2.1 T,S,M window

For the preparatory actions in manual mode, tool selection and spindle control are both performed centrally in a screen form.

In addition to the main spindle (S1), there is another tool spindle (S2) for powered tools.

Your turning machine can also be equipped with a counter-spindle (S3).

In manual mode, you can select a tool on the basis of either its name or its revolver location number. If you enter a number, a search is performed for a name first, followed by a location number. This means that if you enter "5", for example, and no tool with the name "5" exists, the tool is selected from location number "5".

Note

Using the revolver location number, you can also swing around an empty space into the machining position and then comfortably install a new tool.

Machine manufacturer

Please refer to the machine manufacturer's specifications.

<table>
<thead>
<tr>
<th>Display</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td>T</td>
<td>Input of the tool (name or location number) You can select a tool from the tool list using the &quot;Select tool&quot; softkey.</td>
</tr>
<tr>
<td>D</td>
<td>Cutting edge number of the tool (1-9)</td>
</tr>
<tr>
<td>DL</td>
<td>Number for additive and setup offset</td>
</tr>
<tr>
<td>ST</td>
<td>Sister tool (for replacement tool strategy)</td>
</tr>
<tr>
<td>Spindles 1 and 2 (e.g. S1)</td>
<td>Spindle selection for master spindle and identification with spindle number</td>
</tr>
<tr>
<td>Spindle M function</td>
<td></td>
</tr>
<tr>
<td>Spindle off: Spindle is stopped</td>
<td></td>
</tr>
<tr>
<td>CCW rotation: Spindle rotates counterclockwise</td>
<td></td>
</tr>
<tr>
<td>CW rotation: Spindle rotates clockwise</td>
<td></td>
</tr>
<tr>
<td>Spindle positioning: Spindle is moved to the desired position.</td>
<td></td>
</tr>
<tr>
<td>Other M functions</td>
<td>Input of machine functions Refer to the machine manufacturer's table for the correlation between the meaning and number of the function.</td>
</tr>
</tbody>
</table>
### Display | Meaning
--- | ---
G zero offset | Selection of the zero offset (basic reference, G54 - 57)
You can select zero offsets from the tool list of settable zero offsets via the "Zero offset" softkey.
Measurement unit | Selection of the unit of measurement (inch, mm).
The setting made here has an effect on the programming.
Machining plane | Selection of the machining plane (G17(XY), G18 (ZX), G19 (YZ))
Gear stage | Specification of the gear stage (auto, I - V)
Stop position | Input of the spindle position in degrees

#### Note

**Spindle positioning**
You can use this function to position the spindle at a specific angle, e.g. during a tool change.
- A stationary spindle is positioned via the shortest possible route.
- A rotating spindle is positioned as it continues to turn in the same direction.

### 5.2.2 Selecting a tool

#### Procedure

1. Select the "JOG" operating mode.

2. Press the "T, S, M" softkey.

3. Enter the name or the number of the tool T.
   - OR -
   Press the "Select tool" softkey to open the tool list, position the cursor on the desired tool and press the "In Manual" softkey.
   The tool is transferred to the "T, S, M... window" and displayed in the field of tool parameter "T".

4. Select tool edge D or enter the number directly in field "D".

5. Press the "CYCLE START" key.
The tool is loaded into the spindle.
5.2.3 Starting and stopping the spindle manually

Procedure

1. Select the "T,S,M" softkey in the "JOG" mode.

2. Select the desired spindle (e.g. S1) and enter the desired spindle speed or cutting speed in the right-hand entry field.

3. If the machine has a gearbox for the spindle, set the gearing step.

4. Select a spindle direction of rotation (clockwise or counterclockwise) in the "Spindle M function" field.

5. Press the <CYCLE START> key.
   The spindle rotates.

6. Select the "Stop" setting in the "Spindle M function" field.
   Press the <CYCLE START> key.
   The spindle stops.

Note

Changing the spindle speed

If you enter the speed in the "Spindle" field while the spindle is rotating, the new speed is applied.
5.2.4 Positioning the spindle

Procedure

1. Select the "T,S,M" softkey in the "JOG" mode.

- OR -

2. Select the "Stop Pos." setting in the "Spindle M function" field. The "Stop Pos." entry field appears.

3. Enter the desired spindle stop position. The spindle position is specified in degrees.

4. Press the <CYCLE START> key.

The spindle is moved to the desired position.

Note
You can use this function to position the spindle at a specific angle, e.g. during a tool change.

- A stationary spindle is positioned via the shortest possible route.
- A rotating spindle is positioned as it continues to turn in the same direction.
5.3 Traversing axes

5.3.1 Traversing the axes

You can traverse the axes in manual mode via the Increment or Axis keys or handwheels.

During a traverse initiated from the keyboard, the selected axis moves at the programmed setup feedrate. During an incremental traverse, the selected axis traverses a specified increment.

Set the default feedrate

Specify the feedrate to be used for axis traversal in the set-up, in the "Settings for Manual Operation" window.

5.3.2 Traverse axes by a defined increment

You can traverse the axes in manual mode via the Increment and Axis keys or handwheels.

Procedure

1. Select the "Machine" operating area.
2. Press the <JOG> key.
3. Press keys 1, 10, etc. up to 10000 in order to move the axis in a defined increment.
   The numbers on the keys indicate the traverse path in micrometers or microinches.
   Example: Press the "100" button for a desired increment of 100 μm (= 0.1 mm).
4. Select the axis to be traversed.
5. Press the <+> or <-> key.
   Each time you press the key the selected axis is traversed by the defined increment.
   Feedrate and rapid traverse override switches can be operative.
Note
When the controller is switched on, the axes can be traversed right up to the limits of the machine as the reference points have not yet been approached and the axes referenced. Emergency limit switches might be triggered as a result.

The software limit switches and the working area limitation are not yet operative!
The feed enable signal must be set.

Machine manufacturer
Please refer to the machine manufacturer's specifications.

5.3.3 Traversing axes by a variable increment

Procedure

1. Select the "Machine" operating area.

2. Press the <JOG> key.

3. Press the "Settings" softkey.
The "Settings for Manual Operation" window is opened.

4. Enter the desired value for the "Variable increment" parameter.
Example: Enter 500 for a desired increment of 500 μm (0.5 mm).

5. Press the <Inc VAR> key.

6. Select the axis to be traversed.

7. Press the <-> or <-> key.
Each time you press the key the selected axis is traversed by the set increment.

Feedrate and rapid traverse override switches can be operative.
5.4 Positioning axes

In order to implement simple machining sequences, you can traverse the axes to certain positions in manual mode.

The feedrate / rapid traverse override is active during traversing.

Procedure

1. If required, select a tool.

2. Select the "JOG" operating mode.

3. Press the "Positions" softkey.

4. Enter the target position or target angle for the axis or axes to be traversed.

5. Specify the desired value for the feedrate F.
   - OR -
   Press the "Rapid traverse" softkey.
   The rapid traverse is displayed in field "F".

6. Press the <CYCLE START> key.
   The axis is traversed to the specified target position.

If target positions were specified for several axes, the axes are traversed simultaneously.
## 5.5 Default settings for manual mode

Specify the configurations for the manual mode in the "Settings for manual operation" window.

### Default settings

<table>
<thead>
<tr>
<th>Settings</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td>Type of feedrate</td>
<td>Here, you select the type of feedrate.</td>
</tr>
<tr>
<td></td>
<td>• G94: Axis feedrate/linear feedrate</td>
</tr>
<tr>
<td></td>
<td>• G95: Revolutions feedrate</td>
</tr>
<tr>
<td>Setup feedrate G94</td>
<td>Enter the desired feedrate in mm/min.</td>
</tr>
<tr>
<td>Setup feedrate G95</td>
<td>Enter the desired feedrate in mm/rev.</td>
</tr>
<tr>
<td>Variable increment</td>
<td>For variable increments, enter the desired increment when traversing axes.</td>
</tr>
<tr>
<td>Spindle speed</td>
<td>Here, enter the desired spindle speed in rpm.</td>
</tr>
</tbody>
</table>

### Procedure

1. Select the "Machine" operating area.

2. Press the <JOG> key.

3. Press the menu forward key and the "Settings" softkey. The "Settings for manual operation" window is opened.

### See also

Switching the unit of measurement (Page 93)
5.5 Default settings for manual mode
6.1 Starting and stopping machining

During execution of a program, the workpiece is machined in accordance with the programming on the machine. After the program is started in automatic mode, workpiece machining is performed automatically.

Preconditions

The following requirements must be met before executing a program:

- The measuring system of the controller is referenced with the machine.
- The necessary tool offsets and work offsets have been entered.
- The necessary safety interlocks implemented by the machine manufacturer are activated.

General sequence

1. Use the Program manager to select the desired program.
2. Select under "NC", "Local. Drive", "USB" or set-up network drives the desired program.
3. Press the "Select" softkey. The program is selected for execution and automatically switched to the "Machine" operating area.
4. Press the <CYCLE START> key. The program is started and executed.

Note

Starting the program in any operating area

If the control system is in the "AUTO" mode, you can also start the selected program when you are in any operating area.
6.1 Starting and stopping machining

**Stopping machining**

Press the <CYCLE STOP> key.
Machining stops immediately, individual blocks do not finish execution.
At the next start, execution is resumed at the same location where it stopped.

**Canceling machining**

Press the <RESET> key.
Execution of the program is interrupted. On the next start, machining will start from the beginning.

**Machine manufacturer**

Please observe the information provided by the machine manufacturer.
6.2 Selecting a program

Procedure

1. Select the "Program Manager" operating area.
   The directory overview is opened.

2. Select the location where the program is archived (e.g. "NC")

3. Place the cursor on the directory containing the program that you want to select.

4. Press the <INPUT> key.
   - OR -
   Press the <Right cursor> key.
   The directory contents are displayed.

5. Place the cursor on the desired program.

6. Press the "Select" softkey.
   When the program has been successfully selected, an automatic changeover to the "Machine" operating area occurs.
6.3 Executing a trial program run

When testing a program, you can select that the system can interrupt the machining of the workpiece after each program block, which triggers a movement or auxiliary function on the machine. In this way, you can control the machining result block-by-block during the initial execution of a program on the machine.

Note

Settings for the automatic mode

Rapid traverse reduction and dry run feed rate are available to run-in or to test a program.

Move by single block

In "Program control" you may select from among several types of block processing:

<table>
<thead>
<tr>
<th>SB mode</th>
<th>Scope</th>
</tr>
</thead>
<tbody>
<tr>
<td>SB1 Single block, coarse</td>
<td>Machining stops after every machine block (except for cycles).</td>
</tr>
<tr>
<td>SB2 Data block</td>
<td>Machining stops after every block, i.e. also for data blocks (except for cycles)</td>
</tr>
<tr>
<td>SB3 Single block, fine</td>
<td>Machining stops after every machine block (also in cycles)</td>
</tr>
</tbody>
</table>

Precondition

A program must be selected for execution in "AUTO" or "MDA" mode.

Procedure

1. Press the "Prog. ctrl." softkey and select the desired variant in the "SBL" field.
2. Press the <SINGLE BLOCK> key.
3. Press the <CYCLE START> key.
   Depending on the execution variant, the first block will be executed. Then the machining stops.
   In the channel status line, the text “Stop: Block in single block ended” appears.
4. Press the <CYCLE START> key.
   Depending on the mode, the program will continue executing until the next stop.
5. Press the <SINGLE BLOCK> key again, if the machining is not supposed to run block-by-block.
   The key is deselected again.
   If you now press the <CYCLE START> key again, the program is executed to the end without interruption.
6.4 Displaying the current program block

6.4.1 Current block display

The window of the current block display shows the program blocks currently being executed.

Display of current program

The following information is displayed in the running program:

- The workpiece name or program name is entered in the header line.
- The program block which is just being processed appears colored.

Display of the machining times

If you set that the machining times are to be recorded in the settings for automatic mode, the measured times are shown at the end of the line as follows:

<table>
<thead>
<tr>
<th>Display</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td>Light green background</td>
<td>Measured machining time of the program block (automatic mode)</td>
</tr>
<tr>
<td>17.18</td>
<td></td>
</tr>
<tr>
<td>Green background</td>
<td>Measured machining time of the program group (automatic mode)</td>
</tr>
<tr>
<td>19.47</td>
<td></td>
</tr>
<tr>
<td>Light blue background</td>
<td>Estimated machining time of the program block (simulation)</td>
</tr>
<tr>
<td>17.31</td>
<td></td>
</tr>
<tr>
<td>Blue background</td>
<td>Estimated machining time of the program group (simulation)</td>
</tr>
<tr>
<td>19.57</td>
<td></td>
</tr>
<tr>
<td>Yellow background</td>
<td>Wait time (automatic mode or simulation)</td>
</tr>
<tr>
<td>4.53</td>
<td></td>
</tr>
</tbody>
</table>

Highlighting of selected G code commands or keywords

In the program editor settings, you can specify whether selected G code commands are to be highlighted in color. The following colors are used as standard:

<table>
<thead>
<tr>
<th>Display</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td>Blue font</td>
<td>D, S, F, T, M and H functions</td>
</tr>
<tr>
<td>Red font</td>
<td>&quot;G0&quot; motion command</td>
</tr>
<tr>
<td>Green font</td>
<td>&quot;G1&quot; motion command</td>
</tr>
</tbody>
</table>
### Machine manufacturer

You can define further highlight colors in the "sleditorwidget.ini" configuration file. Please refer to the machine manufacturer's instructions.

### Editing a program directly

In the Reset state, you can edit the current program directly.

1. Press the <INSERT> key.
2. Place the cursor at the relevant position and edit the program block. Direct editing is only possible for G code blocks in the NC memory, not for external execution.
3. Press the <INSERT> key to exit the program and the edit mode again.

### See also

Settings for the automatic mode (Page 207)

### 6.4.2 Displaying a basic block

If you want precise information about axis positions and important G functions during testing or program execution, you can call up the basic block display. This is how you check, when using cycles, for example, whether the machine is actually traversing.

Positions programmed by means of variables or R parameters are resolved in the basic block display and replaced by the variable value.

You can use the basic block display both in test mode and when machining the workpiece on the machine. All G code commands that initiate a function on the machine are displayed in the "Basic Blocks" window for the currently active program block:

- Absolute axis positions
- G functions for the first G group
- Other modal G functions

<table>
<thead>
<tr>
<th>Display</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td>Blue-green font</td>
<td>&quot;G2&quot; or &quot;G3&quot; motion command</td>
</tr>
<tr>
<td>Gray font</td>
<td>Comment</td>
</tr>
</tbody>
</table>
6.4 Displaying the current program block

- Other programmed addresses
- M functions

Machine manufacturer
Please observe the information provided by the machine manufacturer.

Procedure

1. A program is selected for execution and has been opened in the "Machine" operating area.
2. Press the "Basic blocks" softkey.
   The "Basic Blocks" window opens.
3. Press the <SINGLE BLOCK> key if you wish to execute the program block by block.
4. Press the <CYCLE START> key to start the program execution.
   The axis positions to be approached, modal G functions, etc., are displayed in the "Basic Blocks" window for the currently active program block.
5. Press the "Basic blocks" softkey once again to hide the window again.

6.4.3 Display program level

You can display the current program level during the execution of a large program with several subprograms.

Several program run throughs
If you have programmed several program run throughs, i.e. subprograms are run through several times one after the other by specifying the additional parameter P, then during processing, the program runs still to be executed are displayed in the "Program Levels" window.

Program example
N10 subprogram P25
If, in at least one program level, a program is run through several times, a horizontal scroll bar is displayed that allows the run through counter P to be viewed in the righthand window section.
The scroll bar disappears if multiple run-through is no longer applicable.

Display of program level
The following information will be displayed:
- Level number
- Program name
- Block number, or line number
- Remain program run throughs (only for several program run throughs)

**Precondition**

A program must be selected for execution in "AUTO" mode.

**Procedure**

![Program levels](image)

Press the "Program levels" softkey.
The "Program levels" window appears.
6.5 Correcting a program

As soon as a syntax error in the part program is detected by the controller, program execution is interrupted and the syntax error is displayed in the alarm line.

Correction options

Depending on the state of the control system, you have various options of correcting the program.

- **Stop state**
  Only change lines that have not been executed

- **Reset status**
  Change all lines

---

**Note**

The "program correction" function is also available for execute from external; however, when making program changes, the NC channel must be brought into the reset state.

---

Precondition

A program must be selected for execution in "AUTO" mode.

Procedure

1. The program to be corrected is in the Stop or Reset mode.
2. Press the "Prog. corr." softkey.
   The program is opened in the editor.
   The program preprocessing and the current block are displayed. The current block is also updated in the running program, but not the displayed program section, i.e. the current block moves out of the displayed program section.
   If a subprogram is executed, it is not opened automatically.
3. Make the necessary corrections.
4. Press the "NC Execute" softkey.
   The system switches back to the "Machine" operating area and selects "AUTO" mode.
5. Press the "CYCLE START" key to resume program execution.

---

**Note**

When you exit the editor using the "Close" softkey, you return to the "Program manager" operating area.
6.6 Repositioning axes

After a program interruption in the automatic mode (e.g. after a tool breaks), you can move the tool away from the contour in manual mode.

The coordinates of the interrupt position will be saved. The distances traversed in manual mode are displayed in the actual value window. This path difference is called "REPOS offset".

Resuming program execution
You use the "REPOS" function to return the tool to the contour of the workpiece to continue executing the program.

The interrupt position is not passed as it is blocked by the control system.

The feedrate/rapid traverse override is in effect.

NOTICE
Risk of collision
When repositioning, the axes move with the programmed feedrate and linear interpolation, i.e. in a straight line from the current position to the interrupt point. Therefore, you must first move the axes to a safe position in order to avoid collisions.

If you do not use the "REPOS" function after a program interrupt and then traversing the axes in manual mode, then on changing to automatic mode and starting the machining process, the control automatically traverses the axes in straight lines back to where they were at point of interruption.

Requirement
The following prerequisites must be met when repositioning the axes:

- The program execution was interrupted using <CYCLE STOP>.
- The axes were moved from the interrupt point to another position in manual mode.

Procedure

1. Press the <REPOS> key.
2. Select the axes to be traversed one after the other.
3. Press the <+> or <-> key for the relevant direction.
   The axes are moved to the interrupt position.
6.7 Starting execution at a specific point

6.7.1 Use block search

If you only want to perform a certain section of a program on the machine, then you need not start the program from the beginning. You can start the program from a specified program block.

Applications

- Stopping or interrupting program execution
- Specifying a target position, e.g. during remachining

Determining a search target

- User-friendly search target definition (search positions)
  - Direct specification of the search target by positioning the cursor in the selected program (main program)
    
    **Note:**
    During the block search, you must ensure that the correct tool is in the working position before beginning program execution.
  
    - Search target via text search
  
    - The search target is the interruption point (main program and subprogram)
      
      The function is only available if there is an interruption point. After a program interruption (CYCLE STOP, RESET or power off), the controller saves the coordinates of the interruption point.
  
    - The search target is the higher program level of the interruption point (main program and subprogram)
      
      The level can only be changed if an interruption point in a subprogram is selected. You can then change the program level up to the main program level and back to the level of the interruption point.

- Search pointer
  
  - Direct entry of the program path

**Note**

You can search for a specific point in subprograms with the search pointer if there is no interruption point.

**Software option**

You require the "Extended operator functions" option for the "Search pointer" function (only for 828D).
Cascaded search

You can start another search from the "Search target found" state. It is possible to continue the cascading any number of times after every search target found.

Note

Another cascaded block search can be started from the stopped program execution only if the search target has been found.

Preconditions

- You have selected the desired program.
- The controller is in the reset state.
- The desired search mode is selected.

NOTICE

Risk of collision

Pay attention to a collision-free start position and appropriate active tools and other technological values.

If necessary, manually approach a collision-free start position. Select the target block considering the selected block search version.

Toggling between search pointer and search positions

Press the "Search pointer" softkey again to exit the "Search pointer" window and return to the "Program" window to define search positions.

- OR -

Press the "Back" softkey.

You have now exited the block search function.

Further information

You can find further information on the block search function in the Basic Functions Function Manual.
6.7 Starting execution at a specific point

6.7.2 Continuing program from search target

Press the "CYCLE START" key twice to continue the program from the desired position.

- The first CYCLE START outputs the auxiliary functions collected during the search. The program is then in the Stop state.

- Before the second CYCLE START, you can use the "Overstore" function to create states that are required, but not yet available, for the further program execution.

If the set position is not to be approached automatically after the program start, you can also traverse the tool manually from the current position to the set position by changing to JOG mode for the REPOS function.

6.7.3 Simple search target definition

Requirement

The program is selected and the controller is in Reset mode.

Procedure

1. Press the "Block search" softkey.
2. Place the cursor on a particular program block.
   - OR -
   Press the "Find text" softkey, select the search direction, enter the search text and confirm with "OK”.
3. Press the "Start search" softkey.

   The search starts. Your specified search mode will be taken into account.
   The current block will be displayed in the "Program" window as soon as the target is found.
4. If the located target (for example, when searching via text) does not correspond to the program block, press the "Start search" softkey again until you find your target.
   Press the <CYCLE START> key twice.
   Processing is continued from the defined position.
6.7.4 Defining an interruption point as search target

Requirement

A program was selected in "AUTO" mode and interrupted during execution through CYCLE STOP or RESET.

Software option

You require the "Extended operator functions" option (only for 828D).

Procedure

1. Press the "Block search" softkey.
2. Press the "Interrupt point" softkey.
   The interruption point is loaded.
3. If the "Higher level" and "Lower level" softkeys are available, use these to change the program level.
4. Press the "Start search" softkey.
   The search starts. Your specified search mode will be taken into account.
   The search screen closes.
   The current block will be displayed in the "Program" window as soon as the target is found.
5. Press the <CYCLE START> key twice.
   The execution will continue from the interruption point.

6.7.5 Parameters for block search in the search pointer

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td>Number of program level</td>
<td></td>
</tr>
<tr>
<td>Program:</td>
<td>The name of the main program is automatically entered</td>
</tr>
<tr>
<td>Ext:</td>
<td>File extension</td>
</tr>
<tr>
<td>P:</td>
<td>Number of subprogram repetitions</td>
</tr>
<tr>
<td>Line:</td>
<td>Is automatically filled for an interruption point</td>
</tr>
</tbody>
</table>
### 6.7.6 Block search mode

Set the desired search variant in the “Search Mode” window.

The set mode is retained when the controller is shut down. When you activate the “Search” function after restarting the controller, the current search mode is displayed in the title row.

#### Search variants

<table>
<thead>
<tr>
<th>Block search mode</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td>With calculation - without approach</td>
<td>It is used in order to be able to approach a target position in any circumstance (e.g. tool change position). The end position of the target block or the next programmed position is approached using the type of interpolation valid in the target block. Only the axes programmed in the target block are moved. <strong>Note:</strong> If machine data 11450.1=1 is set, the rotary axes of the active swivel data record are pre-positioned after the block search.</td>
</tr>
<tr>
<td>With calculation - with approach</td>
<td>It is used to be able to approach the contour in any circumstance. The end position of the block prior to the target block is found with &lt;CYCLE START&gt;. The program runs in the same way as in normal program processing.</td>
</tr>
<tr>
<td>With calculation - skip extcall</td>
<td>This is used to speed-up a search with calculation when using EXTCALL programs: EXTCALL programs are not taken into account. <strong>Notice:</strong> Important information, e.g. modal functions, which are located in the EXTCALL program, are not taken into account. In this case, after the search target has been found, the program is not able to be executed. Information such as this should be programmed in the main program.</td>
</tr>
</tbody>
</table>
### Block search mode

<table>
<thead>
<tr>
<th>Block search mode</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td>Without calculation</td>
<td>For a quick search in the main program. Calculations will not be performed during the block search, i.e. the calculation is skipped up to the target block. All settings required for execution have to be programmed from the target block (e.g. feedrate, spindle speed, etc.).</td>
</tr>
<tr>
<td>With program test</td>
<td>Multi-channel block search with calculation (SERUPRO). All blocks are calculated during the block search. Absolutely no axis motion is executed, however, all auxiliary functions are output. The NC starts the selected program in the program test mode. If the NC reaches the specified target block in the actual channel, it stops at the beginning of the target block and deselects program test mode again. After continuing the program with NC start (after REPOS motion) the auxiliary functions of the target block are output. For single-channel systems, the coordination is supported with events running in parallel, e.g. synchronized actions.</td>
</tr>
</tbody>
</table>

**Note**

The search speed depends on MD settings.

---

**Machine manufacturer**

Please refer to the machine manufacturer's specifications.

---

**References**

For additional information, please refer to the following documentation:

SINUMERIK Operate Commissioning Manual
6.8 Controlling the program run

6.8.1 Program control

You can change the program sequence in the "AUTO" and "MDA" modes.

<table>
<thead>
<tr>
<th>Abbreviation/program control</th>
<th>Mode of operation</th>
</tr>
</thead>
<tbody>
<tr>
<td>PRT</td>
<td>The program is started and executed with auxiliary function outputs and dwell times. In this mode, the axes are not traversed. The programmed axis positions and the auxiliary function outputs are controlled this way. <strong>Note:</strong> You can activate program execution without any axis motion using the &quot;Dry run feedrate&quot;.</td>
</tr>
<tr>
<td>DRY</td>
<td>The traversing velocities programmed in conjunction with G1, G2, G3, CIP and CT are replaced by a defined dry run feedrate. The dry run feedrate also applies instead of the programmed revolution feedrate. <strong>Notice:</strong> Do not machine any workpieces when &quot;Dry run feedrate&quot; is active because the altered feedrates might cause the permissible tool cutting rates to be exceeded and the workpiece or machine tool could be damaged.</td>
</tr>
<tr>
<td>RG0</td>
<td>In the rapid traverse mode, the traversing speed of the axes is reduced to the percentage value entered in RG0. <strong>Note:</strong> You define the reduced rapid traverse in the settings for automatic operation.</td>
</tr>
<tr>
<td>M01</td>
<td>The processing of the program stops at every block in which supplementary function M01 is programmed. In this way you can check the already obtained result during the processing of a workpiece. <strong>Note:</strong> In order to continue executing the program, press the &lt;CYCLE START&gt; key again.</td>
</tr>
<tr>
<td>Programmed stop 2 (e.g. M101)</td>
<td>The processing of the program stops at every block in which the &quot;Cycle end&quot; is programmed (e.g. with M101). <strong>Note:</strong> In order to continue executing the program, press the &lt;CYCLE START&gt; key again. <strong>Note:</strong> The display can be changed. Please observe the information provided by the machine manufacturer.</td>
</tr>
<tr>
<td>DRF</td>
<td>Enables an additional incremental work offset while processing in automatic mode with an electronic handwheel. This function can be used to compensate for tool wear within a programmed block. <strong>Note:</strong> You require the &quot;Extended operator functions&quot; option to use the handwheel offset (for 828D).</td>
</tr>
</tbody>
</table>
| SB                           | Individual blocks are configured as follows.  
  - Single block, coarse: The program stops only after blocks which perform a machine function.  
  - Data block: The program stops after each block.  
  - Single block, fine: The program also stops only after blocks which perform a machine function in cycles.  
  Select the desired setting using the <SELECT> key. |
### Abbreviation/program control

<table>
<thead>
<tr>
<th>Abbreviation</th>
<th>Mode of operation</th>
</tr>
</thead>
<tbody>
<tr>
<td>SKP</td>
<td>Skip blocks are skipped during machining.</td>
</tr>
<tr>
<td>GCC</td>
<td>When executing a jobshop program, it is converted into a G-code program.</td>
</tr>
<tr>
<td>MRD</td>
<td>In the program, the measurement results screen display is activated while machining.</td>
</tr>
<tr>
<td>CST</td>
<td>Program processing stops at the points you defined as relevant to stop before the program started. These may be, for example, especially critical points, at which you can check the correctness of the sequence or exclude collisions. You can specify NC functions (auxiliary functions, subprograms) and NC function transitions as being stop-relevant. As standard, the following conditions can be selected as relevant to stop in the &quot;Program control&quot; window.</td>
</tr>
<tr>
<td>CST</td>
<td>Transition G0-G1</td>
</tr>
<tr>
<td>CST</td>
<td>Transition G1-G0</td>
</tr>
<tr>
<td>CST</td>
<td>Transition G0-G0</td>
</tr>
</tbody>
</table>

**Note:** Please observe the information provided by the machine manufacturer.

### Activating program control

You can control the program sequence however you wish by selecting and clearing the relevant checkboxes.

**Display / response of active program controls**

If program control is activated, the abbreviation of the corresponding function appears in the status display as feedback response.

### Procedure

1. Select the "Machine" operating area.
2. Press the <AUTO> or <MDI> key.
3. Press the "Prog. ctrl." softkey. The "Program Control" window opens.

### 6.8.2 Skip blocks

You can skip program blocks that are not to be executed every time the program runs.
The skip blocks are identified by placing a "/" (forward slash) or "/x (x = number of skip level) character in front of the block number. You have the option of hiding several block sequences.

The statements in the skipped blocks are not executed. The program continues with the next block, which is not skipped.

The number of skip levels that can be used depends on a machine datum.

**Machine manufacturer**

Please observe the information provided by the machine manufacturer.

**Software option**

In order to have more than two skip levels, for 828D you require the "Extended operator functions" option.

**Skip levels, activate**

Select the corresponding checkbox to activate the desired skip level.

**Note**

The "Program Control - Skip Blocks" window is only available when more than one skip level is set up.
6.9 Overstore

With overstore, you have the option of executing technological parameters (for example, auxiliary functions, axis feed, spindle speed, programmable instructions, etc.) before the program is actually started. The program instructions act as if they are located in a normal part program. These program instructions are, however, only valid for one program run. The part program is not permanently changed. When next started, the program will be executed as originally programmed.

After a block search, the machine can be brought into another state with overstore (e.g. M function, tool, feed, speed, axis positions etc.), in which the normal part program can be successfully continued.

Software option
You require the "Extended operator functions" option for the overstore function (for 828D).

Requirement
The program to be corrected is in the Stop or Reset mode.

Procedure

1. Select the "Machine" operating area in the "AUTO" mode.

2. Press the "Overstore" softkey.
   The "Overstore" window opens.

3. Enter the required data and NC block.

4. Press the <CYCLE START> key.
   The blocks you have entered are stored. You can observe execution in the "Overstore" window.
   After the entered blocks have been executed, you can append blocks again.
   You cannot change the operating mode while you are in overstore mode.

5. Press the "Back" softkey.
   The "Overstore" window closes.

6. Press the <CYCLE START> key again.
   The program selected before overstoring continues to run.
Note
Block-by-block execution
The <SINGLE BLOCK> key is also active in the overstore mode. If several blocks are entered in the overstore buffer, then these are executed block-by-block after each NC start.

Deleting blocks
Press the "Delete blocks" softkey to delete program blocks you have entered.
6.10 Editing a program

6.10.1 Processing a program (editor)

With the editor, you are able to render, supplement, or change part programs.

Note

Maximum block length

The maximum block length is 512 characters.

Calling the editor

- The editor is started via the "Program correction" softkey in the "Machine" operating area. You can directly change the program by pressing the <INSERT> key.
- The editor is called via the "Open" softkey as well as with the <INPUT> or <Cursor right> key in the "Program manager" operating area.
- The editor opens in the "Program" operating area with the last executed part program, if this was not explicitly exited via the "Close" softkey.

Note

- Please note that the changes to programs saved in the NC memory take immediate effect.
- If you are editing on a local drive or external drives, you can also exit the editor without saving, depending on the setting. Programs in the NC memory are always automatically saved.
- Exit the program correction mode using the "Close" softkey to return to the "Program manager" operating area.

6.10.2 Searching in programs

You can use the search function to quickly arrive at points where you would like to make changes, e.g. in very large programs.

Various search options are available that enable selective searching.

Search options

- Whole words
  Activate this option and enter a search term if you want to search for texts/terms that are present as words in precisely this form.
  If, for example, you enter the search term "Finishing tool", only single "Finishing tool" terms are displayed. Word combinations such as "Finishing tool_10" are not found.
- Exact expression
  Activate this option if you wish to search for terms with characters, which can also be used as place holders for other characters, e.g. "?" and "***".
Note
Search with place holders
When searching for specific program locations, you have the option of using place holders:
- "**": Replaces any character string
- "?": Replaces any character

Precondition
The desired program is opened in the editor.

Procedure

1. Press the "Search" softkey.
   A new vertical softkey bar appears.
   The "Search" window opens at the same time.
2. Enter the desired search term in the "Text" field.
3. Select "Whole words" if you want to search for whole words only.
   - OR -
   Activate the "Exact expression" checkbox if, for example, you want to search for place holders ("**", "?") in program lines.
4. Position the cursor in the "Direction" field and choose the search direction (forward, backward) with the <SELECT> key.
5. Press the "OK" softkey to start the search.
   If the text you are searching for is found, the corresponding line is highlighted.
6. Press the "Continue search" softkey if the text located during the search does not correspond to the point you are looking for.
   - OR -
   Press the "Cancel" softkey when you want to cancel the search.

Further search options

<table>
<thead>
<tr>
<th>Softkey</th>
<th>Function</th>
</tr>
</thead>
<tbody>
<tr>
<td>Go to start</td>
<td>The cursor is set to the first character in the program.</td>
</tr>
<tr>
<td>Go to end</td>
<td>The cursor is set to the last character in the program.</td>
</tr>
</tbody>
</table>
6.10.3 Replacing program text

You can find and replace text in one step.

Precondition

The desired program is opened in the editor.

Procedure

1. Press the "Search" softkey.
   A new vertical softkey bar appears.
2. Press the "Find and replace" softkey.
   The "Find and Replace" window appears.
3. In the "Text" field, enter the term you are looking for and in the "Replace with" field, enter the text you would like to insert automatically during the search.
4. Position the cursor in the "Direction" field and choose the search direction (forward, backward) with the <SELECT> key.
5. Press the "OK" softkey to start the search.
   If the text you are searching for is found, the corresponding line is highlighted.
6. Press the "Replace" softkey to replace the text.
   - OR -
   Press the "Replace all" softkey to replace all text in the file that corresponds to the search term.
   - OR -
   Press the "Continue search" softkey if the text located during the search should not be replaced.
   - OR -
   Press the "Cancel" softkey when you want to cancel the search.

Note

Replacing texts

- Read-only lines (;*RO*)
  If hits are found, the texts are not replaced.
- Contour lines (;*GP*)
  If hits are found, the texts are replaced as long as the lines are not read-only.
- Hidden lines (;*HD*)
  If hidden lines are displayed in the editor and hits are found, the texts are replaced as long as the lines are not read-only. Hidden lines that are not displayed, are not replaced.
6.10.4 Copying/pasting/deleting program blocks

In the editor, you edit both basic G code as well as program steps such as cycles, blocks and subprogram calls.

Inserting program blocks

The editor responds depending on what type of program block you insert.

- If you insert a G code, then the program block is directly inserted where the write mark is located.

- If you insert a program step, then the program block is always inserted at the next block, independent of the position of the write mark within the actual line. This is necessary as a cycle call always requires its own line. This behavior is in all applications, irrespective of whether the program step is inserted with a screen form using "Accept" or "Insert" is used as editor function.

**Note**

Cutout program step and reinsert

- If you cut out a program step at a specific location and you then directly reinsert it again, the sequence changes.

- Press the shortcut (key combination) <CTRL> + <Z> to undo what you have cut out.

Precondition

The program is opened in the editor.

Procedure

1. Press the "Mark" softkey.

   - OR -

   Press the <SELECT> key.

2. Select the desired program blocks with the cursor or mouse.

3. Press the "Copy" softkey in order to copy the selection to the buffer memory.

4. Place the cursor on the desired insertion point in the program and press the "Paste" softkey.

   The content of the buffer memory is pasted.

   - OR -

   Press the "Cut" softkey to delete the selected program blocks and to copy them into the buffer memory.

**Note:** When editing a program, you cannot copy or cut more than 1024 lines. While a program that is not on the NC is opened (progress display less than 100%), you cannot copy or cut more than 10 lines or insert more than 1024 characters.
Numbering the program blocks

If you have selected the "Automatic numbering" option for the editor, then the newly added program blocks are allocated a block number (N number).

The following rules apply:

- When creating a new program, the first line is allocated the "first block number".
- If, up until now, the program had no N number, then the program block inserted is allocated the starting block number defined in the "First block number" input field.
- If N numbers already exist before and after the insertion point of a new program block, then the N number before the insertion point is incremented by 1.
- If there are no N numbers before or after the insertion point, then the maximum N number in the program is increased by the "increment" defined in the settings.

Note:

After exiting the program, you have the option of renumbering the program blocks.

Note

The buffer memory contents are retained even after the editor is closed, enabling you to paste the contents in another program.

Note

Copy/cut current line

To copy and cut the current line where the cursor is positioned, it is not necessary to mark or select it. You have the option of making the "Cut" softkey only operable for marked program sections via editor settings.

6.10.5 Renumbering a program

You can modify the block numbering of programs opened in the editor at a later point in time.

Precondition

The program is opened in the editor.
Procedure

1. Press the ">>" softkey.
   A new vertical softkey bar appears.
2. Press the "Renumber" softkey.
   The "Renumbering" window appears.
3. Enter the values for the first block number and the increment to be used for numbering.
4. Press the "OK" softkey.
   The program is renumbered.

Note
- If you only want to renumber a section, before the function call, select the program blocks whose block numbering you want to edit.
- When you enter a value of "0" for the increment size, then all of the existing block numbers are deleted from the program and/or from the selected range.

6.10.6 Creating a program block

In order to structure programs to achieve a higher degree of transparency, you have the option of combining several G-code blocks to form program blocks.

Program blocks can be created in two stages. This means that you can form additional blocks within a block (nesting).

You then have the option of opening and closing these blocks depending on your requirement.

Settings for a program block

<table>
<thead>
<tr>
<th>Display</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td>Text</td>
<td>● Block designation</td>
</tr>
<tr>
<td>Spindle</td>
<td>● Selecting the spindle</td>
</tr>
<tr>
<td></td>
<td>◆ Defines at which spindle a program block is to be executed.</td>
</tr>
<tr>
<td>Addit. run-in code</td>
<td>● Yes</td>
</tr>
<tr>
<td></td>
<td>◆ For the case that the block is not executed, as the specified spindle should not be processed, then it is possible to temporarily activate a so-called &quot;Addit. run-in code&quot;.</td>
</tr>
<tr>
<td></td>
<td>◆ No</td>
</tr>
<tr>
<td>Automat. Retraction</td>
<td>● Yes</td>
</tr>
<tr>
<td></td>
<td>◆ Block start and block end are moved to the tool change point, i.e. the tool is brought into a safe range.</td>
</tr>
<tr>
<td></td>
<td>◆ No</td>
</tr>
</tbody>
</table>
Procedure

1. Select the "Program Manager" operating area.

2. Select the storage location and create a program or open a program. The program editor opens.

3. Select the required program blocks that you wish to combine to form a block.

4. Press the "Build block" softkey. The "Build Block" window opens.

5. Enter a designation for the block, assign the spindle, if required, select the additional run-in code and the automatic retraction and then press the "OK" softkey.

Opening and closing blocks

6. Press the ">>" and "View" softkeys.

7. Press the "Open blocks" softkey if you wish to display the program with all blocks.

8. Press the "Close blocks" softkey if you wish to display the program again in a structured form.

Remove block

9. Open the block.

10. Position the cursor at the end of the block.

11. Press the "Remove block" softkey.

Note

You can also open and close blocks with the mouse or using the cursor keys:

- `<Cursor right>` opens the block where the cursor is positioned
- `<Cursor left>` closes the block if the cursor is positioned at the beginning or end of the block
- `<ALT>` and `<Cursor left>` closes the block if the cursor is located within the block

Note

DEF statements in program blocks or block generation in the DEF part of a part program / cycle are not permitted.
6.10.7 Editor settings

Enter the default settings in the "Settings" window that are to take effect automatically when the editor is opened.

### Defaults

<table>
<thead>
<tr>
<th>Setting</th>
<th>Meaning</th>
</tr>
</thead>
</table>
| Number automatically          | ● Yes: A new block number will automatically be assigned after every line change. In this case, the specifications provided under "First block number" and "Increment" are applicable.  
● No: No automatic numbering |
| First block number            | Specifies the starting block number of a newly created program. The field is only visible when "Yes" is selected under "Number automatically". |
| Increment                     | Defines the increment used for the block numbers. The field is only visible when "Yes" is selected under "Number automatically".          |
| Display hidden lines          | ● Yes: Hidden lines marked with "**HD**" (hidden) will be displayed.  
● No: Lines marked with ";**HD;**" will not be displayed. Note: Only visible program lines are taken into account with the "Search" and "Search and Replace" functions. |
| Display block end as symbol   | The "LF" (line feed) symbol ¶ is displayed at the block end.                                                                          |
| Line break                    | ● Yes: Long lines are broken and wrapped around.  
● No: If the program includes long lines, then a horizontal scrollbar is displayed. You can move the section of the screen horizontally to the end of the line. |
| Line break also in cycle calls| ● Yes: If the line of a cycle call becomes too long, then it is displayed over several lines.  
● No: The cycle call is truncated. The field is only visible if "Yes" is entered under "Line break". |
| Visible programs              | ● 1 - 10  
Select how many programs can be displayed next to one another in the editor.  
● Auto  
Specifies that the number of programs entered in a job list or up to ten selected programs will be displayed next to each other. |
| Width of the program with focus| Here, you enter the width of the program that has the input focus in the editor as a percentage of the window width.                        |
| Save automatically            | ● Yes: The changes are saved automatically when you change to another operating area.  
● No: You are prompted to save when changing to another operating area. Save or reject the changes with the "Yes" and "No" softkeys.  
Note: Only for local and external drives. |

Grinding
Operating Manual, 06/2019, A5E44903521B AB
### Setting | Meaning
--- | ---
Cut only after selecting | - Yes: Parts of programs can only be cutout when program lines have been selected, i.e. the "Cutout" softkey only then is active.
- No: The program line, in which the cursor is positioned, can be cut out without having to select it.

Determine machining times | Defines which program runtimes are determined in the simulation or in automatic mode:
- Off: Program runtimes are not determined.
- Block-by-block: The runtimes are determined for each program block.
- Non-modal: The runtimes are determined at the NC block level. **Note:** You also have the option of displaying the cumulative times for blocks.
Please observe the information provided by the machine manufacturer. After the simulation or after executing the program, the required machining times are displayed in the editor.

Saving machining times | Specifies how the machining times determined are processed.
- Yes: A subdirectory with the name "GEN_DATA.WPD" is created in the directory of the part program. There, the machining times determined are saved in an ini file together with the name of the program. The machining times are displayed again when the program or job list are reloaded.
- No: The machining times that have been determined are only displayed in the editor.

Display cycles as machining step | - Yes: The cycle calls in the G code programs are displayed as plain text.
- No: The cycle calls in the G code programs are displayed in the NC syntax.
### Setting | Meaning
--- | ---
Highlight selected G code commands | Defines the display of G code commands.  
- **No**  
  All G code commands are displayed in the standard color.  
- **Yes**  
  Selected G code commands or keywords are highlighted in color. Define the rules for the color assignment in the seditorwidget.ini configuration file.  
  **Note:** Please observe the information provided by the machine manufacturer.

<table>
<thead>
<tr>
<th>Setting</th>
<th>Meaning</th>
</tr>
</thead>
</table>
| Font size | Defines the font size for the editor and the display of the program sequence.  
- **auto**  
  If you open a second program, then the smaller font size is automatically used.  
- **normal (16) - character height in pixels**  
  Standard font size that is displayed with the appropriate screen resolution.  
- **small (14) - character height in pixels**  
  More content is displayed in the editor.  
  **Note:** This setting also has an effect on the current block display.

**Note**

All entries that you make here are effective immediately.

**Requirement**

You have opened a program in the editor.

**Procedure**

1. Select the "Program" operating area.
2. Press the "Edit" softkey.
3. Press the ">>" and "Settings" softkeys.  
   The "Settings" window opens.
4. Make the required changes.
5. Press the "Delete mach. times" softkey if you wish to delete the machining times. The machining times that have been determined are deleted from the editor as well as from the actual block display. If the machining times are saved to an ini file, then this file is also deleted.

6. Press the "OK" softkey to confirm the settings.
6.11 Working with DXF files

6.11.1 Overview

The "DXF-Reader" function allows you to open files created in SINUMERIK Operate directly in a CAD system and accept and store contours directly in G-code.

The DXF file can be displayed in the Program Manager.

**Software option**

You require the "DXF-Reader" software option in order to use this function.

**Machine manufacturer**

Please refer to the machine manufacturer's specifications.

The DXF reader reads the following elements:

- "POINT"
- "LINE"
- "CIRCLE"
- "ARC"
- "TRACE"
- "SOLID"
- "TEXT"
- "SHAPE"
- "BLOCK"
- "ENDBLK"
- "INSERT"
- "ATTDEF"
- "ATTRIB"
- "POLYLINE"
- "VERTEX"
- "SEQEND"
- "3DLINE"
- "3DFACE"
- "DIMENSION"
- "LWPOLYLINE"
- "ELLIPSE"
- "LTYPE"
6.11.2 Displaying CAD drawings

6.11.2.1 Open a DXF file

Procedure

1. Select the "Program Manager" operating area.

2. Choose the desired storage location and position the cursor on the DFX file that you want to display.

3. Press the "Open" softkey. The selected CAD drawing will be displayed with all its layers, i.e. with all graphic levels.

4. Press the "Close" softkey to close the CAD drawing and to return to the Program Manager.

6.11.2.2 Cleaning a DXF file

All contained layers are shown when a DXF file is opened. Layers that do not contain any contour- or position-relevant data can be shown or hidden.

Requirement

The DXF file is open in the Program Manager or in the editor.
Procedure

1. Press the "Clean" and "Layer selection" softkeys if you want to hide specific layers.
   The "Layer Selection" window opens.

2. Deactivate the required layers and press the "OK" softkey.
   - OR -
   Press the "Clean automat." softkey to hide all non-relevant layers.

3. Press the "Clean automat." softkey to redisplay the layers.

6.11.2.3 Enlarging or reducing the CAD drawing

Requirement

The DXF file is opened in the Program Manager.

Procedure

1. Press the "Details" and "Zoom +" softkeys if you wish to enlarge the size of the segment.
   - OR -

2. Press the "Details" and "Zoom -" softkeys if you wish to reduce the size of the segment.
   - OR -

3. Press the "Details" and "Auto zoom" softkeys if you wish to automatically adapt the segment to the size of the window.
4. Press the "Details" and "Zoom elem. selection" softkeys if you want to automatically zoom elements that are in a selection set.

6.11.2.4 Changing the section

If you want to move or change the size of a section of the drawing, for example, to view details or redisplay the complete drawing later, use the magnifying glass. You can use the magnifying glass to determine the section and then change its size.

Requirement

The DXF file is opened in the Program Manager or in the editor.

Procedure

1. Press the "Details" and "Magnifying glass" softkeys. A magnifying glass in the shape of a rectangular frame appears.

2. Press the <+> key to enlarge the frame.

   - OR -
   Press the <-> key to reduce the frame.

   - OR -
   Press a cursor key to move the frame up, down, left or right.

3. Press the "OK" softkey to accept the section.

6.11.2.5 Rotating the view

You can change the orientation of the drawing.

Requirement

The DXF file is open in the Program Manager or in the editor.
6.11.2.6 Displaying/editing information for the geometric data

Precondition

The DXF file is opened in the Program Manager or in the editor.

Procedure

1. Press the "Details" and "Geometry info" softkeys.
   The cursor takes the form of a question mark.

2. Position the cursor on the element for which you want to display its geometric data and press the "Element Info" softkey.
   If, for example, you have selected a straight line, the following window opens "Straight line on layer: ...". You are shown the coordinates corresponding to the actual zero point in the selected layer: Start point for X and Y, end point for X and Y as well as the length.

3. Press the "Back" softkey to close the display window.

4. If you are currently in the editor, press the "Element edit" softkey.
   The coordinate values can be edited.

**Note**

**Editing a geometric element**

You can use this function to make smaller changes to the geometry, e.g. for missing intersections.

You should make larger changes in the input screen of the editor.

You cannot undo any changes that you make with "Element Edit".
6.11.3 Importing and editing a DXF file

6.11.3.1 General procedure

- Creating/opening a G code program
- Call the "Contour" cycle and create a "New contour"
- Import the DXF file
- Select the contour in the DXF file or CAD drawing and click "OK" to accept the cycle
- Add program record with "Accept" to the G code program

6.11.3.2 Setting the tolerance

To allow even inaccurately created drawings to be used, i.e. to compensate for gaps in the geometry, you can enter a snap radius in millimeters. This relates elements.

**Note**

**Large snap radius**

The larger that the snap radius is set, the larger the number of available following elements.

**Procedure**

1. The DXF file is opened in the editor.
2. Press the "Details" and "Snap radius" softkeys.
   The "Input" window appears.
3. Enter the desired value and press the "OK" softkey.

6.11.3.3 Assigning the machining plane

You can select the machining plane in which the contour created with the DXF reader should be located.

**Procedure**

1. The DXF file is opened in the editor.
2. Press the "Select plane" softkey.
   The "Select Plane" window opens.
3. Select the desired plane and press the "OK" softkey.
6.11.3.4 Selecting the machining range / deleting the range and element

You can select ranges in the DXF file and therefore reduce the elements. After accepting the 2nd position, only the contents of the selected rectangle are displayed. Contours are cut to the rectangle.

Requirement

The DXF file is open in the editor.

Procedure

Select the machining range from the DXF file

1. Press the "Reduce" and "Select range" softkeys if you want to select specific ranges of the DXF file.
   An orange rectangle is displayed.
2. Press the "Range +" softkey to enlarge the section or press the "Range -" softkey to reduce the section.
3. Press the "Arrow right", "Arrow left", "Arrow up" or "Arrow down" softkey to move the selection tool.
4. Press the "OK" softkey.
   The machining section is displayed.
   Use the "Cancel" softkey to return to the previous window.
5. Press the "Deselect range" softkey to undo the selection of the machining range.
   The DXF file is reset to the original display.

Delete selected ranges and elements of the DXF file

6. Press the "Reduce" softkey.

Delete range

7. Press the "Range delete" softkey.
   A blue rectangle is displayed.
8. Press the "Range +" softkey to enlarge the section or press the "Range -" softkey to reduce the section.
9. Press the "Arrow right", "Arrow left", "Arrow up" or "Arrow down" softkey to move the selection tool.

- OR -

Delete element

10. Press the "Element delete" softkey, and using the selection tool, select the element that you wish to delete.

11. Press "OK".

6.11.3.5 Saving the DXF file

You can save DXF files that you have reduced and edited.

Requirement

The DXF file is open in the editor.

Procedure

1. Reduce file according to your requirements and/or select the working areas.

- OR -

2. Press the "Back" and ">>" softkeys.

3. Press the "Save DXF" softkey.

4. Enter the required name in the "Save DXF Data" window and press "OK". The "Save As" window opens.

5. Select the required storage location.

6. If required, press the "New directory" softkey, enter the required name in the "New Directory" window and press the "OK" softkey to create a directory.

7. Press the "OK" softkey.
6.11.3.6 Specifying a reference point

Because the zero point of the DXF file normally differs from the zero point of the CAD drawing, specify a reference point.

Procedure

1. The DXF file is opened in the editor.
2. Press the ">>" and "Specify reference point" softkeys.
3. Press the "Element start" softkey to place the zero point at the start of the selected element.
   - OR -
   Press the "Element center" softkey to place the zero point at the center of the selected element.
   - OR -
   Press the "Element end" softkey to place the zero point at the end of the selected element.
   - OR -
   Press the "Arc center" softkey to place the zero point at the center of an arc.
   - OR -
   Press the "Cursor" softkey to define the zero point at any cursor position.
   - OR -
   Press the "Free input" softkey to open the "Reference Point Input" window and enter the values for the positions (X, Y) there.

6.11.3.7 Accepting contours

1. The part program to be processed has been created and you are in the editor.
2. Press the "Contour" softkey.
3. Press the "New contour" softkey.

Select contour

The start and end point are specified for the contour line.
The start point and the direction are selected on a selected element. Beginning at the start point, the automatic contour line takes all subsequent elements of a contour. The contour line
ends as soon as there are no subsequent elements – or intersections with other elements of the
contour occur.

Note
If a contour includes more elements than can be processed, you will be offered the option of
transferring the contour to the program as pure G code.
This contour then can no longer be edited in the editor.

With the "Undo" softkey, you can undo your contour selection back to a
specific point.

Procedure

Opening a DXF file
1. Enter the desired name in the "New Contour" window.
2. Press the "From DXF file" and "Accept" softkeys.
The "Open DXF File" window opens.
3. Select a storage location and place the cursor on the relevant DXF file.
   You can, for example, use the search function to search directly for a DXF
   file in comprehensive folders and directories.
4. Press the "OK" softkey.
The CAD drawing opens and can be edited for contour selection.
The cursor takes the form of a cross.

Specifying a reference point
5. If required, specify a zero point.

Contour line
6. Press the ">>" and "Automatic" softkeys if you want to accept the largest
   possible number of contour elements.
   This makes it fast to accept contours that consist of many individual ele-
   ments.
   - OR -
   Press "Only to 1st cut" if you do not want to accept the complete contour
   elements at once.
   The contour will be followed to the first cut of the contour element.

Defining the start point
7. Press the "Select element" softkey to select the desired element.
8. Press the "Accept element" softkey.
9. Press the "Element start point" softkey to place the contour start at the start point of the element.
- OR -
Press the "Element end point" softkey to place the contour start at the end point of the element.
- OR -
Press the "Element center" softkey to place the contour start at the center of the element.
- OR -
Press the "Cursor" softkey to define the start of the element with the cursor at any position.

9. Press the "OK" softkey to confirm your selection.

10. Press the "Accept element" softkey to accept the offered elements.
The softkey can be operated while elements are still available to be accepted.

Specifying the end point

11. Press the ">>" and "Specify end point" softkeys if you do not want to accept the end point of the selected element.

12. Press the "Current position" softkey if you want to set the currently selected position as end point.
- OR -
Press the "Element center" softkey to place the contour end at the center of the element.
- OR -
Press the "Element end" softkey to place the contour end at the end of the element.
- OR -
Press the "Cursor" softkey to define the start of the element with the cursor at any position.

Transferring the contour to the cycle and to the program

Press the "OK" softkey.
The selected contour is transferred to the contour input screen of the editor.
Press the "Accept contour" softkey.
The program block is transferred to the program.

Operation with mouse and keyboard

In addition to operation using softkeys, you can also operate the functions with the keyboard and with the mouse.
6.12 Display and edit user variables

6.12.1 Overview

The defined user data may be displayed in lists.

User variables

The following variables can be defined:
- Global arithmetic parameters (RG)
- Arithmetic parameters (R parameters)
- Global user data (GUD) is valid in all programs
- Local user variables (LUD) are valid in the program where they have been defined.
- Program-global user variables (PUD) are valid in the program in which they have been defined, as well as in all of the subprograms called by this program

Channel-specific user data can be defined with a different value for each channel.

Entering and displaying parameter values

Up to 15 positions (including decimal places) are evaluated. If you enter a number with more than 15 places, it will be written in exponential notation (15 places + EXXX).

LUD or PUD

Only local or program-global user data can be displayed at one time.

Whether the user data are available as LUD or PUD depends on the current control configuration.

Machine manufacturer

Please observe the information provided by the machine manufacturer.

Note

Reading and writing variables protected

Reading and writing of user data are protected via a keyswitch and protection levels.

Comments

For R arithmetic parameters and global R parameters you have the option of saving associated comments.

Searching for user data

You may search for user data within the lists using any character string.
6.12.2 Global R parameters

Global R parameters are arithmetic parameters, which exist in the control itself, and can be read or written to by all channels.

You use global R parameters to exchange information between channels, or if global settings are to be evaluated for all channels.

These values are retained after the controller is switched off.

Comments

You can save comments in the "Global R parameters with comments" window. These comments can be edited. You have the option of either individually deleting these comments, or using the delete function.

These comments are retained after the control is switched off.

Number of global R parameters

The number of global R parameters is defined in a machine data element.

Range: RG[0]– RG[999] (dependent on the machine data).

There are no gaps in the numbering within the range.

Machine manufacturer

Please observe the information provided by the machine manufacturer.

Procedure

1. Select the "Parameter" operating area.
2. Press the "User variable" softkey.
3. Press the "Global R parameters" softkey.

The "Global R parameters" window opens.
Display comments

1. Press the ">>" and "Display comments" softkeys.
   The "Global R parameters with comments" window opens.

2. Press the "Display comments" softkey once again to return to the "Global R parameters" window.

Deleting R parameters and comments

1. Press the ">>" and "Delete" softkeys.
   The "Delete global R parameters" window opens.

2. In fields "from global R parameters" and "to global R parameters", select the global R parameters whose values you wish to delete.
   - OR -
   Press the "Delete all" softkey.

3. Activate the checkbox "also delete comments" if the associated comments should also be automatically deleted.

4. Press the "OK" softkey.

   - A value of 0 is assigned to the selected global R parameters – or to all global R parameters.
   - The selected comments are also deleted.

6.12.3 R parameters

R parameters (arithmetic parameters) are channel-specific variables that you can use within a G code program. G code programs can read and write R parameters.

These values are retained after the controller is switched off.

Comments

You can save comments in the "R parameters with comments" window.

These comments can be edited. You have the option of either individually deleting these comments, or using the delete function.

These comments are retained after the control is switched off.

Number of channel-specific R parameters

The number of channel-specific R parameters is defined in a machine data element.

Range: R0-R999 (dependent on machine data).
There are no gaps in the numbering within the range.

**Machine manufacturer**

Please observe the information provided by the machine manufacturer.

**Procedure**

1. Select the "Parameter" operating area.
2. Press the "User variable" softkey.
3. Press the "R variables" softkey. The "R parameters" window appears.

**Display comments**

1. Press the ">>" and "Display comments" softkeys. The "R parameters with comments" window opens.
2. Press the "Display comments" softkey once again to return to the "R parameters" window.

**Delete R variables**

1. Press the ">>" and "Delete" softkeys. The "Delete R parameters" window appears.
2. In fields "from R parameters" and "to R parameters", select the R parameters whose values you wish to delete.
   - OR -
   Press the "Delete all" softkey.
3. Activate the checkbox "also delete comments" if the associated comments should also be automatically deleted.
4. Press the "OK" softkey.

- A value of 0 is assigned to the selected R parameters or to all R parameters.
- The selected comments are also deleted.
6.12.4 Displaying global user data (GUD)

Global user variables

Global GUDs are NC global user data (Global User Data) that remains available after switching the machine off.

GUDs apply in all programs.

Definition

A GUD variable is defined with the following:

- Keyword DEF
- Range of validity NCK
- Data type (INT, REAL, ...)
- Variable names
- Value assignment (optional)

Example

DEF NCK INT ZAEHLER1 = 10

GUDs are defined in files with the ending DEF. The following file names are reserved for this purpose:

<table>
<thead>
<tr>
<th>File name</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td>MGUD.DEF</td>
<td>Definitions for global machine manufacturer data</td>
</tr>
<tr>
<td>UGUD.DEF</td>
<td>Definitions for global user data</td>
</tr>
<tr>
<td>GUD4.DEF</td>
<td>User-definable data</td>
</tr>
<tr>
<td>GUD8.DEF, GUD9.DEF</td>
<td>User-definable data</td>
</tr>
</tbody>
</table>

Procedure

1. Select the "Parameter" operating area.
2. Press the "User variable" softkey.
3. Press the "Global GUD" softkeys.

The "Global User Variables" window is displayed. A list of the defined UGUD variables will be displayed.

- OR -
Press the "GUD selection" softkey and the "SGUD" to "GUD6" softkeys if you wish to display SGUD, MGUD, UGUD as well as GUD4 to GUD 6 of the global user variables.

- OR -
Press the "GUD selection" and ">>" softkeys as well as the "GUD7" to "GUD9" softkeys if you want to display GUD 7 to GUD 9 of the global user variables.

Note
After each start-up, a list with the defined UGUD variables is displayed in the "Global User Variables" window.

6.12.5 Displaying channel GUDs

Channel-specific user variables
Like the GUDs, channel-specific user variables are applicable in all programs for each channel. However, unlike GUDs, they have specific values.

Definition
A channel-specific GUD variable is defined with the following:

- Keyword DEF
- Range of validity CHAN
- Data type
- Variable names
- Value assignment (optional)

Example
DEF CHAN REAL X_POS = 100.5
Procedure

1. Select the "Parameter" operating area.

2. Press the "User variable" softkey.

3. Press the "Channel GUD" and "GUD selection" softkeys.

   A new vertical softkey bar appears.

4. Press the "SGUD" ... "GUD6" softkeys if you want to display the SGUD, MGUD, UGUD as well as GUD4 to GUD 6 of the channel-specific user variables.

   - OR -

   Press the "Continue" softkey and the "GUD7" ... "GUD9" softkeys if you want to display GUD 7 and GUD 9 of the channel-specific user variables.

6.12.6 Displaying local user data (LUD)

Local user variables

LUDs are only valid in the program or subprogram in which they were defined.

The controller displays the LUDs after the start of program processing. The display is available until the end of program processing.

Definition

A local user variable is defined with the following:

- Keyword DEF
- Data type
- Variable names
- Value assignment (optional)
**Procedure**

1. Select the "Parameter" operating area.
2. Press the "User variable" softkey.
3. Press the "Local LUD" softkey.

6.12.7 **Displaying program user data (PUD)**

**Program-global user variables**

PUDs are global part program variables (Program User Data). PUDs are valid in all main programs and subprograms, where they can also be written and read.

**Machine manufacturer**

Please refer to the machine manufacturer's specifications.

**Procedure**

1. Select the "Parameter" operating area.
2. Press the "User variable" softkey.
3. Press the "Program PUD" softkey.

6.12.8 **Searching for user variables**

You can search for R parameters and user variables.
Procedure

1. Select the "Parameter" operating area.

2. Press the "User variable" softkey.

3. Press the "R parameters", "Global GUD", "Channel GUD", "Local GUD" or "Program PUD" softkeys to select the list in which you would like to search for user variables.

4. Press the "Search" softkey.

The "Search for R Parameters" or "Search for User Variables" window opens.

5. Enter the desired search term and press "OK".

The cursor is automatically positioned on the R parameters or user variables you are searching for, if they exist.

By editing a DEF/MAC file, you can alter or delete existing definition/macro files or add new ones.

Procedure

1. Select the "Start-up" operating area.

2. Press the "System data" softkey.

3. In the data tree, select the "NC data" folder and then open the "Definitions" folder.

4. Select the file you want to edit.

5. Double-click the file.
   - OR -
   Press the "Open" softkey.

   - OR -
   Press the <INPUT> key.

   - OR -
   Press the <Cursor right> key.

The selected file is opened in the editor and can be edited there.
6. Define the desired user variable.
7. Press the "Exit" softkey to close the editor.

**Activating user variables**

1. Press the "Activate" softkey.
   
   A prompt is displayed.
2. Select whether the current values in the definition files should be retained
   - OR -
   Select whether the current values in the definition files should be deleted.
   This will overwrite the definition files with the initial values.
3. Press the "OK" softkey to continue the process.
6.13 Display G and auxiliary functions

6.13.1 Selected G functions

16 selected G groups are displayed in the "G Function" window.

Within a G group, the G function currently active in the controller is displayed.

Some G codes (e.g. G17, G18, G19) are immediately active after switching the machine control on.

Which G codes are always active depends on the settings.

Please observe the information provided by the machine manufacturer.

### G groups displayed by default

<table>
<thead>
<tr>
<th>Group</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td>G group 1</td>
<td>Modally active motion commands (e.g. G0, G1, G2, G3)</td>
</tr>
<tr>
<td>G group 2</td>
<td>Non-modally active motion commands, dwell time (e.g. G4, G74, G75)</td>
</tr>
<tr>
<td>G group 3</td>
<td>Programmable offsets, working area limitations and pole programming (e.g. TRANS, ROT, G25, G110)</td>
</tr>
<tr>
<td>G group 6</td>
<td>Plane selection (e.g. G17, G18)</td>
</tr>
<tr>
<td>G group 7</td>
<td>Tool radius compensation (e.g. G40, G42)</td>
</tr>
<tr>
<td>G group 8</td>
<td>Settable work offset (e.g. G54, G57, G500)</td>
</tr>
<tr>
<td>G group 9</td>
<td>Offset suppression (e.g. SUPA, G53)</td>
</tr>
<tr>
<td>G group 10</td>
<td>Exact stop - continuous-path mode (e.g. G60, G641)</td>
</tr>
<tr>
<td>G group 13</td>
<td>Workpiece dimensioning inches/metric (e.g. G70, G700)</td>
</tr>
<tr>
<td>G group 14</td>
<td>Workpiece dimensioning absolute/incremental (G90)</td>
</tr>
<tr>
<td>G group 15</td>
<td>Feedrate type (e.g. G93, G961, G972)</td>
</tr>
<tr>
<td>G group 16</td>
<td>Feedrate override on inside and outside curvature (e.g. CFC)</td>
</tr>
<tr>
<td>G group 21</td>
<td>Acceleration profile (e.g. SOFT, DRIVE)</td>
</tr>
<tr>
<td>G group 22</td>
<td>Tool offset types (e.g. CUT2D, CUT2DF)</td>
</tr>
<tr>
<td>G group 29</td>
<td>Radius/diameter programming (e.g. DIAMOF, DIAMCYCOF)</td>
</tr>
<tr>
<td>G group 30</td>
<td>Compressor ON/OFF (e.g. COMPOF)</td>
</tr>
</tbody>
</table>

### G groups displayed by default (ISO code)

<table>
<thead>
<tr>
<th>Group</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td>G group 1</td>
<td>Modally active motion commands (e.g. G0, G1, G2, G3)</td>
</tr>
<tr>
<td>G group 2</td>
<td>Non-modally active motion commands, dwell time (e.g. G4, G74, G75)</td>
</tr>
<tr>
<td>G group 3</td>
<td>Programmable offsets, working area limitations and pole programming (e.g. TRANS, ROT, G25, G110)</td>
</tr>
<tr>
<td>G group 6</td>
<td>Plane selection (e.g. G17, G18)</td>
</tr>
</tbody>
</table>
### Group and Meaning

<table>
<thead>
<tr>
<th>Group</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td>G group 7</td>
<td>Tool radius compensation (e.g. G40, G42)</td>
</tr>
<tr>
<td>G group 8</td>
<td>Settable work offset (e.g. G54, G57, G500)</td>
</tr>
<tr>
<td>G group 9</td>
<td>Offset suppression (e.g. SUPA, G53)</td>
</tr>
<tr>
<td>G group 10</td>
<td>Exact stop - continuous-path mode (e.g. G60, G641)</td>
</tr>
<tr>
<td>G group 13</td>
<td>Workpiece dimensioning inches/metric (e.g. G70, G700)</td>
</tr>
<tr>
<td>G group 14</td>
<td>Workpiece dimensioning absolute/incremental (G90)</td>
</tr>
<tr>
<td>G group 15</td>
<td>Feedrate type (e.g. G93, G961, G972)</td>
</tr>
<tr>
<td>G group 16</td>
<td>Feedrate override on inside and outside curvature (e.g. CFC)</td>
</tr>
<tr>
<td>G group 21</td>
<td>Acceleration profile (e.g. SOFT, DRIVE)</td>
</tr>
<tr>
<td>G group 22</td>
<td>Tool offset types (e.g. CUT2D, CUT2DF)</td>
</tr>
<tr>
<td>G group 29</td>
<td>Radius/diameter programming (e.g. DIAMOF, DIAMCYCOF)</td>
</tr>
<tr>
<td>G group 30</td>
<td>Compressor ON/OFF (e.g. COMPOF)</td>
</tr>
</tbody>
</table>

### Procedure

1. Select the "Machine" operating area.

2. Press the <JOG>, <MDI> or <AUTO> key.

3. Press the "G functions" softkey.
   
The "G Functions" window is opened.

4. Press the "G functions" softkey again to hide the window.

The G groups selection displayed in the "G Functions" window may differ.

### Machine manufacturer

Please observe the information provided by the machine manufacturer.

### Further information

Further information about configuring the displayed G groups can be found in the SINUMERIK Operate Commissioning Manual.
6.13.2 All G functions

All G groups and their group numbers are listed in the "G Functions" window. Within a G group, only the G function currently active in the controller is displayed.

Additional information in the footer

The following additional information is displayed in the footer:

- Actual transformations

<table>
<thead>
<tr>
<th>Display</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td>TRANSMIT</td>
<td>Polar transformation active</td>
</tr>
<tr>
<td>TRACYL</td>
<td>Cylinder surface transformation active</td>
</tr>
<tr>
<td>TRAORI</td>
<td>Orientation transformation active</td>
</tr>
<tr>
<td>TRAANG</td>
<td>Inclined axis transformation active</td>
</tr>
<tr>
<td>TRACON</td>
<td>Cascaded transformation active</td>
</tr>
<tr>
<td></td>
<td>For TRACON, two transformations (TRAANG and TRACYL or TRAANG and</td>
</tr>
<tr>
<td></td>
<td>TRANSMIT) are activated in succession.</td>
</tr>
</tbody>
</table>

- Current work offsets
- Spindle speed
- Path feedrate
- Active tool

6.13.3 G functions for mold making

In the window "G functions", important information for machining free-form surfaces can be displayed using the "High Speed Settings" function (CYCLE832).

Software option

You require the "Advanced Surface" software option in order to use this function.

High-speed cutting information

In addition to the information that is provided in the "All G functions" window, the following programmed values of the following specific information is also displayed:

- CTOL
- OTOL
- STOLF

The tolerances for G0 are only displayed if they are active.

Particularly important G groups are highlighted.

You have the option to configure which G functions are highlighted.
Further information

Further information about the contour tolerance and configuring the displayed G groups can be found at:

- Basic Functions Function Manual: Chapter "Contour/Orientation Tolerance"
- SINUMERIK Operate Commissioning Manual

Procedure

1. Select the "Machine" operating area

2. Press the <JOG>, <MDI> or <AUTO> key.

3. Press the ">>" and "All G functions" softkeys. The "G Functions" window is opened.

6.13.4 Auxiliary functions

Auxiliary functions include M and H functions preprogrammed by the machine manufacturer, which transfer parameters to the PLC to trigger reactions defined by the manufacturer.

Displayed auxiliary functions

Up to five current M functions and three H functions are displayed in the "Auxiliary Functions" window.
Procedure

1. Select the "Machine" operating area.

2. Press the <JOG>, <MDA> or <AUTO> key.

3. Press the "H functions" softkey.
   The "Auxiliary Functions" window opens.

4. Press the "H functions" softkey again to hide the window again.
6.14 Displaying superimpositions

You can display handwheel axis offsets or programmed superimposed movements in the "Superimpositions" window.

<table>
<thead>
<tr>
<th>Input field</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td>Tool</td>
<td>Current superimposition in the tool direction</td>
</tr>
<tr>
<td>Min</td>
<td>Minimum value for superimposition in the tool direction</td>
</tr>
<tr>
<td>Max</td>
<td>Maximum value for superimposition in the tool direction</td>
</tr>
<tr>
<td>DRF</td>
<td>Displays the handwheel axis offset</td>
</tr>
</tbody>
</table>

The selection of values displayed in the "Superimposition" window may differ.

**Machine manufacturer**

Please observe the information provided by the machine manufacturer.

**Procedure**

1. Select the "Machine" operating area.
2. Press the <AUTO>, <MDI> or <JOG> key.
3. Press the ">>" and "Superimposition" softkeys. The "Superimposition" window opens.
4. Enter the required new minimum and maximum values for superimposition and press the <INPUT> key to confirm your entries.
   Note: You can only change the superimposition values in "JOG" mode.
5. Press the "Superimposition" softkey again to hide the window.
6.15 Displaying status of synchronized actions

You can display status information for diagnosing synchronized actions in the "Synchronized Actions" window.

You get a list with all currently active synchronized actions.

In this list, the synchronized action programming is displayed in the same form as in the part program.

References

Programming Guide Job Planning (PGA) Chapter: Motion-synchronous actions

Status of synchronized actions

You can see the status of the synchronized actions in the "Status" column.

- Waiting
- Active
- Blocked

Non-modal synchronized actions can only be identified by their status display. They are only displayed during execution.

Synchronization types

<table>
<thead>
<tr>
<th>Synchronization types</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td>ID=n</td>
<td>Modal synchronized actions in the automatic mode up to the end of program, local to program; n = 1... 254</td>
</tr>
<tr>
<td>IDS=n</td>
<td>Static synchronized actions, modally effective in every operating type, also beyond the end of program; n = 1... 254</td>
</tr>
<tr>
<td>Without ID/IDS</td>
<td>Non-modal synchronized actions in the automatic mode</td>
</tr>
</tbody>
</table>

Note

Numbers from the number range 1 to 254 can only be assigned once, irrespective of the identification number.

Display of synchronized actions

Using softkeys, you have the option of restricting the display to activated synchronized actions.
Procedure

1. Select the "Machine" operating area.

2. Press the <AUTO>, <MDA> or <JOG> key.

3. Press the menu forward key and the "Synchron." softkey.
   The "Synchronized Actions" window appears.
   You obtain a display of all activated synchronized actions.

4. Press the "ID" softkey if you wish to hide the modal synchronized actions in the automatic mode.
   - AND / OR -
   Press the "IDS" softkey if you wish to hide static synchronized actions.
   - AND / OR -
   Press the "Blockwise" softkey if you wish to hide the non-modal synchronized actions in the automatic mode.

5. Press the "ID", "IDS" or "Blockwise" softkeys to re-display the corresponding synchronized actions.

...
6.16 Displaying the program runtime and counting workpieces

To gain an overview of the program runtime and the number of machined workpieces, open the "Times, Counter" window.

Machine manufacturer
Please observe the information provided by the machine manufacturer.

Displayed times

- Program
  Pressing the softkey the first time shows how long the program has already been running. At every further start of the program, the time required to run the entire program the first time is displayed. If the program or the feedrate is changed, the new program runtime is corrected after the first run.

- Program remainder
  Here you can see how long the current program still has to run. In addition, you can follow how much of the current program has been completed in percent on a progress bar. The first program execution differs in the calculation of the additional program executions. When a program is executed for the first time, the progress is estimated based on the program size and the actual program offset. The larger the program and the more linear that it is executed, the more precise the first estimate. This estimate is very inaccurate as a result of the system for programs with steps and/or subprograms. For each additional program execution, the measured overall program execution time is used as basis for the program progress display.

- Influencing the time measurement
  The time measurement is started with the start of the program and ends with the end of the program (M30) or with an agreed M function. When the program is running, the time measurement is interrupted with CYCLE STOP and continued with CYCLE START. The time measurement starts at the beginning with RESET and subsequent CYCLE START. The time measurement stops with CYCLE STOP or a feedrate override = 0.

Counting workpieces

You can also display program repetitions and the number of completed workpieces. For the workpiece count, enter the actual and planned workpiece numbers.

Workpiece count

Completed workpieces can be counted via the end of program command (M30) or an M command.
**Procedure**

1. Select the "Machine" operating area.

2. Press the <AUTO> key.

3. Press the "Times, Counter" softkey. The "Times, Counter" window opens.

4. Select "Yes" under "Count workpieces" if you want to count completed workpieces.

5. Enter the number of workpieces needed in the " Desired workpieces" field. The number of workpieces already finished is displayed in "Actual workpieces". You can correct this value when required. After the defined number of workpieces is reached, the current workpieces display is automatically reset to zero.
6.17 Settings for the automatic mode

Before machining a workpiece, you can test the program in order to identify programming errors at an early stage. Use the dry run feedrate for this purpose.

In addition, you have the option of additionally limiting the traversing speed for rapid traverse so that when running-in a new program with rapid traverse, no undesirable high traversing speeds occur.

Dry run feedrate

The feedrate defined here replaces the programmed feedrate during execution if you have selected "DRY run feedrate" under program control.

Reduced rapid traverse

The value entered here reduces the rapid traverse to the entered percentage value if you selected "RG0 reduced rapid traverse" under program control.

Recording machining times

To provide support when creating and optimizing a program, you have the option of displaying the machining times.

You define whether the time determination function is activated while the workpiece is being machined.

- Off
  The time determination function is deactivated while machining a workpiece, i.e. the machining times are not determined.

- Non-modal
  The machining times are determined for each traversing block of a main program.

- Block-by-block
  Machining times are determined for all blocks.

Note

Utilization of resources

The more machining times are displayed, the more resources are utilized.

More machining times are determined and saved with the non-modal setting as with the block-by-block setting.

Saving machining times

Here, you specify how the determined machining times are processed.

- Yes
  A subdirectory with the name "GEN_DATA.WPD" is created in the directory of the part program. The determined machining times are saved there in an ini file together with the name of the program.

- No
  The determined machining times are displayed only in the program block display.
Procedure

1. Select the "Machine" operating area.

2. Press the <AUTO> key.

3. Press the menu forward key and the "Settings" softkey. The "Settings for Automatic Operation" window opens.

4. In "DRY run feedrate," enter the desired dry run speed.

5. Enter the desired percentage in the "Reduced rapid traverse RG0" field. RG0 has not effect if you do not change the specified amount of 100%.

6. Select the required entry in the "Record machining times" and where relevant, in the "Save machining times" field.

See also

Current block display (Page 40)
7 Simulating machining

7.1 Overview

During simulation, the current program is calculated in its entirety and the result is displayed in graphical form. The result of programming is verified without moving the machine axes. Incorrectly programmed machining steps are detected at an early stage and incorrect machining on the workpiece is prevented.

Graphical display

In order that workpiece simulation is also possible for tools that have either not been measured or have been incompletely entered, certain assumptions are made regarding the tool geometry. For instance, the length of a grinding wheel or a dressing tool is set to a value proportional to the tool radius so that cutting can be simulated.

Defining a blank

The blank dimensions that are entered in the program editor are used for the workpiece. The blank is clamped with reference to the coordinate system that was valid when the blank was defined. This means that before the blank is defined in G code programs, the required output conditions must be established, e.g. by selecting a suitable zero offset.

Display of the traversing paths

The traversing paths are displayed in color. Rapid traverse is red and the feedrate is green.

Depth display

The depth infeed is color-coded. The depth display indicates the actual depth at which machining is currently taking place. "The deeper, the darker" applies for the depth display.

Machine coordinate system references

The simulation is implemented as workpiece simulation. This means that it is not assumed that the work offset has already been precisely scratched or is known.

In spite of this, there are unavoidable machine coordinate system references in the programming, such as the tool change point in the machine coordinate system, the retraction position for swiveling, and the table components of swivel kinematics. Depending on the current work offset, in the worst case, these machine coordinate system references can mean that collisions are shown in the simulation that would not occur with a realistic work offset, or conversely that collisions are not shown that could occur with a realistic work offset.
Simulating machining

7.1 Overview

Programmable frames
All frames and zero offsets are taken into account in the simulation.

Note
Manually swiveled axes
Note that swivel movement in simulation and during simultaneous recording is also displayed when the axes are swiveled manually at the start.

Simulation display
For the simulation or simultaneous recording, the traversing paths, i.e. the programmed tool paths, are displayed.

Display variants
You can choose between three variants of the graphical display:

- Simulation before machining of the workpiece
  Before you machine the workpiece on the machine, you will graphically display a quick run-through of how the program will be executed.

- Simultaneous recording before machining of the workpiece
  Before machining the workpiece on the machine, you can graphically display how the program will be executed during the program test and dry run feedrate. The machine axes do not move if you have selected "no axis motion".

- Simultaneous recording during machining of the workpiece
  You can follow machining of the workpiece on the screen while the program is being executed on the machine.

Views of simultaneous recording and simulation
The following views are available for all three variants:

- Side view
- Dressing view
- 3D view (with option)
- Face view - for cylindrical grinding
- Further side views - surface grinding
- Machine space (with "collision avoidance" option) - 840D sl only

Status display
The current axis coordinates, the override, the current tool with cutting edge, the current program block, the feedrate and the machining time are displayed.
In all views, a clock is displayed during graphical processing. The machining time is displayed in hours, minutes and seconds. It is approximately equal to the time that the program requires for processing including the tool change.

**Software options**

You require the "3D simulation of the finished part" option for the 3D view.
You require the option "Simultaneous recording (real-time simulation)" for the "Simultaneous recording" function.

**Determining the program runtime**

The program runtime is determined when executing the simulation. The program runtime is displayed in the editor until the next program change at the end of the program.

**Properties of simultaneous recording and simulation**

**Traversing paths**

In simulation, the traversing paths displayed are saved in a ring buffer. If this buffer is full, then the oldest traversing path is deleted with each new traversing path.

**Optimum display**

If simultaneous machining is stopped or has been completed, the display is again converted into a high-resolution image. In some cases, this is not possible. In such cases, the following message is output: "High-resolution image cannot be generated".

**Working zone limitation**

No working area limitations and software limit switches are effective in the tool simulation.

**Start position for simulation and simultaneous recording**

During simulation, the start position is converted to the workpiece coordinate system by application of the zero offset.

The simultaneous recording starts at the position at which the machine is currently located.

**Restrictions**

- Traori: 5-axis motion is linearly interpolated. More complex motion cannot be displayed.
- Referencing: G74 from a program run does not function.
- Alarm 15110 "REORG block not possible" is not displayed.
- Compile cycles are only partly supported.
- No PLC support.
- Axis containers are not supported.

**Supplementary conditions**

- All of the existing data records (Toolcarrier / TRAORI, TRACYL) are evaluated and must be correctly commissioned for correct simulation.
- Transformations with swiveled linear axis (TRAORI 64 - 69) as well as OEM transformations (TRAORI 4096 - 4098) are not supported.
Simulating machining

7.1 Overview

- Changes to the toolcarrier or transformation data only become effective after Power On.
- Transformation change and swivel data record change are supported. However, a real kinematic change is not supported, where a swivel head is physically changed.
7.2 Simulation before machining of the workpiece

7.2.1 Simulation before machining of the workpiece

Before machining the workpiece on the machine, you have the option of performing a quick run-through in order to graphically display how the program will be executed. This provides a simple way of checking the result of the programming.

Feedrate override

The rotary switch (override) on the control panel only influences the functions of the "Machine" operating area.

Press the "Program control" softkey to change the simulation feedrate. You can select the simulation feedrate in the range 0 to 120%.

7.2.2 Start simulation

Procedure

1. Select the "Program Manager" operating area.

2. Select the storage location and position the cursor on the program to be simulated.

3. Press the <INPUT> or <Cursor right> key.

- OR -

Double-click the program.

The selected program is opened in the "Program" operating area.

4. Press the "Simulation" softkey.

- OR -

5. Press "Start" softkey.

The program execution is displayed graphically on the screen. The machine axes do not move.

6. To stop simulation, press the "Stop" softkey.

- OR -
Press the "Reset" softkey to cancel the simulation.

7. Press the "Start" softkey to restart or continue the simulation.

---

Note

Operating area switchover

The simulation is exited if you switch into another operating area. If you restart the simulation, then this starts again at the beginning of the program.
7.3 Simultaneous recording before machining of the workpiece

7.3.1 Overview

Before machining the workpiece on the machine, you can graphically display the execution of the program on the screen to monitor the result of the programming.

Software option

You require the option "Simultaneous recording (real-time simulation)" for the simultaneous recording.

You can replace the programmed feedrate with a dry run feedrate to influence the speed of execution and select the program test to disable axis motion.

If you would like to view the current program blocks again instead of the graphical display, you can switch to the program view.

7.3.2 Starting simultaneous recording

Procedure

1. Load a program in the "AUTO" mode.

2. Press the "Prog. ctrl." softkey and activate the checkboxes "PRT no axis movement" and "DRY run feedrate".

   The program is executed without axis movement. The programmed feedrate is replaced by a dry run feedrate.

3. Press the "Sim. rec." softkey.

   - OR -

4. Press the <CYCLE START> key.

   The program execution is displayed graphically.

5. Press the "Sim. rec." softkey again to stop the recording.

   - OR -
7.4 Simultaneous recording during machining of the workpiece

If the view of the work space is blocked by coolant, for example, while the workpiece is being machined, you can also track the program execution.

**Software option**

You require the option "Simultaneous recording (real-time simulation)" for simultaneous recording.

**Procedure**

1. Load a program in the "AUTO" mode.
2. Press the "Sim. rec." softkey.

- OR

3. Press the <CYCLE START> key.
   The machining of the workpiece is started and graphically displayed.

- OR

4. Press the "Sim. rec." softkey again to stop the recording.

**Note**

**Display of the traversing paths**

- If you switch-off simultaneous recording during machining and then switch-on the function again at a later time, then the traversing paths generated in the intermediate time will not be displayed.
7.5 Program control during simulation

7.5.1 Changing the feedrate

You have the capability of changing the feedrate at any time during the simulation. You track the changes in the status bar.

Note

If you are working with the "Simultaneous recording" function, you use the rotary switch (override) on the control panel.

Procedure

1. Simulation is started.
2. Press the "Program control" softkey.
3. Press the "Override +" or "Override -" softkey to increase or decrease the feedrate by 5%, respectively.
   - OR -
   Press the "100% override" softkey to set the feedrate to 100%.
   - OR -
   Press the "<<" softkey to return to the main screen and perform the simulation with changed feedrate.

Toggling between "Override +" and "Override -"

Simultaneously press the <Ctrl> and <cursor down> or <cursor up> keys to toggle between the "Override +" and "Override -" softkeys.

Selecting the maximum feedrate

Press the <Ctrl> and <M> keys simultaneously to select the maximum feedrate of 120%.
7.5.2 Simulating the program block by block

You have the capability of controlling the program execution during the simulation, i.e. to execute a program, e.g. program block by program block.

Procedure

1. Simulation is started.
2. Press the "Program control" and "Single block" softkeys.
3. Press the "Back" and "Start SBL" softkeys.
   The pending program block is simulated and then stops.
4. Press "Start SBL" as many times as you want to simulate a single program block.
5. Press the "Program control" and the "Single block" softkeys to exist the single block mode.

Switching a single block on and off

Press the <CTRL> and <S> keys simultaneously to enable and disable the single block mode.
7.6 Different views of a workpiece

7.6.1 Overview
The following views are available:
- Side view
- Dressing view
- 3D view (with option)
- Face view - for cylindrical grinding
- Further side views - for surface grinding
- Machine space (with option)

7.6.2 Side view

Displaying a side view

1. Simultaneous recording or simulation is started.
2. Press the "Side view" softkey.
   The side view shows the workpiece in the Z-X plane.

Changing the display
You can enlarge, reduce, pan, and rotate the simulation graphic and the segment.

7.6.3 Dressing view

Displaying the dressing view

1. Simultaneous recording or simulation is started.
2. Press the "Dressing view" softkey.
   In the dressing view, the contour is shown mirrored twice.

Changing the display
You can enlarge, reduce, pan, and rotate the simulation graphic and the segment.
7.6.4 3D view

Displaying the 3D view

1. Simultaneous recording or simulation is started.
2. Press the "Other views" and "3D view" softkeys.

Software option
You require the option "3D simulation (finished part)" for the simulation.

Changing the display
You can enlarge, reduce, pan, and rotate the simulation graphic and the segment.

Displaying and moving cutting planes
You can display and move cutting planes X, Y, and Z.

7.6.5 Face view - for cylindrical grinding

Displaying the face view

1. Simultaneous recording or simulation is started.
2. Press the "Further views" and "Face view" softkeys.

The face view shows the workpiece in the X-Y plane.

Changing the display
You can enlarge, reduce, pan, and rotate the simulation graphic and the segment.

7.6.6 Further side views - for surface grinding

Displaying further side views

1. Simultaneous recording or simulation is started.
2. Press the "Further views" softkey.
3. Press the "From the front" softkey if you wish to view from the front.

- OR -
Press the "From the rear" softkey if you wish to view from the rear.

- OR -
Press the "From the left" softkey if you wish to view from the left.

- OR -
Press the "From the right" softkey if you wish to view from the right.

Changing the display
You can enlarge, reduce, pan, and rotate the simulation graphic and the segment.

7.6.7 Machine space (with "collision avoidance" option)

Displaying the machine space view

1. Simultaneous recording or the simulation is started.

4. Press the "Other views" and "Machine space" softkeys.

During simultaneous recording, an active machine model is displayed.

Changing the display
You have the capability of enlarging, reducing, moving, and rotating the simulation graphic and to change the section.
7.7 Editing the simulation display

7.7.1 Deleting the tool path

The path display follows the programmed tool path of the selected program. The path is continuously updated as a function of the tool movement.

Procedure

1. Simultaneous recording is started.
2. Press the ">>" softkey.
   The tool paths are displayed.
3. Press the "Delete tool path" softkey.
   All tool paths recorded up until now are deleted.
7.8 Editing and adapting a simulation graphic

7.8.1 Enlarging or reducing the graphical representation

Precondition
The simulation or the simultaneous recording is started.

Procedure

1. Press the <+> and <-> keys if you wish to enlarge or reduce the graphic display.
   The graphic display enlarged or reduced from the center.

- OR -
Press the "Details" and "Zoom +" softkeys if you wish to increase the size of the segment.

- OR -
Press the "Details" and "Zoom -" softkeys if you wish to decrease the size of the segment.

- OR -
Press the "Details" and "Auto zoom" softkeys if you wish to automatically adapt the segment to the size of the window.
   The automatic scaling function "Fit to size" takes account of the largest expansion of the workpiece in the individual axes.

Note
Selected section
The selected sections and size changes are kept as long as the program is selected.
7.8.2 Panning a graphical representation

Precondition
The simulation or the simultaneous recording is started.

Procedure
1. Press a cursor key if you wish to move the graphic up, down, left, or right.

7.8.3 Rotating the graphical representation

In the 3D view you can rotate the position of the workpiece to view it from all sides.

Requirement
The simulation or simultaneous recording has been started and the 3D view is selected.

Procedure
1. Press the "Details" softkey.
2. Press the "Rotate view" softkey.

... 

- OR - 

Keep the <Shift> key pressed and then turn the workpiece in the desired direction using the appropriate cursor keys.
7.8.4 Modifying the viewport

If you would like to move, enlarge or decrease the size of the segment of the graphical display, e.g. to view details or display the complete workpiece, use the magnifying glass.

Using the magnifying glass, you can define your own section and then enlarge or reduce its size.

Precondition

The simulation or the simultaneous recording is started.

Procedure

1. Press the "Details" softkey.
2. Press the "Magnifying glass" softkey.
   A magnifying glass in the shape of a rectangular frame appears.
3. Press the "Magnify +" or <+> softkey to enlarge the frame.
   - OR -
   Press the "Magnify -" or <-> softkey to reduce the frame.
   - OR -
   Press one of the cursor keys to move the frame up, down, left or right.
4. Press the "Accept" softkey to accept the selected section.
Simulating machining

7.8 Editing and adapting a simulation graphic
Generating a G-code program

8.1 Graphical programming

Functions

The following functionality is available:

- Context-sensitive online help for every input window
- Support with contour input (geometry processor)

Call and return conditions

- The G functions active before the cycle call and the programmable frame remain active beyond the cycle.
- The starting position must be approached in the higher-level program before the cycle is called. The coordinates are programmed in a clockwise coordinate system.
8.2 Program views

You can display a G code program in various ways.

- Program view
- Parameter screen, either with help screen or graphic view

Note
Help screens / animations

Please note that not all the conceivable kinematics can be displayed in help screens and animations of the cyclic support.

---

**Figure 8-1** Program view of a G code program

### Representation of the machining times

<table>
<thead>
<tr>
<th>Representation</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td>Light green background</td>
<td>Measured machining time of the program block (automatic mode)</td>
</tr>
<tr>
<td>![17.18]</td>
<td></td>
</tr>
<tr>
<td>Green background</td>
<td>Measured machining time of the program group (automatic mode)</td>
</tr>
<tr>
<td>![19.47]</td>
<td></td>
</tr>
<tr>
<td>Representation</td>
<td>Meaning</td>
</tr>
<tr>
<td>------------------------</td>
<td>--------------------------------------------------------------</td>
</tr>
<tr>
<td>Light blue background</td>
<td>Estimated machining time of the program block (simulation)</td>
</tr>
<tr>
<td>17.31</td>
<td></td>
</tr>
<tr>
<td>Blue background</td>
<td>Estimated machining time of the program group (simulation)</td>
</tr>
<tr>
<td>19.57</td>
<td></td>
</tr>
<tr>
<td>Yellow background</td>
<td>Wait time (automatic mode or simulation)</td>
</tr>
<tr>
<td>4.53</td>
<td></td>
</tr>
</tbody>
</table>

See also

Editor settings (Page 172)
8.3 Program structure

G-code programs can always be freely programmed. The most important commands that are normally included are:

- Set a machining plane
- Call a tool (T and D)
- Call a zero offset
- Technology values such as feedrate (F), feedrate type (G94, G95,...), speed and direction of rotation of the spindle (S and M)
- Positions and calls, technology functions (cycles)
- End of program

For G-code programs, before calling cycles, a tool must be selected and the required technology values F, S programmed.

A blank can be specified for simultaneous recording.
8.4 Basic principles

8.4.1 Machining planes

A plane is defined by means of two coordinate axes. The third coordinate axis (tool axis) is perpendicular to this plane and determines the infeed direction of the tool (e.g. for 2½-D machining).

When programming, it is necessary to specify the working plane so that the control system can calculate the tool offset values correctly. The plane is also relevant to certain types of circular programming and polar coordinates.

Working planes

Working planes are defined as follows:

<table>
<thead>
<tr>
<th>Plane</th>
<th>Tool axis</th>
</tr>
</thead>
<tbody>
<tr>
<td>X/Y</td>
<td>G17</td>
</tr>
<tr>
<td>Z/X</td>
<td>G18</td>
</tr>
<tr>
<td>Y/Z</td>
<td>G19</td>
</tr>
</tbody>
</table>

8.4.2 Programming a tool (T)

Calling a tool

1. You are in a part program
2. Press the "Select tool" softkey.
   The “Tool Selection” window opens.
3. Position the cursor on the desired tool and press the "To program" soft-key.
   The selected tool is loaded into the G-code editor. Text such as the following is displayed at the current cursor position in the G-code editor: T="WHEEL100"
4. Press the "Tool list" and "New tool" softkeys.

5. Then select the required tool using the softkeys on the vertical softkey bar, parameterize it and then press the "To program" softkey. The selected tool is loaded into the G-code editor.

6. Then program the tool change (M6), the spindle direction (M3/M4), the spindle speed (S...), the feedrate (F), the feedrate type (G94, G95,...), the coolant (M7/M8) and, if required, further tool-specific functions.
8.5 Generating a G code program

Create a separate program for each new workpiece that you would like to produce. The program contains the individual machining steps that must be performed to produce the workpiece.

Part programs in the G code can be created under the "Workpieces" folder or under the "Part programs" folder.

Procedure

1. Select the "Program Manager" operating area.

2. Select the required archiving location.

Creating a new part program

3. Position the cursor on the folder "Part programs" and press the "New" softkey.
   The "New G Code Program" window opens.

4. Enter the required name and press the "OK" softkey.
   The name can contain up to 28 characters (name + dot + 3-character extension). You can use any letters (except accented), digits or the underscore symbol (_).
   The program type (MPF) is set by default.
   The project is created and opened in the Editor.

Creating a new part program for a workpiece

5. Position the cursor on the folder "Workpieces" and press the "New" softkey.
   The "New G Code Program" window opens.

6. Select the file type (MPF or SPF), enter the desired name of the program and press the "OK" softkey.
   The project is created and opened in the Editor.

7. Enter the desired G code commands.
8.6 Selection of the cycles via softkey

Overview of machining steps

The following softkeys are available to insert machining steps.

- Contour
- New contour
- Contour call

⇒
Programming technology functions

9.1 Programming contours

Function

The free contour programming allows you to create simple or complex contours. You define open or closed contours.

A contour comprises separate contour elements, whereby at least two and up to 250 elements result in a defined contour. Radii, chamfers and tangential transitions are available as contour transition elements.

The integrated contour calculator calculates the intersection points of the individual contour elements taking into account the geometrical relationships, which allows you to enter incompletely dimensioned elements.

You must always program the geometry of the contour before you program the technology.

9.1.1 Representation of the contour

G-code program

In the editor, the contour is represented in a program section using individual program blocks. If you open an individual block, then the contour is opened.

The cycle represents a contour as a program block in the program. If you open this block, the individual contour elements are listed symbolically and displayed in broken-line graphics.

Symbolic representation

The individual contour elements are represented by icons adjacent to the graphics window. They appear in the order in which they were entered.

<table>
<thead>
<tr>
<th>Contour element</th>
<th>Icon</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td>Starting point</td>
<td></td>
<td>Starting point of the contour</td>
</tr>
<tr>
<td>Straight line up</td>
<td></td>
<td>Straight line in 90° grid</td>
</tr>
<tr>
<td>Straight line down</td>
<td></td>
<td>Straight line in 90° grid</td>
</tr>
<tr>
<td>Straight line left</td>
<td></td>
<td>Straight line in 90° grid</td>
</tr>
<tr>
<td>Straight line right</td>
<td></td>
<td>Straight line in 90° grid</td>
</tr>
</tbody>
</table>
### Contour elements

<table>
<thead>
<tr>
<th>Contour element</th>
<th>Icon</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td>Straight line in any direction</td>
<td>![straight_line_icon]</td>
<td>Straight line with any gradient</td>
</tr>
<tr>
<td>Arc right</td>
<td>![arc_right_icon]</td>
<td>Circle</td>
</tr>
<tr>
<td>Arc left</td>
<td>![arc_left_icon]</td>
<td>Circle</td>
</tr>
<tr>
<td>Pole</td>
<td>![pole_icon]</td>
<td>Polar coordinates</td>
</tr>
<tr>
<td>Finish contour</td>
<td>END</td>
<td>End of contour definition</td>
</tr>
</tbody>
</table>

The different colors of the icons indicate their status:

<table>
<thead>
<tr>
<th>Foreground</th>
<th>Background</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td>Black</td>
<td>Blue</td>
<td>Cursor on new element</td>
</tr>
<tr>
<td>Black</td>
<td>Orange</td>
<td>Cursor on current element</td>
</tr>
<tr>
<td>Black</td>
<td>White</td>
<td>Normal element</td>
</tr>
<tr>
<td>Red</td>
<td>White</td>
<td>Element not currently evaluated (element will only be evaluated when it is selected with the cursor)</td>
</tr>
</tbody>
</table>

### Graphical display

The progress of contour programming is shown in broken-line graphics while the contour elements are being entered.

When the contour element has been created, it can be displayed in different line styles and colors:

- Black: Programmed contour
- Orange: Current contour element
- Green dashed: Alternative element
- Blue dotted: Partially defined element

The scaling of the coordinate system is adjusted automatically to match the complete contour. The position of the coordinate system is displayed in the graphics window.

### 9.1.2 Creating a new contour

**Function**

You must create a new contour for each contour that you want to create.
The contours are saved at that position in the program where they are defined.

---

**Note**
Ensure that the contours are positioned after the program end identification!

---

The first step in creating a contour is to specify a starting point. Enter the contour element. The contour processor then automatically defines the end of the contour.

If you alter the machining plane, the cycle will automatically adjust the associated starting point axes. You can enter any additional commands (max. 40 characters) in G-code format for the starting point.

**Additional commands**
You can program feedrates and M commands, for example, using additional G-code commands. You can enter the additional commands (max. 40 characters) in the extended parameter screens ("All parameters" softkey). However, make sure that the additional commands do not collide with the generated G code of the contour. Therefore, do not use any G-code commands of group 1 (G0, G1, G2, G3), no coordinates in the plane and no G-code commands that have to be programmed in a separate block.

**Procedure**

1. The part program to be executed has been created and you are in the editor.
2. Press the "Contour" and "New contour" softkeys. The "New Contour" input window opens.
3. Enter a contour name.
4. Press the "Accept" softkey. The input screen for the starting point of the contour appears. You can enter Cartesian or Polar coordinates.

**Cartesian starting point**

1. Select the machining plane and enter the contour starting point.
2. Enter any additional commands in G-code format, as required.
3. Press the "Accept" softkey.
4. Enter the individual contour elements.
Polar starting point

1. Select the machining plane and press the "Pole" softkey.

2. Enter the pole position in Cartesian coordinates.

3. Enter the starting point for the contour in Polar coordinates.

4. Enter any additional commands in G-code format, as required.

5. Press the "Accept" softkey.

6. Enter the individual contour elements.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Description</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>PL</td>
<td>Machining plane</td>
<td></td>
</tr>
<tr>
<td>X</td>
<td>Cartesian:</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Starting point X (abs)</td>
<td>mm</td>
</tr>
<tr>
<td>Y</td>
<td>Starting point Y (abs)</td>
<td>mm</td>
</tr>
<tr>
<td>X</td>
<td>Polar:</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Position pole (abs)</td>
<td>mm</td>
</tr>
<tr>
<td>Y</td>
<td>Position pole (abs)</td>
<td>Degrees</td>
</tr>
<tr>
<td>L1</td>
<td>Distance to pole, end point (abs)</td>
<td>mm</td>
</tr>
<tr>
<td>$\phi$1</td>
<td>Polar angle to the pole, end point (abs)</td>
<td>Degrees</td>
</tr>
</tbody>
</table>

The contour is finished in continuous-path mode (G64). As a result, contour transitions such as corners, chamfers or radii may not be machined precisely. If you wish to avoid this, then it is possible to use additional commands when programming. Example: For a contour, first program the X parallel straight line and then enter "G9" (non-modal exact stop) for the additional command parameter. Then program the Y-parallel straight line. The corner will be machined exactly, as the feedrate at the end of the X-parallel straight line is briefly zero.

9.1.3 Creating contour elements

After you have created a new contour and specified the starting point, you can define the individual elements that make up the contour.

The following contour elements are available for the definition of a contour:

- Straight vertical line
- Straight horizontal line
- Diagonal line
- Circle/arc
- Pole
For each contour element, you must parameterize a separate parameter screen.

The coordinates for a horizontal or vertical line are entered in Cartesian format; however, for the contour elements Diagonal line and Circle/arc you can choose between Cartesian and Polar coordinates. If you wish to enter Polar coordinates you must first define a pole. If you have already defined a pole for the starting point, you can also refer the Polar coordinates to this pole. Therefore, in this case, you do not have to define an additional pole.

Parameter entry

Parameter entry is supported by various help screens that explain the parameters. If you leave certain fields blank, the geometry processor assumes that the values are unknown and attempts to calculate them from other parameters. Conflicts may result if you enter more parameters than are absolutely necessary for a contour. In such a case, try to enter fewer parameters and allow the geometry processor to calculate as many parameters as possible.

Contour transition elements

As a transition between two contour elements, you can choose a radius or a chamfer. The transition element is always attached at the end of a contour element. The contour transition element is selected in the parameter screen of the respective contour element.

You can use a contour transition element whenever there is an intersection between two successive elements which can be calculated from the input values. Otherwise you must use the straight/circle contour elements.

The contour end is an exception. Although there is no intersection to another element, you can still define a radius or a chamfer as a transition element for the blank.

If the value of the transition element is "NULL", no transition element will be parameterized.

Additional functions

The following additional functions are available for programming a contour:

- Tangent to preceding element
  You can program the transition to the preceding element as tangent.

- Dialog box selection
  If two different possible contours result from the parameters entered previously, select the desired contour.

  Confirm the selection.

- Close contour
  From the current position, you can close the contour with a straight line to the starting point.
9.1.3.1 Entering a contour element

After you have created a new contour and specified the starting point, you can define the individual elements that make up the contour.

Creating contour elements

For each contour element, you must parameterize a separate parameter screen. The coordinates for a horizontal or vertical line are entered in Cartesian format; however, for the contour elements Diagonal line and Circle/arc, you can choose between Cartesian and Polar coordinates.

If you want to enter Polar coordinates, first define a pole. If you have already defined a pole for the starting point, the Polar coordinates can also refer to this pole. You do not require any further pole in this case.

1. Open the part program and create a contour with the "Contour" and "New contour" softkeys.
2. Position the cursor at the desired entry position.
3. Press one of the softkeys to create a contour element.
   - OR -
   - OR -
   - OR -
   - OR -
4. Enter all the data available from the workpiece drawing in the appropriate input screens (e.g. length of straight line, target position, transition to next element, angle of lead, etc.).
5. Press the "Accept" softkey.
   The contour element is added to the contour.
6. When entering data for a contour element, you can program the transition to the preceding element as a tangent.
   Press the "Tangent to prec. elem." softkey. The "tangential" selection appears in the parameter α2 entry field.
7. Repeat the procedure until the contour is complete.
8. Press the "Accept" softkey.
The programmed contour is transferred to the program view.

9. If you want to display further parameters for certain contour elements, e.g. to enter additional commands, press the "All parameters" softkey.

### 9.1.3.2 Cylindrical grinding

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Description</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>X</td>
<td>End point X (abs or inc)</td>
<td>mm</td>
</tr>
<tr>
<td>α1</td>
<td>Starting angle, e.g. to the X axis (only for information)</td>
<td>Degrees</td>
</tr>
<tr>
<td>α2</td>
<td>Angle to the predecessor element (only for information)</td>
<td>Degrees</td>
</tr>
<tr>
<td>Transition to next element</td>
<td>Type of transition</td>
<td></td>
</tr>
<tr>
<td></td>
<td>● Radius</td>
<td></td>
</tr>
<tr>
<td></td>
<td>● Chamfer</td>
<td></td>
</tr>
<tr>
<td>Radius</td>
<td>R Transition to following element - radius</td>
<td>mm</td>
</tr>
<tr>
<td>Chamfer</td>
<td>FS Transition to following element - chamfer</td>
<td>mm</td>
</tr>
<tr>
<td>Additional commands</td>
<td>Additional G-code commands</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Description</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>Z</td>
<td>End point Z (abs or inc)</td>
<td>mm</td>
</tr>
<tr>
<td>α1</td>
<td>Starting angle, e.g. to the Z axis (only for information)</td>
<td>Degrees</td>
</tr>
<tr>
<td>Transition to next element</td>
<td>Type of transition</td>
<td></td>
</tr>
<tr>
<td></td>
<td>● Radius</td>
<td></td>
</tr>
<tr>
<td></td>
<td>● Chamfer</td>
<td></td>
</tr>
<tr>
<td>Radius</td>
<td>R Transition to following element - radius</td>
<td>mm</td>
</tr>
<tr>
<td>Chamfer</td>
<td>FS Transition to following element - chamfer</td>
<td>mm</td>
</tr>
<tr>
<td>Additional commands</td>
<td>Additional G-code commands</td>
<td></td>
</tr>
</tbody>
</table>

### Contour element "Circle"

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Description</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>Direction of rotation</td>
<td>Clockwise direction of rotation</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Counterclockwise direction of rotation</td>
<td></td>
</tr>
<tr>
<td>R</td>
<td>Radius</td>
<td>mm</td>
</tr>
<tr>
<td>e.g. X</td>
<td>End point X (abs or inc)</td>
<td>mm</td>
</tr>
<tr>
<td>e.g. Z</td>
<td>End point Z (abs or inc)</td>
<td>mm</td>
</tr>
<tr>
<td>e.g. I</td>
<td>Circle center point I (abs or inc)</td>
<td>mm</td>
</tr>
<tr>
<td>e.g. J</td>
<td>Circle center point J (abs or inc)</td>
<td>mm</td>
</tr>
</tbody>
</table>
### Programming technology functions

#### 9.1 Programming contours

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Description</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>α1</td>
<td>Starting angle to X axis</td>
<td>Degrees</td>
</tr>
<tr>
<td>α2</td>
<td>Angle to the preceding element</td>
<td>Degrees</td>
</tr>
<tr>
<td>β1</td>
<td>End angle to Z axis</td>
<td>Degrees</td>
</tr>
<tr>
<td>β2</td>
<td>Opening angle</td>
<td>Degrees</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Transition to next element</th>
<th>Type of transition</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Radius</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Chamfer</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Radius</th>
<th>R</th>
<th>Transition to following element - radius</th>
<th>mm</th>
</tr>
</thead>
<tbody>
<tr>
<td>Chamfer</td>
<td>FS</td>
<td>Transition to following element - chamfer</td>
<td>mm</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Additional commands</th>
<th>Additional G-code commands</th>
</tr>
</thead>
</table>

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Description</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>X</td>
<td>End point X (abs or inc)</td>
<td>mm</td>
</tr>
<tr>
<td>Z</td>
<td>End point Z (abs or inc)</td>
<td>mm</td>
</tr>
<tr>
<td>L</td>
<td>Length</td>
<td>mm</td>
</tr>
<tr>
<td>α1</td>
<td>Starting angle, e.g. to the X axis</td>
<td>Degrees</td>
</tr>
<tr>
<td>α2</td>
<td>Angle to the preceding element</td>
<td>Degrees</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Transition to next element</th>
<th>Type of transition</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Radius</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Chamfer</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Radius</th>
<th>R</th>
<th>Transition to following element - radius</th>
<th>mm</th>
</tr>
</thead>
<tbody>
<tr>
<td>Chamfer</td>
<td>FS</td>
<td>Transition to following element - chamfer</td>
<td>mm</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Additional commands</th>
<th>Additional G-code commands</th>
</tr>
</thead>
</table>

### Contour element "Pole"

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Description</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>X</td>
<td>Position pole (abs)</td>
<td>mm</td>
</tr>
<tr>
<td>Z</td>
<td>Position pole (abs)</td>
<td>mm</td>
</tr>
</tbody>
</table>

### Contour element "End"

The data for the transition at the contour end of the previous contour element is displayed in the "End" parameter screen.

The values cannot be edited.

#### 9.1.3.3 Surface grinding

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Description</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>Z</td>
<td>End point Z (abs or inc)</td>
<td>mm</td>
</tr>
<tr>
<td>α1</td>
<td>Starting angle to Z axis (only for information)</td>
<td>Degrees</td>
</tr>
</tbody>
</table>

Grinding
Operating Manual, 06/2019, A5E44903521B AB
### Programming technology functions

#### 9.1 Programming contours

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Description</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>( \alpha_2 )</td>
<td>Angle to the preceding element</td>
<td>Degrees</td>
</tr>
<tr>
<td>Transition to next element</td>
<td>Type of transition</td>
<td></td>
</tr>
<tr>
<td>● Radius</td>
<td></td>
<td></td>
</tr>
<tr>
<td>● Undercut</td>
<td></td>
<td></td>
</tr>
<tr>
<td>● Chamfer</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Radius</td>
<td>( R )</td>
<td>Transition to following element - radius</td>
</tr>
<tr>
<td>Undercut</td>
<td>Form E</td>
<td>Undercut size e.g. E1.0x0.4</td>
</tr>
<tr>
<td></td>
<td>Form F</td>
<td>Undercut size e.g. F0.6x0.3</td>
</tr>
<tr>
<td>DIN thread</td>
<td>( P )</td>
<td>Thread pitch</td>
</tr>
<tr>
<td></td>
<td>( \alpha )</td>
<td>Insertion angle</td>
</tr>
<tr>
<td></td>
<td>Thread</td>
<td>( Z_1 )</td>
</tr>
<tr>
<td></td>
<td></td>
<td>( Z_2 )</td>
</tr>
<tr>
<td></td>
<td></td>
<td>( R_1 )</td>
</tr>
<tr>
<td></td>
<td></td>
<td>( R_2 )</td>
</tr>
<tr>
<td></td>
<td></td>
<td>( T )</td>
</tr>
<tr>
<td>Chamfer</td>
<td>FS</td>
<td>Transition to following element - chamfer</td>
</tr>
<tr>
<td>CA</td>
<td>Grinding allowance</td>
<td>mm</td>
</tr>
<tr>
<td></td>
<td>● ( \uparrow )</td>
<td>Grinding allowance to right of contour</td>
</tr>
<tr>
<td></td>
<td>● ( \downarrow )</td>
<td>Grinding allowance to left of contour</td>
</tr>
<tr>
<td>Additional commands</td>
<td>Additional G-code commands</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Description</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>( \alpha_1 )</td>
<td>Starting angle, e.g. to the Y axis (only for information)</td>
<td>Degrees</td>
</tr>
<tr>
<td>( \alpha_2 )</td>
<td>Angle to the preceding element</td>
<td>Degrees</td>
</tr>
<tr>
<td>Transition to next element</td>
<td>Type of transition</td>
<td></td>
</tr>
<tr>
<td>● Radius</td>
<td></td>
<td></td>
</tr>
<tr>
<td>● Undercut</td>
<td></td>
<td></td>
</tr>
<tr>
<td>● Chamfer</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Radius</td>
<td>( R )</td>
<td>Transition to following element - radius</td>
</tr>
<tr>
<td>Undercut</td>
<td>Form E</td>
<td>Undercut size e.g. E1.0x0.4</td>
</tr>
<tr>
<td></td>
<td>Form F</td>
<td>Undercut size e.g. F0.6x0.3</td>
</tr>
<tr>
<td>DIN thread</td>
<td>( P )</td>
<td>Thread pitch</td>
</tr>
<tr>
<td></td>
<td>( \alpha )</td>
<td>Insertion angle</td>
</tr>
<tr>
<td></td>
<td>Thread</td>
<td>( Z_1 )</td>
</tr>
<tr>
<td></td>
<td></td>
<td>( Z_2 )</td>
</tr>
<tr>
<td></td>
<td></td>
<td>( R_1 )</td>
</tr>
<tr>
<td></td>
<td></td>
<td>( R_2 )</td>
</tr>
<tr>
<td></td>
<td></td>
<td>( T )</td>
</tr>
<tr>
<td>Chamfer</td>
<td>FS</td>
<td>Transition to following element - chamfer</td>
</tr>
</tbody>
</table>
## Programming technology functions

### 9.1 Programming contours

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Description</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>CA</td>
<td>Grinding allowance</td>
<td>mm</td>
</tr>
<tr>
<td></td>
<td>● Grinding allowance to right of contour</td>
<td></td>
</tr>
<tr>
<td></td>
<td>● Grinding allowance to left of contour</td>
<td></td>
</tr>
<tr>
<td>Additional commands</td>
<td>Additional G-code commands</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Description</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>Z</td>
<td>End point Z (abs or inc)</td>
<td>mm</td>
</tr>
<tr>
<td>Y</td>
<td>End point Z ⊘ (abs or end point Z (inc))</td>
<td>mm</td>
</tr>
<tr>
<td>α1</td>
<td>Starting angle, e.g. to the Z axis (only for information)</td>
<td>Degrees</td>
</tr>
<tr>
<td>α2</td>
<td>Angle to the preceding element</td>
<td>Degrees</td>
</tr>
<tr>
<td>Transition to next element</td>
<td>Type of transition</td>
<td></td>
</tr>
<tr>
<td></td>
<td>● Radius</td>
<td></td>
</tr>
<tr>
<td></td>
<td>● Chamfer</td>
<td></td>
</tr>
<tr>
<td>Radius</td>
<td>R</td>
<td>Transition to following element - radius</td>
</tr>
<tr>
<td>Chamfer</td>
<td>FS</td>
<td>Transition to following element - chamfer</td>
</tr>
<tr>
<td>CA</td>
<td>Grinding allowance</td>
<td>mm</td>
</tr>
<tr>
<td></td>
<td>● Grinding allowance to right of contour</td>
<td></td>
</tr>
<tr>
<td></td>
<td>● Grinding allowance to left of contour</td>
<td></td>
</tr>
<tr>
<td>Additional commands</td>
<td>Additional G-code commands</td>
<td></td>
</tr>
</tbody>
</table>

### Contour element "Circle"

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Description</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>Direction of rotation</td>
<td>● Clockwise direction of rotation</td>
<td></td>
</tr>
<tr>
<td></td>
<td>● Counterclockwise direction of rotation</td>
<td></td>
</tr>
<tr>
<td>Z</td>
<td>End point Z (abs or inc)</td>
<td>mm</td>
</tr>
<tr>
<td>X</td>
<td>End point Y ⊘ (abs or end point Y (inc))</td>
<td>mm</td>
</tr>
<tr>
<td>K</td>
<td>Circle center point K (abs or inc)</td>
<td>mm</td>
</tr>
<tr>
<td>I</td>
<td>Circle center point I ⊘ (abs or circle center point I (inc))</td>
<td>mm</td>
</tr>
<tr>
<td>α1</td>
<td>Starting angle to the Z axis (only for information)</td>
<td>Degrees</td>
</tr>
<tr>
<td>β1</td>
<td>End angle to the Z axis (only for information)</td>
<td>Degrees</td>
</tr>
<tr>
<td>β2</td>
<td>Opening angle</td>
<td>Degrees</td>
</tr>
<tr>
<td>Transition to next element</td>
<td>Type of transition</td>
<td></td>
</tr>
<tr>
<td></td>
<td>● Radius</td>
<td></td>
</tr>
<tr>
<td></td>
<td>● Chamfer</td>
<td></td>
</tr>
<tr>
<td>Radius</td>
<td>R</td>
<td>Transition to following element - radius</td>
</tr>
<tr>
<td>Chamfer</td>
<td>FS</td>
<td>Transition to following element - chamfer</td>
</tr>
</tbody>
</table>
### Contour element "Pole"

<table>
<thead>
<tr>
<th>Parameters</th>
<th>Description</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>Z</td>
<td>Position pole (abs)</td>
<td>mm</td>
</tr>
<tr>
<td>Y</td>
<td>Position pole (abs)</td>
<td>Degrees</td>
</tr>
</tbody>
</table>

### Contour element "End"

The data for the transition at the contour end of the previous contour element is displayed in the "End" parameter screen.

The values cannot be edited.

### 9.1.4 Changing the contour

#### 9.1.4.1 Overview

**Function**

You can change a previously created contour later.

Individual contour elements can be

- added,
- changed,
- inserted or
- deleted.

#### 9.1.4.2 Modifying a contour element

**Procedure for changing a contour element**

1. Open the part program to be machined.
2. Open the contour.
3. With the cursor, select the program block where you want to change the contour. Open the geometry processor. The individual contour elements are listed.

4. Position the cursor at the position where a contour element is to be inserted or changed.

5. Select the desired contour element with the cursor.

6. Enter the parameters in the input screen or delete the element and select a new element.

7. Press the "Accept" softkey. The desired contour element is inserted in the contour or changed.

Procedure for deleting a contour element

1. Open the part program to be machined.

2. Open the contour.

3. Position the cursor on the contour element that you want to delete.

4. Press the "Delete element" softkey.

5. Press the "Delete" softkey.

Note

Ensure that the complete contour and the transition to the following element remain.

9.1.5 Contour call (CYCLE62)

9.1.5.1 Function

Function

The input creates a reference to the selected contour.

There are four ways to call the contour:

1. Contour name
   The contour is in the calling main program.

2. Labels
   The contour is in the calling main program and is limited by the labels that have been entered.
3. Subprogram
   The contour is located in a subprogram in the same workpiece.

4. Labels in the subprogram
   The contour is in a subprogram and is limited by the labels that have been entered.

9.1.5.2 Calling the cycle

Procedure

1. The part program to be executed has been created and you are in the editor.

2. Press the "Contour" and "Contour call" softkeys.
   The "Contour Call" input window opens.

3. Assign parameters to the contour selection.

9.1.5.3 Parameter

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Description</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>Contour selection</td>
<td>• Contour name</td>
<td></td>
</tr>
<tr>
<td></td>
<td>• Labels</td>
<td></td>
</tr>
<tr>
<td></td>
<td>• Subprogram</td>
<td></td>
</tr>
<tr>
<td></td>
<td>• Labels in the subprogram</td>
<td></td>
</tr>
<tr>
<td>Contour name</td>
<td>CON: Contour name</td>
<td></td>
</tr>
<tr>
<td>Labels</td>
<td>• LAB1: Label 1</td>
<td></td>
</tr>
<tr>
<td></td>
<td>• LAB2: Label 2</td>
<td></td>
</tr>
<tr>
<td>Subprogram</td>
<td>PRG: Subprogram</td>
<td></td>
</tr>
<tr>
<td>Labels in the subprogram</td>
<td>• PRG: Subprogram</td>
<td></td>
</tr>
<tr>
<td></td>
<td>• LAB1: Label 1</td>
<td></td>
</tr>
<tr>
<td></td>
<td>• LAB2: Label 2</td>
<td></td>
</tr>
</tbody>
</table>

Note
Calling via EXTCALL

Calling a part program via EXTCALL without EES: via "contour name" or "labels." This behavior is monitored in the cycle.

Contour calls via "subprogram" or "labels in subprogram" are only possible if EES is active.
9.2 Set dresser coordinate system - CYCLE435

9.2.1 Note regarding the dressing tool position

The cycle to calculate the dressing tool position is not programmed using an input screen form in SINUMERIK Operate.

Syntax

\[
\text{CYCLE435}(<_T>, <_DD>, <S_TA>, <S_DA>, <S_AD>, <S_AL>, <S_PVD>, <S_PVL>, <S_PD>, <S_PL>, <_AMODE>)
\]

Parameter

<table>
<thead>
<tr>
<th>No.</th>
<th>Parameter mask</th>
<th>Parameter internal</th>
<th>Data type</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>&lt;_T&gt;</td>
<td>STRING[32]</td>
<td>Tool name of the grinding wheel</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>&lt;_DD&gt;</td>
<td>INT</td>
<td>Cutting edge number of the grinding wheel</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>&lt;S_TA&gt;</td>
<td>STRING[32]</td>
<td>Dressing tool reference point - dressing tool name</td>
<td></td>
</tr>
<tr>
<td>4</td>
<td>&lt;S_DA&gt;</td>
<td>INT</td>
<td>Cutting edge number of the dressing tool</td>
<td></td>
</tr>
<tr>
<td>5</td>
<td>&lt;S_AD&gt;</td>
<td>REAL</td>
<td>Dressing value, diameter</td>
<td></td>
</tr>
<tr>
<td>6</td>
<td>&lt;S_AL&gt;</td>
<td>REAL</td>
<td>Dressing value, face</td>
<td></td>
</tr>
<tr>
<td>7</td>
<td>&lt;S_PVD&gt;</td>
<td>REAL</td>
<td>Form-truing offset, diameter</td>
<td></td>
</tr>
<tr>
<td>8</td>
<td>&lt;S_PVL&gt;</td>
<td>REAL</td>
<td>Form-truing offset, face</td>
<td></td>
</tr>
<tr>
<td>9</td>
<td>&lt;S_PD&gt;</td>
<td>REAL</td>
<td>Form-truing allowance, diameter</td>
<td></td>
</tr>
<tr>
<td>10</td>
<td>&lt;S_PL&gt;</td>
<td>REAL</td>
<td>Form-truing allowance, face</td>
<td></td>
</tr>
<tr>
<td>11</td>
<td>&lt;_AMODE&gt;</td>
<td>INT</td>
<td>Alternative mode</td>
<td></td>
</tr>
</tbody>
</table>

UNITS:

<table>
<thead>
<tr>
<th>active tool at the end of the cycle</th>
</tr>
</thead>
<tbody>
<tr>
<td>0 = dressing tool active</td>
</tr>
<tr>
<td>1 = wheel active</td>
</tr>
</tbody>
</table>

9.2.2 Function

The cycle is used to activate a coordinate system for dressing. A decision can be made as to whether, after the cycle has been called, the transferred dressing tool or the transferred grinding tool is active. The dimension offsets are already taken into account in the cycle so that the required form-truing contour can be machined.

After the dressing contour has been machined, the activated coordinate system must be cleared again by calling the cycle without transfer parameter.
Sequence

In the cycle, the tool data of the tool that is not active is transferred into the cycle frame. This means that the dressing contour can be subsequently called relative to the geometry parameters.

After machining the dressing contour, the cycle frame is cleared by calling the cycle without transfer parameter.

Example

```
T="WHEEL"D1
CYCLE435(„WHEEL“,1,“DRESSER“,1,0.01,0.01,10,10,0,0,0)
G01 G64 F200
X=10
Z=10
...
;Machining the dressing contour
...
CYCLE435()
```

In the example, after the cycle, the "DRESSER" tool with cutting edge 1 is active.

Internally, an offset of 0.01 mm is taken into account in X and Z (for G18) and the contour itself is offset by 10 mm, so that workpiece drawing dimensions can be programmed. A form-truing allowance that can be taken into account is 0. After this, the dressing contour can be machined.

After machining, the cycle frame is cleared by calling CYCLE435() without transfer parameter.
9.3 Form-truing (CYCLE495)

The form-truing cycle is not programmed using an input screen form in SINUMERIK Operate.

Software option
For the function "Form-truing type: parallel to the axis" to be used, you need the "SINUMERIK Grinding Advanced" software option.

Syntax

```
CYCLE495(<_T>, <_DD>, <_SC>, <_F>, <_VARI>, <_D>, <_DX>, <_DZ>,
<S_PA>, <S_N>, <_DMODE>, <_AMODE>, <S_FW>, <S_HW>)
```

Parameter

<table>
<thead>
<tr>
<th>No.</th>
<th>Parameter mask</th>
<th>Parameter internal</th>
<th>Data type</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>&lt;_T&gt;</td>
<td>STRING[20]</td>
<td></td>
<td>Tool name of the grinding wheel</td>
</tr>
<tr>
<td>2</td>
<td>&lt;_DD&gt;</td>
<td>INT</td>
<td></td>
<td>Cutting edge number of the grinding wheel</td>
</tr>
<tr>
<td>3</td>
<td>&lt;_SC&gt;</td>
<td>REAL</td>
<td></td>
<td>Lift-off distance for avoiding obstacles, incremental</td>
</tr>
<tr>
<td>4</td>
<td>&lt;_F&gt;</td>
<td>REAL</td>
<td></td>
<td>Form-truing feedrate</td>
</tr>
<tr>
<td>5</td>
<td>&lt;_VARI&gt;</td>
<td>INT</td>
<td></td>
<td>Machining type</td>
</tr>
</tbody>
</table>

UNITs:

- **Form-truing type**
  - 1 = Parallel to the axis
  - 2 = Parallel to the contour

TENS:

- **Machining direction**
  - 0 = Pulling
    - Possible with cutting edge positions 1 to 4
  - 1 = Pushing
    - Possible with cutting edge positions 1 to 4
  - 2 = Alternating
    - Possible with cutting edge positions 1 to 8
  - 3 = Start → end
    - Possible with cutting edge positions 1 to 8
  - 4 = End → start
    - Possible with cutting edge positions 1 to 8

HUNDREDS:

- **Infeed direction**
  - 1 = Infeed X for G18 or Y- for G19
  - 2 = Infeed X+ for G18 or Y+ for G19
  - 3 = Infeed Z- for G18 and for G19
  - 4 = Infeed Z+ for G18 and for G19
<table>
<thead>
<tr>
<th>No.</th>
<th>Parameter mask</th>
<th>Parameter internal</th>
<th>Data type</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td>6</td>
<td>&lt;_D&gt;</td>
<td></td>
<td>REAL</td>
<td>Dressing value for form-truing type parallel to the axis</td>
</tr>
<tr>
<td>7</td>
<td>&lt;_DX&gt;</td>
<td></td>
<td>REAL</td>
<td>Dressing value X for G18 or Y for G19 for form-truing type parallel to the contour</td>
</tr>
<tr>
<td>8</td>
<td>&lt;_DZ&gt;</td>
<td></td>
<td>REAL</td>
<td>Dressing value Z for G18 and G19 for form-truing type parallel to the contour</td>
</tr>
<tr>
<td>9</td>
<td>&lt;S_PA&gt;</td>
<td></td>
<td>REAL</td>
<td>Form-truing allowance</td>
</tr>
<tr>
<td>10</td>
<td>&lt;S_N&gt;</td>
<td>INT</td>
<td>Number of strokes in the form-truing program</td>
<td></td>
</tr>
<tr>
<td>11</td>
<td>&lt;_DMODE&gt;</td>
<td>INT</td>
<td>Display mode</td>
<td></td>
</tr>
</tbody>
</table>

**UNITS:**

<table>
<thead>
<tr>
<th></th>
<th>Machining plane G17/G18/G19</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>Compatibility, the plane effective before the cycle call remains active</td>
</tr>
<tr>
<td>1</td>
<td>G17 (only active in the cycle)</td>
</tr>
<tr>
<td>2</td>
<td>G18 (only active in the cycle)</td>
</tr>
<tr>
<td>3</td>
<td>G19 (only active in the cycle)</td>
</tr>
</tbody>
</table>

| 12  | <_AMODE>      | INT               | Alternative mode |

**UNITS:**

<table>
<thead>
<tr>
<th></th>
<th>Form-truing selection, new/continue</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>New</td>
</tr>
<tr>
<td>2</td>
<td>Continue</td>
</tr>
</tbody>
</table>

**TENS:**

<table>
<thead>
<tr>
<th></th>
<th>Select form-truing allowance</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>From the rough contour to the lowest point of the contour</td>
</tr>
<tr>
<td>1</td>
<td>From the rough contour to the highest point of the contour</td>
</tr>
</tbody>
</table>

| 13  | <S_FW>      | REAL               | Clear angle of the dressing tool |
| 14  | <S_HW>      | REAL               | Holder angle of the dressing tool |

Parameter S_N specifies how many strokes are generated in a form-truing program.

**Note**

**Machining direction**

For "alternating" machining direction, parameter S_N should be set to 1, so that the direction changes at each stroke. Otherwise, use values >1, to achieve shorter machining times.

In addition, the following setting data are used:

SD55880 $SCS_GRIND_CONT_RELEASE_ANGLE
SD55881 $SCS_GRIND_CONT_RELEASE_DIST
SD55884 $SCS_GRIND_CONT_BLANK_OFFSET
References

For additional information, please refer to the following documentation:

- SINUMERIK Operate (IM9) / SINUMERIK 840D sl Commissioning Manual
- SINUMERIK Operate (IH9) / SINUMERIK 828D Commissioning Manual

Function

With the cycle, you form-true grinding wheels using a dressing tool. A dressing tool must be selected as active tool.

The OEM or the end user is responsible for the management of this.

The cycle runs in the active coordinate system, i.e. the cycle does not change any tool offsets or work offsets.

You program the grinding wheel contour as G code ("free contour"). This can be located in a subroutine or in the main program between two labels. The contour is transferred to the form-truing cycle CYCLE495 using contour call cycle CYCLE62. Form-truing is carried out until the form-truing allowance has been machined.

You have the option of form-truing either parallel with the axis or parallel with the contour:

- For form-truing parallel with the contour, the contour is moved along with each stroke. Taking into account the unmachined part (blank), contour segments can be omitted while machining.
- For form-truing in parallel to the axis, the form is established through longitudinal cuts parallel to the machining axis.

Form-truing can be interrupted, and continued again at the point of interruption. However, between stopping and continuing, no additional form-truing may be started, no wheel changed and no power on carried out.

The actual form-truing allowance is saved to the channel-specific GUD variables S_GC_CONT_R[2] for diagnostic purposes/progress display and for continuing after an interruption.

Sequence

Main program:

- Selection of the dressing tool coordinate system
  - with CYCLE435 (Page 248)(,,,) or
  - with OEM or end user cycle
    As a consequence, a dressing tool must be selected as active tool.

- Prepositioning, dressing tool
  The position of the dressing tool should be selected so that the tool can approach the grinding wheel for form-truing without any risk of collision.

- Contour call with CYCLE62(,,,) (Page 246)
  With the contour call, the form of the grinding wheel is transferred to the form-truing cycle.
• Form-truing with CYCLE495(,,)
  The grinding wheel form is trued using CYCLE495.

• Deselection of the dressing tool coordinate system
  – with CYCLE435 (Page 248)(,,) or
  – with OEM or end user cycle

Example

;*Main program
T="WHEEL" D1
CYCLE435("WHEEL",1,"DRESSER_6",1,0,0,10,10,0,0,0)
G0 X10
Z-40
CYCLE62("CONTOUR",0,,)
CYCLE495("WHEEL",1,.5,100,131,0.01,,0.345,100,0,11,90,85)
CYCLE435()
M30

;*
N10 G00 G90
N20 G01 X=2.5 Z=-37 F=100
N30 Z=-23.03906 F=100
N40 G03 X=0 Z=-23 CR=400 RND=0 F=50
N50 G03 X=0 Z=-3 CR=50 F=100
N60 G01 X=0 Z=-2.8
N70 Z=-0.2
N80 X=-1 F=50
N90 Z=2 F=100
N100 M17
9.4 Oscillating cycles (CYCLE4071 ... CYCLE4079)

9.4.1 Note regarding oscillating cycles

Oscillating cycles are not programmed using input screen forms in SINUMERIK Operate.

9.4.2 CYCLE4071 - longitudinal grinding with infeed at the reversal point

Syntax

\[ \text{CYCLE4071(<S_A>, <S_B>, <S_W>, <S_U>, <S_I>, <S_K>, <S_H>, <S_A1>, <S_A2>)} \]

Parameters

<table>
<thead>
<tr>
<th>No.</th>
<th>Parameter</th>
<th>Data type</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>&lt;S_A&gt;</td>
<td>REAL</td>
<td>Infeed depth at the start</td>
</tr>
<tr>
<td>2</td>
<td>&lt;S_B&gt;</td>
<td>REAL</td>
<td>Infeed depth at the end</td>
</tr>
<tr>
<td>3</td>
<td>&lt;S_W&gt;</td>
<td>REAL</td>
<td>Grinding width</td>
</tr>
<tr>
<td>4</td>
<td>&lt;S_U&gt;</td>
<td>REAL</td>
<td>Sparking-out time</td>
</tr>
<tr>
<td>5</td>
<td>&lt;S_I&gt;</td>
<td>REAL</td>
<td>Feedrate for infeed</td>
</tr>
<tr>
<td>6</td>
<td>&lt;S_K&gt;</td>
<td>REAL</td>
<td>Feedrate for transverse infeed</td>
</tr>
<tr>
<td>7</td>
<td>&lt;S_H&gt;</td>
<td>INT</td>
<td>Number of repetitions</td>
</tr>
<tr>
<td>8</td>
<td>&lt;S_A1&gt;</td>
<td>AXIS</td>
<td>Infeed axis (optional) or 1st geometry axis</td>
</tr>
<tr>
<td>9</td>
<td>&lt;S_A2&gt;</td>
<td>AXIS</td>
<td>Oscillating axis (optional) or 2nd geometry axis</td>
</tr>
</tbody>
</table>

Function

The cycle is used for the execution of repeating infeeds. The infeed depth at the start and at the end can be different. There is a tangential motion between the infeeds.
**Sequence**

1. Start of the cycle at the current position of the oscillating axis.
2. Traversing of the infeed axis to the infeed depth at the start \(<S_A>\) with the feedrate for infeed \(<S_I>\).
3. Sparking out with the sparking-out time \(<S_U>\).
4. Traversing of the oscillating axis with the grinding width \(<S_W>\) as travel path and the feedrate for transverse infeed \(<S_K>\).
5. Traversing of the infeed axis to the infeed depth at the end \(<S_B>\) with the feedrate for infeed \(<S_I>\).
6. Sparking out with the sparking-out time \(<S_U>\).
7. Traversing of the oscillating axis with the grinding width \(<S_W>\) as travel path to the starting point and the feedrate for transverse infeed \(<S_K>\).

Indicates reiterating sequential steps.

The sequence is repeated until the programmed number of repetitions \(<S_H>\) has been reached.

---

**Note**

The sequence cannot be interrupted with a single block.

---

**Example**

Executing two oscillating motions with the following cycle parameters:

- Infeed depth at the start: 0.02 mm
- Infeed depth at the end: 0.01 mm
- Stroke: 100 mm
- Sparking-out time: 1 s
- Infeed feedrate: 1 mm/min
- Transverse feedrate: 1000 mm/min
- Repetitions: 2
- Oscillating and infeed axes: Standard geometry axes

**Program code**

```
N10 T1 D1
```
9.4 Oscillating cycles (CYCLE4071 ... CYCLE4079)

Program code

N20 CYCLE4071(0.02,0.01,100,1,1,1000,2)
N30 M30

9.4.3 CYCLE4072 - longitudinal grinding with infeed at the reversal point and cancel signal

Syntax

CYCLE4072(<S_GAUGE>, <S_A>, <S_B>, <S_W>, <S_U>, <S_I>, <S_K>, <S_H>, <S_A1>, <S_A2>)

Parameters

<table>
<thead>
<tr>
<th>No.</th>
<th>Parameter</th>
<th>Data type</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>&lt;S_GAUGE&gt;</td>
<td>STRING</td>
<td>Cancel conditions for infeed: 1. Number of a rapid input 2. Logical expression</td>
</tr>
<tr>
<td>2</td>
<td>&lt;S_A&gt;</td>
<td>REAL</td>
<td>Infeed depth at the start</td>
</tr>
<tr>
<td>3</td>
<td>&lt;S_B&gt;</td>
<td>REAL</td>
<td>Infeed depth at the end</td>
</tr>
<tr>
<td>4</td>
<td>&lt;S_W&gt;</td>
<td>REAL</td>
<td>Grinding width</td>
</tr>
<tr>
<td>5</td>
<td>&lt;S_U&gt;</td>
<td>REAL</td>
<td>Sparking-out time</td>
</tr>
<tr>
<td>6</td>
<td>&lt;S_I&gt;</td>
<td>REAL</td>
<td>Feedrate for infeed</td>
</tr>
<tr>
<td>7</td>
<td>&lt;S_K&gt;</td>
<td>REAL</td>
<td>Feedrate for transverse infeed</td>
</tr>
<tr>
<td>8</td>
<td>&lt;S_H&gt;</td>
<td>INT</td>
<td>Number of repetitions</td>
</tr>
<tr>
<td>9</td>
<td>&lt;S_A1&gt;</td>
<td>AXIS</td>
<td>Infeed axis (optional) or 1st geometry axis</td>
</tr>
<tr>
<td>10</td>
<td>&lt;S_A2&gt;</td>
<td>AXIS</td>
<td>Oscillating axis (optional) or 2nd geometry axis</td>
</tr>
</tbody>
</table>

Function

The cycle is used for the execution of repeating infeeds taking into account an external cancel signal. The infeed depth can be different at the start and at the end. There is a tangential motion between the infeeds. The depth infeed is cancelled when the cancel condition is satisfied. A complete stroke is always performed after the cancellation of the depth infeed.
Sequence

Cancellation of the infeed at the end

[Diagram showing the sequence of operations with labeled steps (1 to 9)]
Cancellation of the infeed at the start

① Start of the cycle at the current position of the oscillating axis.
② Traversing of the infeed axis to the infeed depth at the start $<S_A>$ with the feedrate for infeed $<S_I>$.
③ Sparking out with the sparking-out time $<S_U>$.
④ Traversing of the oscillating axis with the grinding width $<S_W>$ as travel path and the feedrate for transverse infeed $<S_K>$.
⑤ Traversing of the infeed axis to the infeed depth at the end $<S_B>$ with the feedrate for infeed $<S_I>$.
⑥ Sparking out with the sparking-out time $<S_U>$.
⑦ Traversing of the oscillating axis with the grinding width $<S_W>$ as travel path to the starting point and the feedrate for transverse infeed $<S_K>$.
⑧ Cancel signal: The machining stops when the next start point is reached.
⑨ Without Cancel signal: The sequence is repeated until the programmed number of repetitions $<S_H>$ has been reached.

Indicates reiterating sequential steps.

Note
The sequence cannot be interrupted with a single block.

Resources
As resources, the cycle uses a block-wide synchronized action and a synchronized action variable. The synchronized action is determined dynamically from the free area of the synchronized action range (CUS.DIR - 1 ..., CMA.DIR - 1000 ..., CST.DIR – 1199 ...). SYG_IS[1] is used as the synchronized action variable.
Examples

Example 1: Oscillation with two strokes:
Cycle parameters
- Infeed depth at the start: 0.02 mm
- Infeed depth at the end: 0.01 mm
- Stroke: 100 mm
- Sparking-out time: 1 s
- Infeed feedrate: 1 mm/min
- Transverse feedrate: 1000 mm/min
- Repetitions: 2
- Oscillating and infeed axes: Standard geometry axes
Cancel signal: Rapid input 1 ($A_IN[1]$)

Program code
N10 T1 D1
N20 CYCLE4072("1",0.02,0.01,100,1,1,1000,2)
N30 M30

Example 2: Oscillation with two strokes:
Cycle parameters
- Infeed depth at the start: 0.02 mm
- Infeed depth at the end: 0.01 mm
- Stroke: 100 mm
- Sparking-out time: 1 s
- Infeed feedrate: 1 mm/min
- Transverse feedrate: 1000 mm/min
- Repetitions: 2
- Oscillating and infeed axes: Standard geometry axes
Cancel signal: Variable $A_DBR[20] < 0.01$

Program code
N10 T1 D1
N20 CYCLE4072("($A_DBR[20]<0.01)",0.02,0.01,100,1,1,1000,2)
N30 M30
9.4.4 CYCLE4073 - longitudinal grinding with continuous infeed

Syntax

CYCLE4073(<S_A>, <S_B>, <S_W>, <S_U>, <S_K>, <S_H>, <S_A1>, <S_A2>)

Parameters

<table>
<thead>
<tr>
<th>No.</th>
<th>Parameter</th>
<th>Data type</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>&lt;S_A&gt;</td>
<td>REAL</td>
<td>Infeed depth at the start</td>
</tr>
<tr>
<td>2</td>
<td>&lt;S_B&gt;</td>
<td>REAL</td>
<td>Infeed depth at the end</td>
</tr>
<tr>
<td>3</td>
<td>&lt;S_W&gt;</td>
<td>REAL</td>
<td>Grinding width</td>
</tr>
<tr>
<td>4</td>
<td>&lt;S_U&gt;</td>
<td>REAL</td>
<td>Sparking-out time</td>
</tr>
<tr>
<td>5</td>
<td>&lt;S_K&gt;</td>
<td>REAL</td>
<td>Feedrate for transverse infeed</td>
</tr>
<tr>
<td>6</td>
<td>&lt;S_H&gt;</td>
<td>INT</td>
<td>Number of repetitions</td>
</tr>
<tr>
<td>7</td>
<td>&lt;S_A1&gt;</td>
<td>AXIS</td>
<td>Infeed axis (optional) or 1st geometry axis</td>
</tr>
<tr>
<td>8</td>
<td>&lt;S_A2&gt;</td>
<td>AXIS</td>
<td>Oscillating axis (optional) or 2nd geometry axis</td>
</tr>
</tbody>
</table>

Function

The cycle is used for the execution of repeating infeeds. The infeed from the start to the end and from the end to the start can be different.

Sequence

1. Start of the cycle at the current position of the oscillating axis with infeed depth 0.
2. Traversing of the oscillating axis with the grinding width <S_W> as travel path and feedrate for transverse infeed <S_K> with continuous increase in the infeed depth up to the infeed depth at the start <S_A>.
3. Sparking out with the sparking-out time <S_U>.
4. Traversing of the oscillating axis with the grinding width <S_W> as travel path to the starting point and feedrate for transverse infeed <S_K> with continuous increase in the infeed depth up to the infeed depth at the end <S_B>.
5. Sparking out with the sparking-out time <S_U>.

Indicates reiterating sequential steps.

The sequence is repeated until the programmed number of repetitions <S_H> has been reached.
Note

The sequence cannot be interrupted with a single block.

Example

Oscillation with two strokes:

Cycle parameters

- Infeed depth at the start: 0.02 mm
- Infeed depth at the end: 0.01 mm
- Stroke: 100 mm
- Sparking-out time: 1 s
- Transverse feedrate: 1000 mm/min
- Repetitions: 2
- Oscillating and infeed axes: Standard geometry axes

Program code

N10 T1 D1
N20 CYCLE4073(0.02,0.01,100,1,1000,2)
N30 M30

9.4.5 CYCLE4074 - longitudinal grinding with continuous infeed and cancel signal

Syntax

CYCLE4074(<S_GAUGE>, <S_A>, <S_B>, <S_W>, <S_U>, <S_K>, <S_H>,<S_A1>, <S_A2>)

Parameters

<table>
<thead>
<tr>
<th>No.</th>
<th>Parameter</th>
<th>Data type</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>&lt;S_GAUGE&gt;</td>
<td>STRING</td>
<td>Cancel conditions for infeed:</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>1. Number of a rapid input</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>2. Logical expression</td>
</tr>
<tr>
<td>2</td>
<td>&lt;S_A&gt;</td>
<td>REAL</td>
<td>Infeed depth at the start</td>
</tr>
<tr>
<td>3</td>
<td>&lt;S_B&gt;</td>
<td>REAL</td>
<td>Infeed depth at the end</td>
</tr>
<tr>
<td>4</td>
<td>&lt;S_W&gt;</td>
<td>REAL</td>
<td>Grinding width</td>
</tr>
<tr>
<td>5</td>
<td>&lt;S_U&gt;</td>
<td>REAL</td>
<td>Sparking-out time</td>
</tr>
<tr>
<td>6</td>
<td>&lt;S_K&gt;</td>
<td>REAL</td>
<td>Feedrate for transverse infeed</td>
</tr>
</tbody>
</table>
Programmable technology functions

9.4 Oscillating cycles (CYCLE4071 ... CYCLE4079)

<table>
<thead>
<tr>
<th>No.</th>
<th>Parameter</th>
<th>Data type</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td>7</td>
<td>&lt;S_H&gt;</td>
<td>INT</td>
<td>Number of repetitions</td>
</tr>
<tr>
<td>8</td>
<td>&lt;S_A1&gt;</td>
<td>AXIS</td>
<td>Infeed axis (optional) or 1st geometry axis</td>
</tr>
<tr>
<td>9</td>
<td>&lt;S_A2&gt;</td>
<td>AXIS</td>
<td>Oscillating axis (optional) or 2nd geometry axis</td>
</tr>
</tbody>
</table>

**Function**

The cycle is used for the execution of repeating infeeds taking into account e.g. an external cancel signal. The infeed depth can be different at the start and at the end. The depth infeed is cancelled when the cancel condition is satisfied. A complete stroke is always performed after the cancellation of the depth infeed.

**Sequence**

Cancellation of the infeed from the end to the start
Cancellation of the infeed from the start to the end

1. Start of the cycle at the current position of the oscillating axis with infeed depth 0.
2. Traversing of the oscillating axis with the grinding width \(<S_W>\) as travel path and feedrate for transverse infeed \(<S_K>\) with continuous increase in the infeed depth up to the infeed depth at the start \(<S_A>\).
3. Sparking out with the sparking-out time \(<S_U>\).
4. Traversing of the oscillating axis with the grinding width \(<S_W>\) as travel path to the starting point and feedrate for transverse infeed \(<S_K>\) with continuous increase in the infeed depth up to the infeed depth at the end \(<S_B>\).
5. Sparking out with the sparking-out time \(<S_U>\).
6. Cancel signal: The depth infeed is canceled. The machining stops when the next start point is reached.
7. Without Cancel signal: The sequence is repeated until the programmed number of repetitions \(<S_H>\) has been reached.

Indicates reiterating sequential steps.

Note
The sequence cannot be interrupted with a single block.

Resources
As resources, the cycle uses a block-wide synchronized action and a synchronized action variable. The synchronized action is determined dynamically from the free area of the synchronized action range (CUS.DIR - 1 ..., CMA.DIR - 1000 ..., CST.DIR – 1199 ...). SYG_IS[1] is used as the synchronized action variable.
Examples

Example 1: Oscillation with two strokes:
Cycle parameters
- Infeed depth at the start: 0.02 mm
- Infeed depth at the end: 0.01 mm
- Stroke: 100 mm
- Sparking-out time: 1 s
- Transverse feedrate: 1000 mm/min
- Repetitions: 2
- Oscillating and infeed axes: Standard geometry axes
Cancel signal: Rapid input 1 ($A_{IN}[1]$)

Program code
N10 T1 D1
N20 CYCLE4074("1",0.02,0.01,100,1,1000,2)
N30 M30

Example 2: Oscillation with two strokes:
Cycle parameters
- Infeed depth at the start: 0.02 mm
- Infeed depth at the end: 0.01 mm
- Stroke: 100 mm
- Sparking-out time: 1 s
- Transverse feedrate: 1000 mm/min
- Repetitions: 2
- Oscillating and infeed axes: Standard geometry axes
Cancel signal: Variable $A_{DBR}[20] < 0.01$

Program code
N10 T1 D1
N20 CYCLE4074("($A_{DBR}[20]<0.01)$",0.02,0.01,100,1,1000,2)
N30 M30

9.4.6 CYCLE4075 - surface grinding with infeed at the reversal point

Syntax
CYCLE4075(<S_I>, <S_J>, <S_K>, <S_A>, <S_R>, <S_F>, <S_P>, <S_A1>, <S_A2>)
Parameters

<table>
<thead>
<tr>
<th>No.</th>
<th>Parameter</th>
<th>Data type</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>(&lt;S_I&gt;)</td>
<td>REAL</td>
<td>Infeed depth at the start</td>
</tr>
<tr>
<td>2</td>
<td>(&lt;S_J&gt;)</td>
<td>REAL</td>
<td>Infeed depth at the end</td>
</tr>
<tr>
<td>3</td>
<td>(&lt;S_K&gt;)</td>
<td>REAL</td>
<td>Total infeed depth</td>
</tr>
<tr>
<td>4</td>
<td>(&lt;S_A&gt;)</td>
<td>REAL</td>
<td>Grinding width</td>
</tr>
<tr>
<td>5</td>
<td>(&lt;S_R&gt;)</td>
<td>REAL</td>
<td>Feedrate for infeed</td>
</tr>
<tr>
<td>6</td>
<td>(&lt;S_F&gt;)</td>
<td>REAL</td>
<td>Feedrate for transverse infeed</td>
</tr>
<tr>
<td>7</td>
<td>(&lt;S_P&gt;)</td>
<td>REAL</td>
<td>Sparking-out time</td>
</tr>
<tr>
<td>8</td>
<td>(&lt;S_A1&gt;)</td>
<td>AXIS</td>
<td>Infeed axis (optional)</td>
</tr>
<tr>
<td>9</td>
<td>(&lt;S_A2&gt;)</td>
<td>AXIS</td>
<td>Oscillating axis (optional)</td>
</tr>
</tbody>
</table>

Function

The cycle is used for machining with a total infeed depth in infeed steps. The infeed depths at the start and at the end can be different. There is a tangential motion between the infeeds.

The positional data P1 to P4 can be negative or positive.

The specification of the infeed axis and/or oscillating axis is optional. If one or both parameters are not specified, the cycle uses the first two geometry axes of the channel.

If the sum of the infeed depth at the start and end is 0 or the total infeed depth is 0, only one sparking-out stroke is performed.

Sequence

Total infeed depth reached with infeed at the second reversal point
Total infeed depth reached with infeed at the first reversal point

① Start of the cycle at the current position of the oscillating axis.
② Traversing of the infeed axis to the infeed depth at the start \(<S_I>\) with the feedrate for infeed \(<S_R>\).
③ Sparking out with the sparking-out time \(<S_P>\).
④ Traversing of the oscillating axis with the grinding width \(<S_A>\) as travel path and the feedrate for transverse infeed \(<S_F>\).
⑤ Traversing of the infeed axis to the infeed depth at the end \(<S_J>\) with the feedrate for infeed \(<S_R>\).
⑥ Sparking out with the sparking-out time \(<S_P>\).
⑦ Traversing of the oscillating axis with the grinding width \(<S_A>\) as travel to the starting point and the feedrate for transverse infeed \(<S_F>\).

Indicates reiterating sequential steps.
The sequence is repeated until the total infeed depth \(<S_K>\) has been reached. The last stroke is then distributed unevenly.

Note
The sequence cannot be interrupted with a single block.

Example
Oscillation with:
- 0.02 mm infeed depth at the start
- 0.01 mm infeed depth at the end
- Total infeed depth 1 mm
- 100 mm stroke
- Infeed feedrate 1 mm/min
- Transverse feedrate 1000 mm/min
- 1 second sparking-out time
- Standard geometry axes

**Program code**

```
N10 T1 D1
N20 CYCLE4075(0.02,0.01,1,100,1,1000,1)
N30 M30
```

9.4.7 CYCLE4077 - surface grinding with infeed at the reversal point and cancel signal

**Syntax**

```
CYCLE4077(<S_GAUGE>, <S_I>, <S_J>, <S_K>, <S_A>, <S_R>, <S_F>,
<S_P>, <S_A1>, <S_A2>)
```

**Parameters**

<table>
<thead>
<tr>
<th>No.</th>
<th>Parameter</th>
<th>Data type</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>&lt;S_GAUGE&gt;</td>
<td>STRING</td>
<td>Cancel condition for infeed:</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>• Number of a rapid input</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>• Logical expression</td>
</tr>
<tr>
<td>2</td>
<td>&lt;S_I&gt;</td>
<td>REAL</td>
<td>Infeed depth at the start</td>
</tr>
<tr>
<td>3</td>
<td>&lt;S_J&gt;</td>
<td>REAL</td>
<td>Infeed depth at the end</td>
</tr>
<tr>
<td>4</td>
<td>&lt;S_K&gt;</td>
<td>REAL</td>
<td>Total infeed depth</td>
</tr>
<tr>
<td>5</td>
<td>&lt;S_A&gt;</td>
<td>REAL</td>
<td>Grinding width</td>
</tr>
<tr>
<td>6</td>
<td>&lt;S_R&gt;</td>
<td>REAL</td>
<td>Feedrate for infeed</td>
</tr>
<tr>
<td>7</td>
<td>&lt;S_F&gt;</td>
<td>REAL</td>
<td>Feedrate for transverse infeed</td>
</tr>
<tr>
<td>8</td>
<td>&lt;S_P&gt;</td>
<td>REAL</td>
<td>Sparking-out time</td>
</tr>
<tr>
<td>9</td>
<td>&lt;S_A1&gt;</td>
<td>AXIS</td>
<td>Infeed axis (optional)</td>
</tr>
<tr>
<td>10</td>
<td>&lt;S_A2&gt;</td>
<td>AXIS</td>
<td>Oscillating axis (optional)</td>
</tr>
</tbody>
</table>

**Function**

The cycle is used for machining with a total infeed depth in infeed steps. The infeed depths at the start and at the end can be different. There is a tangential motion between the infeeds. The depth infeed is cancelled when the cancel signal of the rapid input is 1 or the cancel condition is satisfied. A complete stroke is performed after the cancellation.

The positional data P2 to P5 can be negative or positive.

The specification of the infeed axis and/or oscillating axis is optional. If one or both parameters are not specified, the cycle uses the first two geometry axes of the channel.

If the sum of the infeed depth at the start and end is 0 or the total infeed depth is 0, only one sparking-out stroke is performed.
Sequence

Cancellation of the infeed at the end

Programming technology functions

9.4 Oscillating cycles (CYCLE4071 ... CYCLE4079)
Cancellation of the infeed at the start

1. Start of the cycle at the current position of the oscillating axis.
2. Traversing of the infeed axis to the infeed depth at the start \( S_I \) with the feedrate for infeed \( S_R \).
3. Sparking out with the sparking-out time \( S_P \).
4. Traversing of the oscillating axis with the grinding width \( S_A \) as travel path and the feedrate for transverse infeed \( S_F \).
5. Traversing of the infeed axis to the infeed depth at the end \( S_J \) with the feedrate for infeed \( S_R \).
6. Sparking out with the sparking-out time \( S_P \).
7. Traversing of the oscillating axis with the grinding width \( S_A \) as travel path to the starting point and the feedrate for transverse infeed \( S_F \).
8. Cancel signal: The machining stops when the next start point is reached.
9. Without Cancel signal: The sequence is repeated until the total infeed depth \( S_K \) has been reached. The last stroke is then distributed unevenly.

Indicates reiterating sequential steps.

**Note**

The sequence cannot be interrupted with a single block.

**Resources**

As resources, the cycle uses a block-wide synchronized action and a synchronized action variable. The synchronized action is determined dynamically from the free area of the synchronized action range \( (CUS.DIR - 1 ..., CMA.DIR - 1000 ..., CST.DIR – 1199 ...) \). \( SYG-IS[1] \) is used as the synchronized action variable.
Examples

Example 1
Oscillation with:
- 0.02 mm infeed depth at the start
- 0.01 mm infeed depth at the end
- Total infeed depth 1 mm
- 100 mm stroke
- Infeed feedrate 1 mm/min
- Transverse feedrate 1000 mm/min
- 1 second sparking-out time
- Standard geometry axes
Cancel signal: Rapid input 1 ($A\_IN[1]$

Program code

```
N10 T1 D1
N20 CYCLE4077("1",0.02,0.01,1,100,1,1000,1)
N30 M30
```

Example 2
Oscillation with:
- 0.02 mm infeed depth at the start
- 0.01 mm infeed depth at the end
- Total infeed depth 1 mm
- 100 mm stroke
- Infeed feedrate 1 mm/min
- Transverse feedrate 1000 mm/min
- 1 second sparking-out time
- Standard geometry axes
Cancel signal: Dual-port RAM variable 20 less than 0.01 ($A\_DBR[20] < 0.01$)

Program code

```
N10 T1 D1
N20 CYCLE4077("($A\_DBR[20]<0.01$",0.02,0.01,1,100,1,1000,1)
N30 M30
```
9.4.8 CYCLE4078 - surface grinding with continuous infeed

Syntax

```plaintext
CYCLE4078(<S_I>, <S_J>, <S_K>, <S_A>, <S_F>, <S_P>, <S_A1>, <S_A2>)
```

Parameters

<table>
<thead>
<tr>
<th>No.</th>
<th>Parameter</th>
<th>Data type</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>&lt;S_I&gt;</td>
<td>REAL</td>
<td>Infeed depth from the start to the end</td>
</tr>
<tr>
<td>2</td>
<td>&lt;S_J&gt;</td>
<td>REAL</td>
<td>Infeed depth from the end to the start</td>
</tr>
<tr>
<td>3</td>
<td>&lt;S_K&gt;</td>
<td>REAL</td>
<td>Total infeed depth</td>
</tr>
<tr>
<td>4</td>
<td>&lt;S_A&gt;</td>
<td>REAL</td>
<td>Grinding width</td>
</tr>
<tr>
<td>5</td>
<td>&lt;S_F&gt;</td>
<td>REAL</td>
<td>Feedrate</td>
</tr>
<tr>
<td>6</td>
<td>&lt;S_P&gt;</td>
<td>REAL</td>
<td>Sparking-out time</td>
</tr>
<tr>
<td>7</td>
<td>&lt;S_A1&gt;</td>
<td>AXIS</td>
<td>Infeed axis (optional)</td>
</tr>
<tr>
<td>8</td>
<td>&lt;S_A2&gt;</td>
<td>AXIS</td>
<td>Oscillating axis (optional)</td>
</tr>
</tbody>
</table>

Function

The cycle is used for machining with a total infeed depth by means of continuous infeed. The infeed depths from the start to the end and from the end to the start can be different.

The positional data P1 to P4 can be negative or positive.

The specification of the infeed axis and/or oscillating axis is optional. If one or both parameters are not specified, the cycle uses the first two geometry axes of the channel.

If the sum of the infeed depths P1 and P2 is 0 or the total infeed depth is 0, only one sparking-out stroke is performed.
Programming technology functions

9.4 Oscillating cycles (CYCLE4071 ... CYCLE4079)

Sequence

1. Start of the cycle at the current position of the oscillating axis with infeed depth 0.
2. Traversing of the oscillating axis with the grinding width \(<S_A>\) as travel path and feedrate \(<S_F>\) with continuous increase in the infeed depth up to the infeed depth at the start \(<S_I>\).
3. Sparking out with the sparking-out time \(<S_P>\).
4. Traversing of the oscillating axis with the grinding width \(<S_A>\) as travel path to the starting point and feedrate \(<S_F>\) with continuous increase in the infeed depth up to the infeed depth at the end \(<S_J>\).
5. Sparking out with the sparking-out time \(<S_P>\).
6. Traversing of the oscillating axis with the grinding width \(<S_A>\) as travel path to the starting point and feedrate \(<S_F>\).

Indicates reiterating sequential steps.

The sequence is repeated until the total infeed depth \(<S_K>\) has been reached. The last stroke is then distributed unevenly.

Note

The sequence cannot be interrupted with a single block.

Example

Oscillation with:
- 20 mm infeed depth at the start
- 10 mm infeed depth at the end
- Total infeed depth 100 mm
- 100 mm stroke
- Feedrate 1000 mm/min
- 1 second sparking-out time
- Standard geometry axes
9.4.9 CYCLE4079 - surface grinding with intermittent infeed

Syntax

CYCLE4079(<S_I>, <S_J>, <S_K>, <S_A>, <S_R>, <S_F>, <S_P>, <S_A1>, <S_A2>)

Parameters

<table>
<thead>
<tr>
<th>No.</th>
<th>Parameter</th>
<th>Data type</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>&lt;S_I&gt;</td>
<td>REAL</td>
<td>Infeed depth at the start</td>
</tr>
<tr>
<td>2</td>
<td>&lt;S_J&gt;</td>
<td>REAL</td>
<td>Infeed depth at the end</td>
</tr>
<tr>
<td>3</td>
<td>&lt;S_K&gt;</td>
<td>REAL</td>
<td>Total infeed depth</td>
</tr>
<tr>
<td>4</td>
<td>&lt;S_A&gt;</td>
<td>REAL</td>
<td>Grinding width</td>
</tr>
<tr>
<td>5</td>
<td>&lt;S_R&gt;</td>
<td>REAL</td>
<td>Feedrate for infeed</td>
</tr>
<tr>
<td>6</td>
<td>&lt;S_F&gt;</td>
<td>REAL</td>
<td>Feedrate for transverse infeed</td>
</tr>
<tr>
<td>7</td>
<td>&lt;S_P&gt;</td>
<td>REAL</td>
<td>Sparking-out time</td>
</tr>
<tr>
<td>8</td>
<td>&lt;S_A1&gt;</td>
<td>AXIS</td>
<td>Infeed axis (optional)</td>
</tr>
<tr>
<td>9</td>
<td>&lt;S_A2&gt;</td>
<td>AXIS</td>
<td>Oscillating axis (optional)</td>
</tr>
</tbody>
</table>

Function

The cycle is used for machining with a total infeed depth in infeed steps. The infeed depths at the start and at the end can be different. There is a tangential motion between the infeeds.

The positional data P1 to P4 can be negative or positive.

The specification of the infeed axis and/or oscillating axis is optional. If one or both parameters are not specified, the cycle uses the first two geometry axes of the channel.

If the sum of the infeed depth at the start and end is 0 or the total infeed depth is 0, only one sparking-out stroke is performed.
Programming technology functions

9.4 Oscillating cycles (CYCLE4071 ... CYCLE4079)

Sequence

Total infeed depth reached with infeed at the second reversal point
Total infeed depth reached with infeed at the first reversal point

① Start of the cycle at the current position of the oscillating axis.
② Traversing of the infeed axis to the infeed depth at the start \(<S_I>\) with the feedrate for infeed \(<S_R>\).
③ Sparking out with the sparking-out time \(<S_P>\).
④ Traversing of the oscillating axis with the grinding width \(<S_A>\) as travel path and the feedrate for transverse infeed \(<S_F>\).
⑤ Traversing of the infeed axis to the infeed depth at the end \(<S_J>\) with the feedrate for infeed \(<S_R>\).
⑥ Sparking out with the sparking-out time \(<S_P>\).
⑦ Traversing of the oscillating axis with the grinding width \(<S_A>\) as travel path to the starting point and the feedrate for transverse infeed \(<S_F>\).

Indicates reiterating sequential steps.
The sequence is repeated until the total infeed depth \(<S_K>\) has been reached. The last stroke is then distributed unevenly.

Note
The sequence cannot be interrupted with a single block.

Example

Oscillation with:
- 0.02 mm infeed depth at the start
- 0.01 mm infeed depth at the end
- Total infeed depth 1 mm
- 100 mm stroke
- Infeed feedrate 1 mm/min
- Transverse feedrate 1000 mm/min
Programming technology functions

9.4 Oscillating cycles (CYCLE4071 ... CYCLE4079)

- 1 second sparking-out time
- Standard geometry axes

<table>
<thead>
<tr>
<th>Program code</th>
</tr>
</thead>
<tbody>
<tr>
<td>N10 T1 D1</td>
</tr>
<tr>
<td>N20 CYCLE4079(0.02,0.01,1,100,1,1000,1)</td>
</tr>
<tr>
<td>N30 M30</td>
</tr>
</tbody>
</table>
9.5 Aligning a grinding wheel (CYCLE400)

9.5.1 Function

Grinding machines with B axis that can be swiveled are supported using the "Aligning grinding wheel" functions.

The maximum angular range when aligning is limited by the traversing range of the participating rotary axes. Technological limits are also placed on the angular range depending on the tool used. After aligning, the cutting-edge position is automatically adapted.

Definition of angle $\beta$

Angle $\beta$ – that is independent of the machine – is used to align grinding wheels.

In the initial state of the machine kinematics, a grinding wheel can be oriented according to Z or X.

Machining the opposite side

It can be selected as to whether the side of the grinding wheel that corresponds to the cutting-edge position is machined – or the opposite side.

Retraction

Retraction is possible before swiveling the grinding wheel.

Machine manufacturer

Please refer to the machine manufacturer's specifications.

9.5.2 Calling the cycle

Procedure

1. The part program to be executed has been created and you are in the editor.
2. Press the "Various" softkey.
3. Press the "Align grinding wheel" softkey.
   The “Align Grinding Wheel” window opens.
### Programming technology functions

#### 9.5 Aligning a grinding wheel (CYCLE400)

**Parameter**

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Description</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>TC</td>
<td>Name of the swivel data set</td>
<td></td>
</tr>
<tr>
<td>b</td>
<td>Angle of the tool to the axis of rotation</td>
<td>Degrees</td>
</tr>
<tr>
<td>Machining the opposite side</td>
<td>• Yes: Machining opposite the cutting edge position</td>
<td></td>
</tr>
<tr>
<td></td>
<td>• No: Machining on the cutting edge position side</td>
<td></td>
</tr>
<tr>
<td>Retraction</td>
<td>• Yes: Retract prior to swiveling</td>
<td></td>
</tr>
<tr>
<td></td>
<td>• No: Do no retract prior to swiveling</td>
<td></td>
</tr>
</tbody>
</table>
Grinding with a B axis (only for cylindrical grinding machines)

10.1 Overview

Cylindrical grinding machines with a B axis are supported by means of tool carriers.

![Diagram of tool carrier and grinding wheels]

- **1** External grinding wheel
- **2** Internal grinding wheel
- **3** Face grinding wheel

Figure 10-1  Example: Turret with four grinding spindles

**Tool carrier**

A separate tool carrier is set up for each grinding spindle. Each tool carrier has head kinematics with the B axis as the first rotary axis. A semiautomatic rotary axis in the direction of the grinding spindle is set up as the second rotary axis (value: 0° or 180°).

The respective basic position is determined via the offset angle of the B axis (e.g. 0°, 90°, 180°, 270°, arbitrary). If a spindle is mechanically at a slightly inclined angle to the 90° direction (e.g. 3°), then the direction vector of the second rotary axis is also entered accordingly (however, the direction component Y must be 0).

Select the appropriate tool carrier in the "T, S, M" window.

**Machine manufacturer**

Please refer to the machine manufacturer's specifications.

**"Beta" alignment angle**

You can define a skew angle in relation to the basic position with β.
Changing the cutting edge position

The second rotary axis is controlled with the aid of the "Machining opposite" selection box, and the cutting edge position is changed via the CUTMOD function (e.g. internal grinding).

Machining opposite = no
Machining opposite = yes
10.2 **T, S, M window for set-up B axis**

**Alignment of the B axis**

<table>
<thead>
<tr>
<th>Display</th>
<th>Meaning</th>
</tr>
</thead>
</table>
| T       | Input of the tool (name or location number)  
         | You can select a tool from the tool list using the "Select tool" softkey. |
| D       | Cutting edge number of the tool (1 - 9) |
| ST      | Sister tool (for replacement tool strategy) |
| TC      | Name of the swivel data set |
| β       | Input of the angle for the tool alignment |
| Machining opposite | • Yes: Machining opposite the cutting edge position  
         | • No: Machining on the cutting edge position side |
| Spindles 1 and 2  
(e.g. S1) | Spindle selection for master spindle and identification with spindle number |
| Spindle M function | Spindle positioning: Spindle is moved to the desired position. |
| Spindle off: Spindle is stopped |
| CCW rotation: Spindle rotates counterclockwise |
| CW rotation: Spindle rotates clockwise |
| Other M functions | Input of machine functions  
         | Refer to the machine manufacturer's table for the correlation between the meaning and number of the function. |
| G zero offset | Selection of the zero offset (basic reference, G54 - 57)  
         | You can select zero offsets from the tool list of settable zero offsets via the "Zero offset" softkey. |
| Measurement unit | Selection of the unit of measurement (inch, mm).  
         | The setting made here has an effect on the programming. |
| Machining plane | Selection of the machining plane (G17(XY), G18 (ZX), G19 (YZ)) |
| Gear stage | Specification of the gear stage (auto, I - V) |
| Stop position | Input of the spindle position in degrees |

**Note**

**Spindle positioning**

You can use this function to position the spindle at a specific angle, e.g. during a tool change.  
- A stationary spindle is positioned via the shortest possible route.  
- A rotating spindle is positioned and continues to turn in the same direction.
Procedure

1. Select the "JOG" operating mode.

2. Press the "T, S, M" softkey.

3. Enter the name or the number of the tool T.
   - OR -
   Press the "Select tool" softkey to open the tool list, position the cursor on the desired tool and press the "In manual" softkey.
   The tool is transferred to the "T, S, M... window" and displayed in the field of tool parameter "T".

4. Enter the desired parameters.

5. Press the <CYCLE START> key.
   The tool is loaded into the spindle.

---

**Note**

**Angular alignment and cutting edge position**

The "Beta" and "Machining opposite" fields must always be entered together.

---

**Note**

**Selection of the swivel data set**

If only one swivel data set is available, the "TC" selection box is omitted.

Please also refer to the machine manufacturer's instructions.
10.3 Measuring in JOG

10.3.1 Grinding wheel alignment for grinding

The tool measurement screens contain entry fields for aligning the grinding wheel or dresser.

Aligning the tools for the B axis

- TC  
  Name of the swivel data set  
  Note: If only one swivel data set is available, the "TC" selection field is omitted.

- \( \beta \)  
  Input of the angle for the tool alignment

- Machining the opposite side  
  Yes: Machining opposite the cutting edge position  
  No: Machining on the cutting edge position side

10.3.2 Manually measuring the grinding tool (with B axis)

Reference point

A dresser serves as reference point for the measurement of length X or Z.

The reference point of the dresser can be represented by a zero offset or a dressing tool. This setting is permanently stored in the machine data and specified by the machine manufacturer.

Machine manufacturer

Please refer to the machine manufacturer's specifications.

Procedure

1. Select "JOG" mode in the "Machine" operating area.

2. Press the "Meas. tool" softkey.

3. Press the "Measure wheel" softkey.
4. Press the "Select tool" softkey.
The "Tool Selection" window opens.

5. In the "Tool Selection" window, select the grinding tool that you want to measure and press the "OK" softkey.
The cutting edge location must be entered in the tool list.
- OR -
Press the "Tool list" softkey, select in the tool list the grinding tool that you want to measure and press the "In manual" softkey.

The tool is transferred to the "Measure: "Grinding Wheel" window.

6. Select the "Dresser" entry in the "Reference point" selection field.

7. Position the cursor in the "TR" field, press the "Select dresser" softkey, select the dresser for measuring the tool length and press the "OK" softkey.
- OR -
Position the cursor to the "Zero offset" field and press the "Select ZO" softkey.
Select in the "Zero Offset - G54 ... G509" window the desired zero offset and press the "In manual" softkey.

8. Press the "X" or "Z" softkey, depending on which tool length you want to measure.

9. Scratch the dresser using the tool.

10. Press the "Set length" softkey.
The tool length is calculated automatically and entered in the tool list.
The cutting edge location is considered automatically.

---

Note
Active grinding tool
Tool measurement is possible only with an active grinding tool.

10.3.3 Manually measuring the dresser (with B axis)

Reference point
A grinding wheel serves as reference point for measuring length X and length Z.
Procedure

1. Select "JOG" mode in the "Machine" operating area.

2. Press the "Meas. tool" softkey.

3. Press the "Measure dresser" softkey.

**Dresser as tool**

4. Press the "Select dresser" softkey.
   The "Tool Selection" window opens.

5. In the "Tool Selection" window, select the dressing tool to be measured and press the "OK" softkey.
   The cutting edge location must be entered in the tool list.
   - OR -
   Press the "Tool list" softkey, select in the tool list the dressing tool to be measured and press the "In manual" softkey.
   The tool is transferred to the "Measure: Dresser" window.
   - OR -

**Dresser as zero offset**

Position the cursor in the "Zero offset" field and press the "Select ZO" softkey.

Select in the "Zero Offset - G54 ... G509" window the desired zero offset and press the "In manual" softkey.

6. Position the cursor in the "TR" field, press the "Select grinding wheel" softkey.

7. Select the grinding wheel to be used for measuring the dresser length and press the "OK" softkey.

8. Press the "X" or "Z" softkey, depending on which tool length you want to measure.

9. Scratch the dresser using the grinding tool.

10. Press the "Set length" softkey.
10.3.4 Calibrating the swivel axis

Requirement

The grinding wheel must be loaded before the calibration.

Procedure

1. Select "JOG" mode in the "Machine" operating area.
2. Press the "Meas. tool" softkey.
3. Press the "Swivel axis" softkey.
   The "Calibration: Swivel Axis" window opens.
4. Select the required swivel data set in the "TC" selection field, and select on which side of the cutting edge position the machining is to be performed with "Yes" or "No" button in the "Machining opposite" selection field.
5. Select the measuring axis (X or Z).
6. Press the <CYCLE START> key.
   The first β angle is automatically swung in to the fixed default angle of 0°.
7. Then scratch the workpiece and press the "Save β0" softkey.
8. Enter the required angles in the "β1" and "β2" fields, scratch the workpiece and press the corresponding "Save β1" and "Save β2" softkeys.
   The calculation can be performed when all the measurements have been made.
   The offset vector L3 (X and Z) of the swivel data set is displayed as measurement result.
Grinding with a B axis (only for cylindrical grinding machines)

10.3 Measuring in JOG

---

**Note**

The calibration is possible only with an active tool.
Grinding with a B axis (only for cylindrical grinding machines)

10.3 Measuring in JOG
Collision avoidance allows you to avoid collisions and damage while machining a workpiece or creating programs.

**Software option**
You require the "Collision Avoidance ECO (machine)" software option in order to use this function for geometrically primitive protection area elements.

**Software option**
You require the "Collision Avoidance (machine, working area)" software option in order to use this function additionally for protection area elements in the STL and NPP data formats.
(840D sl only)

**Software option**
You require the "Collision Avoidance ADVANCED (machine, workpiece)" software option in order to use this function additionally for the autonomous realization of collision avoidance application.
(840D sl only)

**Machine manufacturer**
Please observe the information provided by the machine manufacturer.

Collision avoidance is based on a machine model. The kinematics of the machine are described as a kinematic chain. For machine parts to be protected, protection areas are attached to these chains. The geometry of the protection areas is defined using protection area elements. The control then knows how they move in the machine coordinate system depending on the position of the machine axes. You then subsequently define the collision pairs, i.e. two protection areas, which are monitored with respect to one another.

The "Collision avoidance" function regularly calculates the clearance from these protection areas. When two protection areas approach one another and a specific safety clearance is reached, an alarm is displayed and the program is stopped before the corresponding traversing block and/or the traversing motion is stopped.

**Note**
The collision monitoring is only valid for single-channel machines.
Note

Referenced axes

The positions of the axes in the machine area must be known so that the protection areas can be monitored. For this reason, collision avoidance is only active after the referencing.

NOTICE

No complete machine protection

Incomplete models, e.g. machine parts, workpieces that have not been modeled or new objects in the working area, are not monitored and can therefore cause collisions.

Additional information

You can find additional information about collision avoidance at:

- Function Manual Basic Functions
- Function Manual Transformations
11.1 Activating collision avoidance

Precondition

- Collision avoidance is setup and an active machine model (kinematic chains) is available.
- The setting "Collision avoidance" has been selected for the AUTO operating mode or for the JOG and MDA operating modes.

Procedure

1. Select the "Machine" operating area
2. Press the <AUTO> key.
3. Press the "Sim. rec." softkey.
4. Press the "Other views" and "Machine area" softkeys.

An active machine model (kinematic chains) is displayed for simultaneous recording.
11.2 Set collision avoidance

Using "Settings", you have the option of separately activating or deactivating the collision monitoring for the Machine operating area (operating modes, AUTO, JOG and MDI) separately for the machine and tools.

Using machine data, you define from which protection level the collision avoidance for the machine or the tool can be activated or deactivated in the operating modes JOG/MDI or AUTO.

**Machine manufacturer**
Please refer to the machine manufacturer's instructions.

<table>
<thead>
<tr>
<th>Setting</th>
<th>Effect</th>
</tr>
</thead>
<tbody>
<tr>
<td>JOG/MDI operating mode</td>
<td>They switch the collision avoidance for the JOG/MDI operating modes on or off.</td>
</tr>
<tr>
<td>Collision avoidance</td>
<td></td>
</tr>
<tr>
<td>AUTO mode</td>
<td>They switch the collision avoidance for the AUTO operating mode on or off depending on machine data $MN_JOG_MODE_MASK</td>
</tr>
<tr>
<td>Collision avoidance</td>
<td></td>
</tr>
<tr>
<td><strong>Note:</strong></td>
<td>Please refer to the machine manufacturer's instructions.</td>
</tr>
<tr>
<td>JOG/MDI Machine</td>
<td>If the collision monitoring for the JOG/MDI operating modes is activated, then as a minimum, the machine protection areas are monitored. The parameter cannot be changed.</td>
</tr>
<tr>
<td>AUTO Machine</td>
<td>If the collision monitoring for the AUTO operating mode is activated, then as a minimum, the machine protection areas are monitored. The parameter cannot be changed.</td>
</tr>
<tr>
<td>JOG/MDI Tools</td>
<td>They switch the collision avoidance of the tool protection areas for the operating modes JOG/MDI on or off.</td>
</tr>
<tr>
<td>AUTO Tools</td>
<td>They switch the collision avoidance of the tool protection areas for the operating mode AUTO on or off.</td>
</tr>
</tbody>
</table>

**Procedure**

1. Select the "Machine" operating area.
2. Select the "JOG", "MDI" or "AUTO" mode.
3. Press the menu forward key and the "Settings" softkey.
4. Press the "Collision avoidance" softkey. The "Collision Avoidance" window opens.
5. In the "Collision avoidance" line for the required operating modes (e.g. for JOG/MDI), select the entry "On" to activate the collision avoidance or "Off" to deactivate collision avoidance.
6. Deactivate the "Tools" checkbox if you only want to monitor the machine protection areas.

See also

Actual value window (Page 37)
Collision avoidance

11.2 Set collision avoidance
12.1 Multi-channel view

The multi-channel view allows you to simultaneously view several channels in the following operating areas:

- "Machine" operating area
- "Program" operating area
12.2 Multi-channel view in the "Machine" operating area

With a multi-channel machine, you have the option of simultaneously monitoring and influencing the execution of several programs.

**Machine manufacturer**
Please observe the information provided by the machine manufacturer.

Displaying the channels in the "Machine" operating area

In the "Machine" operating area, you can display 2 - 4 channels simultaneously.

Using the appropriate settings, you can define the sequence in which channels are displayed. Here, you can also select if you wish to hide a channel.

**Note**
The "REF POINT" function is shown only in the single-channel view.

**Multi-channel view**

2 - 4 channels are simultaneously displayed in channel columns on the user interface.

- Two windows are displayed one above the other for each channel.
- The actual value display is always in the upper window.
- The same window is displayed for both channels in the lower window.
- You can select the display in the lower window using the vertical softkey bar.
  The following exceptions apply when making a selection using the vertical softkeys:
  - The "Actual values MCS" softkey switches over the coordinate systems of both channels.
  - The "Zoom actual value" and "All G functions" softkeys switch into the single-channel view.

**Single-channel view**

If you only wish to monitor one channel for your multi-channel machine, then you can set a permanent single-channel view.

**Horizontal softkeys**

- Block search
  When selecting the block search, the multi-channel view is kept. The block display is displayed as search window.

- Program control
  The "Program Control" window is displayed for the channels configured in the multi-channel view. The data entered here applies for these channels together.

- If you press an additional horizontal softkey in the "Machine" operating area (e.g. "Overstore", "Synchronized actions"), then you change into a temporary single-channel view. If you close the window again, then you return to the multi-channel view.
**Switching between single- and multi-channel view**

Press the <MACHINE> key in order to briefly switch between the single- and multi-channel view in the machine area.

Press the <NEXT WINDOW> key in order to switch between the upper and lower window within a channel column.

**Editing a program in the block display**

You can perform simple editing operations as usual with the <INSERT> key in the actual block display.

If there is not sufficient space, you switch over into the single-channel view.

**Running-in a program**

You select individual channels to run-in the program at the machine.

**Requirement**

- Several channels have been set-up.
- The setting "2 channels", "3 channels" or "4 channels" is selected.

**Displaying/hiding a multi-channel view**

1. Select the "Machine" operating area

2. Select the "JOG", "MDA" or "AUTO" mode.

3. Press the menu forward key and the "Settings" softkey.

4. Press the "Multi-channel view" softkey.
5. In the window "Settings for Multi-Channel View" in the selection box "View", select the required entry (e.g. "2 channels") and define the channels as well as the sequence in which they are to be displayed.

In the basic screen for the "AUTO", "MDA" and JOG" operating modes, the upper window of the left-hand and right-hand channel columns are occupied by the actual value window.

6. Press the "T,F,S" softkey if you wish to view the "T,F,S" window. The "T,F,S" window is displayed in the lower window of the left-hand and right-hand channel column.

**Note:**
The "T,F,S" softkey is present only for smaller operator panels, i.e. up to OP012.
12.3 Multi-channel view for large operator panels

On the OP015 and OP019 operator panels as well as on the PC, you have the option of displaying up to four channels next to each one. This simplifies the creation and run-in for multi-channel programs.

Constraints

- OP015 with a resolution of 1024x768 pixels: up to three channels visible
- OP019 with a resolution of 1280x1024 pixels: up to four channels visible
- The operation of a OP019 requires a PCU50.5

3- or 4-channel view in the "Machine" operating area

Use the multi-channel view settings to select the channels and specify the view.

<table>
<thead>
<tr>
<th>Channel view</th>
<th>Display in the &quot;Machine&quot; operating area</th>
</tr>
</thead>
</table>
| 3-channel view | The following windows are displayed one above the other for each channel:  
|               |   ● Actual Value window  
|               |   ● T,F,S window  
|               |   ● Block Display window  
|               | Selecting functions  
|               |   ● The T,F,S window is overlaid by pressing one of the vertical softkeys. |
| 4-channel view | The following windows are displayed one above the other for each channel:  
|               |   ● Actual Value window  
|               |   ● G functions (the "G functions" softkey is omitted). "All G functions" is accessed with the Menu forward key.  
|               |   ● T,S,F window  
|               |   ● Block Display window  
|               | Selecting functions  
|               |   ● The window showing the G codes is overlaid if you press one of the vertical softkeys. |

Toggling between the channels

Press the <CHANNEL> key to toggle between the channels.

Press the <NEXT WINDOW> key to toggle within a channel column between the three or four windows arranged one above the other.
12.3 Multi-channel view for large operator panels

Note

2-channel display

Unlike the smaller operator panels, the T,F,S window is visible for a 2-channel view in the "Machine" operating area.

Program operating area

You can display as many as ten programs next to each other in the editor.

Displaying a program

You can define the width of the program in the Editor window using the settings in the editor. This means that you can distribute programs evenly - or you can widen the column with the active program.

Channel status

When required, channel messages are displayed in the status display.

Machine manufacturer

Please refer to the machine manufacturer's specifications.
### Setting the multi-channel view

<table>
<thead>
<tr>
<th>Setting</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td>View</td>
<td>Here, you specify how many channels are displayed.</td>
</tr>
<tr>
<td></td>
<td>- 1 channel</td>
</tr>
<tr>
<td></td>
<td>- 2 channels</td>
</tr>
<tr>
<td></td>
<td>- 3 channels</td>
</tr>
<tr>
<td></td>
<td>- 4 channels</td>
</tr>
<tr>
<td>Channel selection and sequence</td>
<td>You specify which channels in which sequence are displayed in the multi-channel view.</td>
</tr>
<tr>
<td>(for &quot;2 - 4 channels&quot; view)</td>
<td></td>
</tr>
<tr>
<td>Visible</td>
<td>Here, you specify which channels are displayed in the multi-channel view.</td>
</tr>
<tr>
<td>(for &quot;2 - 4 channels&quot; view)</td>
<td>You can quickly hide channels from the view.</td>
</tr>
</tbody>
</table>

#### Example

Your machine has 6 channels.

You configure channels 1 - 4 for the multi-channel view and define the display sequence (e.g. 1,3,4,2).

In the multi-channel view, for a channel switchover, you can only switch between the channels configured for the multi-channel view; all others are not taken into consideration. Using the <CHANNEL> key, advance the channel in the "Machine" operating area - you obtain the following views: Channels "1" and "3", channels "3" and "4", channels "4" and "2". Channels "5" and "6" are not displayed in the multi-channel view.

In the single-channel view, toggle between all of the channels (1...6) without taking into account the configured sequence for the multi-channel view.

Using the channel menu, you can always select all channels, also those not configured for multi-channel view. If you switch to another channel, which is not configured for the multi-channel view, then the system automatically switches into the single-channel view. There is no automatic switchback into the multi-channel view, even if a channel is again selected, which has been configured for multi-channel view.

#### Procedure

1. Select the "Machine" operating area.

2. Select the "JOG", "MDA" or "AUTO" mode.
3. Press the menu forward key and the "Settings" softkey.

4. Press the "Multi-channel view" softkey.
   The "Settings for Multi-Channel View" window is opened.

5. Set the multi-channel or single-channel view and define which channels are to be seen in the "Machine" operating area - and in the editor - in which sequence.
13.1 Lists for the tool management

All tools and also all magazine locations that have been created or configured in the NC are displayed in the lists in the Tool area.

All lists display the same tools in the same order. When switching between the lists, the cursor remains on the same tool in the same screen segment.

The lists have different parameters and softkey assignments. Switching between lists is a specific change from one topic to the next.

- **Tool list**
  All parameters and functions required to create and set up tools are displayed.

- **Tool wear**
  All parameters and functions that are required during operation, e.g. wear and monitoring functions, are listed here.

- **Magazine**
  You will find the magazine and magazine location-related parameters and functions for the tools / magazine locations here.

- **Tool data OEM**
  This list can be freely defined by the OEM.

**Sorting the lists**

You can change the sorting within the lists according to:

- The magazine
- The name (tool identifier, alphabetic)
- The tool type
- The T number (tool identifier, numerical)
- The D number

**Filtering the lists**

You can filter the lists according to the following criteria:

- Only display the first cutting edge
- Only tools that are ready to use
- Only tools that have reached the pre-alarm limit
- Only locked tools
- Only tools with active code
Search functions
You have the option of searching through the lists according to the following objects:

- Tool
- Magazine location
- Empty location
13.2 Magazine management

Depending on the configuration, the tool lists support a magazine management.

**Magazine management functions**

- Press the "Magazine" horizontal softkey to obtain a list that displays tools with magazine-related data.
- The Magazine / Magazine location column is displayed in the lists.
- In the default setting, the lists are displayed sorted according to magazine location.
- The magazine selected via the cursor is displayed in the title line of each list.
- The "Magazine selection" vertical softkey is displayed in the tool list.
- You can load and unload tools to and from a magazine via the tool list.

**Machine manufacturer**

Please refer to the machine manufacturer's specifications.
13.3 Tool types

A number of tool types are available when you create a new tool. The tool type determines which geometry data is required and how it will be computed.

Tool types

<table>
<thead>
<tr>
<th>Figure 13-1</th>
<th>Example for the list of favorites (cylindrical grinding)</th>
</tr>
</thead>
<tbody>
<tr>
<td>New tool – favorites</td>
<td></td>
</tr>
<tr>
<td><strong>Type</strong></td>
<td><strong>Identifier</strong></td>
</tr>
<tr>
<td>498 – Grinding wheel</td>
<td></td>
</tr>
<tr>
<td>498 – Dresser</td>
<td></td>
</tr>
<tr>
<td>494 – Dressing roll</td>
<td></td>
</tr>
<tr>
<td>496 – Dressing wheel</td>
<td></td>
</tr>
<tr>
<td>710 – 30 probe</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Figure 13-2</th>
<th>Example for the list of favorites (surface grinding)</th>
</tr>
</thead>
<tbody>
<tr>
<td>New tool – favorites</td>
<td></td>
</tr>
<tr>
<td><strong>Type</strong></td>
<td><strong>Identifier</strong></td>
</tr>
<tr>
<td>418 – Grinding wheel</td>
<td></td>
</tr>
<tr>
<td>498 – Dresser</td>
<td></td>
</tr>
<tr>
<td>495 – Dressing roll</td>
<td></td>
</tr>
<tr>
<td>497 – Dressing wheel</td>
<td></td>
</tr>
<tr>
<td>710 – 30 probe</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Figure 13-3</th>
<th>Tools listed in the &quot;New Tool - Grinding Tools&quot; window (cylindrical grinding)</th>
</tr>
</thead>
<tbody>
<tr>
<td>New tool – grinding tools</td>
<td></td>
</tr>
<tr>
<td><strong>Type</strong></td>
<td><strong>Identifier</strong></td>
</tr>
<tr>
<td>498 – Grinding wheel</td>
<td></td>
</tr>
<tr>
<td>498 – Dresser</td>
<td></td>
</tr>
<tr>
<td>494 – Dressing roll</td>
<td></td>
</tr>
<tr>
<td>496 – Dressing wheel</td>
<td></td>
</tr>
</tbody>
</table>
### 13.3 Tool types

#### New tool - grinding tools

<table>
<thead>
<tr>
<th>Type</th>
<th>Identifier</th>
<th>Tool position</th>
</tr>
</thead>
<tbody>
<tr>
<td>418</td>
<td>Grinding wheel</td>
<td></td>
</tr>
<tr>
<td>490</td>
<td>Dresser</td>
<td></td>
</tr>
<tr>
<td>495</td>
<td>Dressing roll</td>
<td></td>
</tr>
<tr>
<td>497</td>
<td>Dressing wheel</td>
<td></td>
</tr>
</tbody>
</table>

![New tool - grinding tools](image1)

Figure 13-4  Tools listed in the "New Tool - Grinding Tools" window (surface grinding)

#### New tool - special tools

<table>
<thead>
<tr>
<th>Type</th>
<th>Identifier</th>
<th>Tool position</th>
</tr>
</thead>
<tbody>
<tr>
<td>718</td>
<td>3D probe</td>
<td></td>
</tr>
<tr>
<td>711</td>
<td>Edge finder</td>
<td></td>
</tr>
<tr>
<td>712</td>
<td>Mono probe</td>
<td></td>
</tr>
<tr>
<td>713</td>
<td>L probe</td>
<td></td>
</tr>
<tr>
<td>714</td>
<td>Star probe</td>
<td></td>
</tr>
<tr>
<td>726</td>
<td>Calibrating tool</td>
<td></td>
</tr>
</tbody>
</table>

![New tool - special tools](image2)

Figure 13-5  Available tools in the "New Tool - Special Tools" window
13.4 Tool dimensioning

This section provides an overview of the dimensioning of tools.

Tool types

- **G17**
  - Length 1 in Y
  - Length 2 in X
  - Radius in X/Y
- **G18**
  - Length 1 in X
  - Length 2 in Z
  - Radius in Z/X
- **G19**
  - Length 1 in Z
  - Length 2 in Y
  - Radius in Y/Z

Machine manufacturer

The tool length is measured to the center of the ball or to the ball circumference. Please observe the information provided by the machine manufacturer.
Note
A 3D probe must be calibrated before use.
13.5 Tool list

13.5.1 Tool list

All parameters and functions that are required to create and set up the tools are displayed in the tool list.

Each tool is uniquely identified by the tool identifier and the sister tool number.

For the tool display, i.e. when displaying the cutting edge positions, the machine coordinate system is taken into account.

### Tool parameters

<table>
<thead>
<tr>
<th>Column heading</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td>Location</td>
<td>Magazine/location number</td>
</tr>
<tr>
<td>W, L, B, BS</td>
<td>The magazine location numbers</td>
</tr>
<tr>
<td>*</td>
<td>The magazine number is specified first, followed by the location number in the magazine.</td>
</tr>
<tr>
<td></td>
<td>If there is only one magazine, only the location number is displayed.</td>
</tr>
<tr>
<td></td>
<td>Transfer location</td>
</tr>
<tr>
<td></td>
<td>Loader</td>
</tr>
<tr>
<td></td>
<td>Loading station</td>
</tr>
<tr>
<td></td>
<td>Load position</td>
</tr>
<tr>
<td></td>
<td>The following icons can also be displayed for other magazine types (e.g. for a chain):</td>
</tr>
<tr>
<td></td>
<td>Spindle location as an icon</td>
</tr>
<tr>
<td></td>
<td>Locations for gripper 1 and gripper 2 (applies only when a spindle with dual gripper is used) as icons.</td>
</tr>
<tr>
<td>Type</td>
<td>Tool type</td>
</tr>
<tr>
<td></td>
<td>Specific tool offset data is displayed depending on the tool type (represented as an icon).</td>
</tr>
<tr>
<td></td>
<td>The icon identifies the position of the tool; this was selected when the tool was created.</td>
</tr>
<tr>
<td></td>
<td>You have the option of changing the tool position or the tool type using the &lt;SELECT&gt; key.</td>
</tr>
<tr>
<td>Tool name</td>
<td>The tool is identified by the name and the sister tool number. You can enter the name as text or number.</td>
</tr>
<tr>
<td></td>
<td><strong>Note:</strong> The maximum length of tool names is 31 ASCII characters. The number of characters is reduced for Asian characters or Unicode characters. The following special characters are not permitted:</td>
</tr>
<tr>
<td>ST</td>
<td>Sister tool number (for replacement tool strategy).</td>
</tr>
<tr>
<td>D</td>
<td>Cutting edge number</td>
</tr>
<tr>
<td>Wheel Ø</td>
<td>Wheel diameter</td>
</tr>
<tr>
<td></td>
<td>(for grinding wheel - type 400, type 410)</td>
</tr>
</tbody>
</table>
### Column heading | Meaning
---|---
Length X or length Z - Cylindrical grinding | Tool length
| Geometry data length X or length Z
Length Y or length Z - Surface grinding | Tool length
| Geometry data length Y or length Z
Cutting edge radius | Tool radius (for grinding wheel - type 400, type 410; dresser - type 490, dressing roller - type 495, dressing wheel - type 497)
Ø | Tool diameter (for 3D-probe - type 710; edge-probe - type 711; mono-probe - type 712; L-probe - type 713; calibration tool - type 725)
Outer Ø | Outer diameter (for star-type probe - type 714)

### Further parameters
If you have set up unique cutting edge numbers, these are displayed in the first column.

### Column heading | Meaning
---|---
D no. | Unique cutting edge number
SN | Cutting edge number
EC | Setup offsets
| Display of the existing setup offsets

You use the configuration file to specify the selection of parameters in the list.

**Machine manufacturer**

Please refer to the machine manufacturer's specifications.

### References
Information on the configuration and setting up of the tool list can be found in the following references:
Commissioning Manual SINUMERIK Operate (IM9) / SINUMERIK 840D sl

### Icons in the tool list

| Icon/ Marking | Meaning |
---|---|
Tool type | |
Red cross | The tool is disabled. |
Yellow triangle pointing downward | The prewarning limit has been reached. |
Yellow triangle pointing upward | The tool is in a special state. Place the cursor on the marked tool. A tool tip will provide a brief description. |
### Procedure

1. Select the "Parameter" operating area.

2. Press the "Tool list" softkey.
   - The "Tool List" window opens.

### 13.5.2 Additional data

The following tool type requires geometry data that is not included in the tool list display.

#### Tool types with additional geometry data

<table>
<thead>
<tr>
<th>Tool type</th>
<th>Additional parameters</th>
</tr>
</thead>
</table>
| 710 3D-probe | Geometry length (length X, length Y, length Z)  
               | Wear length (Δ length X, Δ length Y, Δ length Z)  
               | Adapter length (length X, length Y, length Z)  |
| 712 Mono-probe | Geometry length (length X, length Y, length Z)  
                | Wear length (Δ length X, Δ length Y, Δ length Z)  
                | Adapter length (length X, length Y, length Z)  
                | Offset angle (angle) |
| 713 L-probe | Geometry length (length X, length Y, length Z)  
               | Wear length (Δ length X, Δ length Y, Δ length Z)  
               | Adapter length (length X, length Y, length Z)  
               | Offset angle (angle)  
               | Boom length (length)  |
### Tool type

<table>
<thead>
<tr>
<th>Tool type</th>
<th>Additional parameters</th>
</tr>
</thead>
<tbody>
<tr>
<td>714 Star-type probe</td>
<td>Geometry length (length X, length Y, length Z)</td>
</tr>
<tr>
<td></td>
<td>Wear length (Δ length X, Δ length Y, Δ length Z)</td>
</tr>
<tr>
<td></td>
<td>Adapter length (length X, length Y, length Z)</td>
</tr>
<tr>
<td></td>
<td>Offset angle (angle)</td>
</tr>
<tr>
<td></td>
<td>Ball diameter (Ø)</td>
</tr>
<tr>
<td>725 Calibration tool</td>
<td>Geometry length (length X, length Y, length Z)</td>
</tr>
<tr>
<td></td>
<td>Wear length (Δ length X, Δ length Y, Δ length Z)</td>
</tr>
<tr>
<td></td>
<td>Adapter length (length X, length Y, length Z)</td>
</tr>
</tbody>
</table>

You can use the configuration file to specify the data to be displayed in the "Additional Data" window.

**Machine manufacturer**

Please refer to the machine manufacturer's specifications.

### Procedure

1. The tool list is opened.
2. In the list, select an appropriate tool, e.g. an angle head cutter.
3. Press the "Additional data" softkey.

   The "Additional Data - ..." window opens.

   The "Additional data" softkey is only active if a tool for which the "Additional Data" window is configured is selected.

### 13.5.3 Creating a new tool

When creating a new tool, the "New Tool - Favorites" window offers you a number of selected tool types, known as "favorites".

If you do not find the desired tool type in the favorites list, then select the requested grinding or special tool via the corresponding softkeys.

**Note**

**Grinding tools**

Additional grinding tools are available depending on the particular machine configuration.

### Procedure

1. The tool list opens.
2. Place the cursor in the tool list at the position where the new tool should be created.
To do this, you can select an empty magazine location or the NC tool memory outside of the magazine.

You may also position the cursor on an existing tool in the area of the NC tool memory. Data from the displayed tool will not be overwritten.

3. Press the "New tool" softkey.

The "New Tool - Favorites" window opens.

- OR -

If you want to create a tool that is not in the list of favorites, press "Grinders 400-499" or "Spec. tool 700-900".

The "New Tool - Grinding Tools" or "New Tool - Special Tools" window opens.

4. Select the tool by placing the cursor on the corresponding tool type and on the icon of the desired cutting edge position.

5. If more than four cutting edge positions are available, select the desired cutting edge position with the aid of the <cursor left> and <cursor right> keys.

6. Press the "OK" softkey.

The tool is added to the tool list with a predefined name. If the cursor is located on an empty magazine location in the tool list, then the tool is loaded to this magazine location.

The tool creation sequence can be defined differently.

**Multiple load positions**

If you have configured several load positions for a magazine, then the "Select Load Position" window opens when a tool is created directly in an empty magazine location or when the "Load" softkey is pressed.

Select the required load position and confirm with the "OK" softkey.

**Additional data**

If configured accordingly, the "New Tool" window opens after the required tool has been selected and confirmed with "OK".

You can define the following data in this window:

- Name
- Tool location type
- Size of tool

**References:**

For a description of configuration options, refer to the
13.5.4 Measuring a tool - tool list

You can measure the tool offset data for the individual tools directly from the tool list.

Note
Tool measurement is only possible with an active tool.

Procedure

1. The tool list opens.

2. Select the tool that you want to measure in the tool list and press the "Measure tool" softkey.
   You jump to the "JOG" operating area and the tool to be measured is entered in the "Measure: Length manual" screen in the "T" field.

3. Select the cutting edge number D and the sister tool number ST.

4. Press the "X", "Y" or "Z" softkey, depending on which tool length you want to measure.

5. Traverse the tool towards the workpiece in the direction that is to be measured and scratch it.

6. Enter the position of the workpiece edge in X0, Y0 or Z0.
   If no value is entered for X0, Y0 or Z0, the value is taken from the actual value display.

7. Press the "Set length" softkey.
   The tool length is calculated automatically and entered in the tool list.

13.5.5 Managing several cutting edges

In the case of tools with more than one cutting edge, a separate set of offset data is assigned to each cutting edge. The number of possible cutting edges depends on the control configuration.

Tool cutting edges that are not required can be deleted.
13.5 Tool list

Procedure

1. The tool list opens.
2. Position the cursor on the tool for which you would like to store more cutting edges.
3. Press the "Cutting edges" softkey in the "Tool list".
4. Press the "New cutting edge" softkey.
   A new data set is stored in the list.
   The cutting edge number is incremented by 1. The offset data is preassigned with the values of the cutting edge where the cursor is located.
5. Enter the offset data for the second cutting edge.
6. Repeat this process if you wish to create more tool edge offset data.
7. Position the cursor on the cutting edge that you want to delete and press the "Delete cutting edge" softkey.
   The data set is deleted from the list. The first tool cutting edge cannot be deleted.

13.5.6 Deleting a tool

Tools that are no longer in use can be deleted from the tool list for a clearer overview.

Procedure

1. The tool list is opened.
2. Place the cursor on the tool that you would like to delete.
3. Press the "Delete tool" softkey.
   A safety prompt is displayed.
4. Press the "OK" softkey if you really want to delete the tool.
   Use this softkey to delete the tool.
   If the tool is in a magazine location, it is unloaded and then deleted.

Multiple load points - tool in magazine location

If you have configured several loading points for a magazine, then the "Loading Point Selection" window appears after pressing the "Delete tool" softkey.

Select the required load point and press the "OK" softkey to unload and delete the tool.
13.5.7 Loading and unloading tools

You can load and unload tools to and from a magazine via the tool list. When a tool is loaded, it is taken to a magazine location. When it is unloaded, it is removed from the magazine and stored in the tool list.

When you are loading a tool, the application automatically suggests an empty location. You may also directly specify an empty magazine location.

You can unload tools from the magazine that you are not using at present. HMI then automatically saves the tool data in the tool list in the NC memory outside the magazine.

Should you want to use the tool again later, simply load the tool with the tool data into the corresponding magazine location again. Then the same tool data does not have to be entered more than once.

Procedure

1. The tool list is opened.

2. Place the cursor on the tool that you want to load into the magazine (if the tools are sorted according to magazine location number you will find it at the end of the tool list).

3. Press the "Load" softkey.

   The "Load to... " window opens.
   The "... location" field is pre-assigned with the number of the first empty magazine location.

4. Press the "OK" softkey to load the tool into the suggested location.

   - OR -
   Enter the location number you require and press the "OK" softkey.

   - OR -
   Press the "Spindle" softkey.

   The tool is loaded into the specified magazine location or spindle.

Several magazines

If you have configured several magazines, the "Load to ..." window appears after pressing the "Load" softkey.

If you do not want to use the suggested empty location, then enter your desired magazine and magazine location. Confirm your selection with "OK".

Multiple load points

If you have configured several loading points for a magazine, then the "Load Point Selection" window appears after pressing the "Load" softkey.
Select the required loading point and confirm with "OK".

**Unloading tools**

1. Place the cursor on the tool that you would like to unload from the magazine and press the "Unload" softkey.
2. Select the required load point in the "Load Point Selection" window.
3. Confirm your selection with "OK".

- OR -

Undo your selection with "Cancel".

### 13.5.8 Selecting a magazine

You can directly select the buffer memory, the magazine, or the NC memory.

**Procedure**

1. The tool list is opened.
2. Press the "Magazine selection" softkey.

If there is only one magazine, you will move from one area to the next (i.e. from the buffer memory to the magazine, from the magazine to the NC memory, and from the NC memory back to the buffer memory) each time you press the softkey. The cursor is positioned at the beginning of the magazine each time.

- OR -

If there is more than one magazine, the "Magazine Selection" window opens. Position the cursor on the desired magazine in this window and press the "Go to ..." softkey. The cursor jumps directly to the beginning of the specified magazine.

**Hiding magazines**

Deactivate the checkbox next to the magazines that you do not want to appear in the magazine list.
The magazine selection behavior with multiple magazines can be configured in different ways.

**Machine manufacturer**
Please refer to the machine manufacturer’s specifications.

**References**
For a description of configuration options, refer to the
Commissioning Manual SINUMERIK Operate (IM9) / SINUMERIK 840D sl

### 13.5.9 Code carrier connection (only 840D sl)

#### 13.5.9.1 Overview
You have the option of configuring a code carrier connection.
This means that the following functions are available in SINUMERIK Operate:
- Creating a new tool from code carrier
- Unloading tools on code carrier

**Software option**
In order to use the functions, you require the option "Tool Ident Connection".

**Further information**
Further information about tool management with code carrier and the configuration of the user interface in SINUMERIK Operate can be found at:
- Function Manual SINUMERIK Integrate for Production AMB, AMC AMM/E
- SINUMERIK Operate Commissioning Manual
- Function Manual for tools
13.5.9.2 Managing tools on a code carrier

With a code carrier connection, in the list of favorites, there is also a tool available.

![Figure 13-8](image)

Creating a new tool from code carrier

1. The tool list is opened.
2. Place the cursor in the tool list at the position where the new tool should be created.
   To do this, you can select an empty magazine location or the NC tool memory outside of the magazine.
   You may also position the cursor on an existing tool in the area of the NC tool memory. Data from the displayed tool will not be overwritten.
3. Press the "New tool" softkey.
   The "New Tool - Favorites" window is opened.
4. Position the cursor on the entry "Tool from code carrier" and press the "OK" softkey.
   The tool data is read from the code carrier, and is displayed in the "New tool" window with the tool type, tool name and possibly with certain parameters.
5. Press the "OK" softkey.
   The tool is added to the tool list with the specified name. If the cursor is located on an empty magazine location in the tool list, then the tool is loaded to this magazine location.
Unloading tool on code carrier

1. The tool list is opened.

2. Place the cursor on the tool that you would like to unload from the magazine and press the "Unload" and "On code carrier" softkeys.

   The tool is unloaded and the data of the tool are then written to the code carrier.

According to the appropriate setting, after being read out on the code carrier, the unloaded tool on the code carrier is deleted from the NC memory.

Delete tool on code carrier

1. The tool list is opened.

2. Position the cursor on the tool on code carrier that you want to delete.

3. Press the "Delete tool" and "On code carrier" softkeys.

   The tool is unloaded and the data of the tool are written to the code carrier. The tool is then deleted from the NC memory.

   The deletion of the tool can be set differently, i.e. the "On code carrier" softkey is not available.

References

A description of the configuration options can be found in the following reference:
Commissioning Manual SINUMERIK Operate (IM9) / SINUMERIK 840D sl
Managing a tool in a file

If, in the tool list settings, the "Permit tool in/out file" option is activated, then an additional entry is available in the list of favorites.

Creating a new tool from file

1. The tool list is opened.
2. Place the cursor in the tool list at the position where the new tool should be created.
   To do this, you can select an empty magazine location or the NC tool memory outside of the magazine.
   You may also position the cursor on an existing tool in the area of the NC tool memory. Data from the displayed tool will not be overwritten.
3. Press the "New tool" softkey.
   The "New Tool - Favorites" window is opened.
4. Position the cursor on the entry "Tool from file" and press the "OK" softkey.
   The "Load tool data" window opens.
5. Navigate to the required file and press the "OK" softkey.
   The tool data is read from the file and is displayed in the "New tool from file" window with the tool type, tool name and possibly certain parameters.
6. Press the "OK" softkey.
   The tool is added to the tool list with the specified name. If the cursor is located on an empty magazine location in the tool list, then the tool is loaded to this magazine location.
The tool creation sequence can be defined differently.

Unloading a tool in a file

1. The tool list is opened.
2. Place the cursor on the tool that you would like to unload from the magazine and press the "Unload" and "In file" softkeys.
3. Navigate to the required directory and press the "OK" softkey.
4. Enter the required file name in the "Name" field and press the "OK" softkey.

The field is preassigned with the tool names.
The tool is unloaded and the data of the tool are written to the file.

According to the corresponding setting, after having been read out, the unloaded tool is deleted from the NC memory.

Deleting a tool in a file

1. The tool list is opened.
2. Position the cursor on the tool that you wish to delete.
3. Press the "Delete tool" and "In file" softkeys.
4. Navigate to the required directory and press the "OK" softkey.

The field is preassigned with the tool names.
The tool is unloaded and the data of the tool are written to the file. The tool is then deleted from the NC memory.
13.6 Tool wear

13.6.1 Tool wear

All parameters and functions that are required during operation are contained in the tool wear list.

Tools that are in use for long periods are subject to wear. You can measure this wear and enter it in the tool wear list. The control then takes this information into account when calculating the tool length or radius compensation. This ensures a consistent level of accuracy during workpiece machining.

Monitoring types

You can automatically monitor the tools’ working times via the workpiece count, tool life or wear. In addition, you can disable tools when you no longer wish to use them.

Note

Combination of monitoring types

You have the option to activate the monitoring of a tool by type or any combination of monitoring types.

Machine manufacturer

Please refer to the machine manufacturer’s specifications.
## Tool parameters

<table>
<thead>
<tr>
<th>Column heading</th>
<th>Meaning</th>
</tr>
</thead>
</table>
| Location                | Magazine/location number  
  - The magazine location numbers  
  The magazine number is specified first, followed by the location number in the magazine.  
  If there is only one magazine, only the location number is displayed.  
  - Transfer location  
  - Loader  
  - Loading station  
  - Load position in the load magazine                                                                                                           |
| W                       | * If activated in magazine selection                                                                                                                                                                    |
| L                       | * The following icons can also be displayed for other magazine types (e.g. for a chain):  
  - Spindle location as an icon  
  - Locations for gripper 1 and gripper 2 (this applies only when a spindle with dual gripper is used) as icons.                                                                               |
| B                       | BS                                                                                                                                         |
| ST                      | Sister tool number (for replacement tool strategy).                                                                                           |
| D                       | Cutting edge number                                                                                                                         |
| Δ length X, Δ length Z  | Wear for length X or wear for length Z                                                                                                       |
| - Cylindrical grinding  |                                                                                                                                              |
| Δ length Y, Δ length Z  | Wear for length Y or wear for length Z                                                                                                       |
| - Surface grinding      |                                                                                                                                              |
| Δ cutting edge radius   | Tool wear of the cutting edge radius  
  (for grinding wheel - type 400, type 410; dresser - type 490, dressing roller - type 495, dressing wheel - type 497)                                                                 |
| Δ Ø                     | Tool wear of the diameter  
  (for 3D-probe - type 710; edge-probe - type 711; mono-probe - type 712; L-probe - type 713; calibration tool - type 725)                                                                 |
| Δ outer Ø               | Tool wear of the outer diameter  
  (for star-type probe - type 714)                                                                                                                |
| T C                     | Selection of tool monitoring  
  - by tool life (T)  
  - by count (C)  
  - by wear (W)  
  - wear, summed offset (S)  
  The wear monitoring is configured via a machine data item. Please refer to the machine manufacturer's instructions.                                                                 |
### 13.6 Tool wear

<table>
<thead>
<tr>
<th>Column heading</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td>Tool life</td>
<td>Tool life</td>
</tr>
<tr>
<td>Workpiece count</td>
<td>Number of workpieces</td>
</tr>
<tr>
<td>Wear</td>
<td>Tool wear</td>
</tr>
<tr>
<td>Wear, summed offset *</td>
<td>*Parameter depends on selection in TC</td>
</tr>
<tr>
<td>Setpoint</td>
<td>Setpoint for tool life, workpiece count, or wear</td>
</tr>
<tr>
<td>Prewarning limit</td>
<td>Specification of the tool life, workpiece count or wear at which a warning is displayed.</td>
</tr>
<tr>
<td>G</td>
<td>The tool is disabled when the checkbox is activated.</td>
</tr>
</tbody>
</table>

### Further parameters

If you have set up unique cutting edge numbers, these are displayed in the first column.

<table>
<thead>
<tr>
<th>Column heading</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td>D no.</td>
<td>Unique cutting edge number</td>
</tr>
<tr>
<td>SN</td>
<td>Cutting edge number</td>
</tr>
<tr>
<td>SC</td>
<td>Summed offsets</td>
</tr>
<tr>
<td></td>
<td>Display of the existing setup offsets</td>
</tr>
</tbody>
</table>

### Icons in the wear list

<table>
<thead>
<tr>
<th>Icon/Marking</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td>Tool type</td>
<td></td>
</tr>
<tr>
<td>Red cross</td>
<td>The tool is disabled.</td>
</tr>
<tr>
<td>Yellow triangle pointing downward</td>
<td>The prewarning limit has been reached.</td>
</tr>
<tr>
<td>Yellow triangle pointing upward</td>
<td>The tool is in a special state. Place the cursor on the marked tool. A tooltip provides a short description.</td>
</tr>
<tr>
<td>Green frame</td>
<td>The tool is preselected.</td>
</tr>
<tr>
<td>Magazine/location number</td>
<td></td>
</tr>
<tr>
<td>Green double arrow</td>
<td>The magazine location is positioned at the change position.</td>
</tr>
<tr>
<td>Gray double arrow (configurable)</td>
<td>The magazine location is positioned at the load position.</td>
</tr>
<tr>
<td>Red cross</td>
<td>The magazine location is disabled.</td>
</tr>
</tbody>
</table>
Procedure

1. Select the "Parameter" operating area.

2. Press the "Tool wear" softkey.

13.6.2 Reactivating a tool

You can replace disabled tools or make them ready for use again.

Preconditions

In order to be able to reactivate a tool, the monitoring function must be activated and a setpoint must be stored.

Procedure

1. The tool wear list is opened.

2. Position the cursor on the disabled tool which you would like to reuse.

3. Press the "Reactivate" softkey.

   The value entered as the setpoint is entered as the new tool life or work-piece count.

   The disabling of the tool is cancelled.

Reactivating and positioning

When the "Reactivate with positioning" function is configured, the selected tool's magazine location will also be positioned at a loading point. You can exchange the tool.

Reactivation of all monitoring types

When the "Reactivation of all monitoring types" function is configured, all the monitoring types set in the NC for a tool are reset during reactivation.

Machine manufacturer

Please refer to the machine manufacturer's specifications.

References

Commissioning Manual SINUMERIK Operate (IM9) / SINUMERIK 840D sl
Multiple load points

If you have configured several loading points for a magazine, then the "Load Point Selection" window appears after pressing the "Load" softkey.

Select the required load point and confirm with the "OK" softkey.
13.7 Tool data OEM

You have the option of configuring the list according to your requirements. Depending on the machine configuration, grinding-specific parameters are displayed in the list with OEM tool data.

Grinding tool-specific parameters

<table>
<thead>
<tr>
<th>Column heading</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td>Min. radius</td>
<td>Minimum radius&lt;br&gt;Limit value for the radius of the grinding wheel to monitor the geometry.</td>
</tr>
<tr>
<td>Actual radius</td>
<td>Actual radius&lt;br&gt;Displays the sum of the geometry value, wear value and, if set, the base dimension.</td>
</tr>
<tr>
<td>Min. width</td>
<td>Minimum wheel width&lt;br&gt;Limit value for the width of the grinding wheel to monitor the geometry.</td>
</tr>
<tr>
<td>Actual width</td>
<td>Actual wheel width&lt;br&gt;The width of the grinding wheel measured, for example, after the dressing.</td>
</tr>
<tr>
<td>Maximum speed</td>
<td>Specifies the maximum speed</td>
</tr>
<tr>
<td>Max. peripheral velocity</td>
<td>Specifies the maximum peripheral velocity</td>
</tr>
<tr>
<td>Wheel angle</td>
<td>Specifies the angle of inclined grinding wheel</td>
</tr>
<tr>
<td>Spindle number</td>
<td>Specifies the programmed spindle number (e.g. grinding wheel peripheral velocity) and spindle to be monitored (e.g. wheel radius and width).</td>
</tr>
<tr>
<td>Param. rad.calc.</td>
<td>Selects the parameter to calculate the radius&lt;br&gt;- Length X&lt;br&gt;- Length Y&lt;br&gt;- Length Z&lt;br&gt;- Radius</td>
</tr>
<tr>
<td>Chaining rule</td>
<td>This parameter defines which tool parameters of tool cutting edge 2 (D2) and tool cutting edge 1 (D1) have to be chained to one another. When the value of a chained parameter is modified, then the other cutting edge is automatically used when chaining the parameter.</td>
</tr>
</tbody>
</table>

Further information

- You can find further information on grinding tools in the Grinding-specific tool offset and tool monitoring chapter of the Tools Function Manual.
- You can find further information on configuring OEM tool data in the SINUMERIK Operate Commissioning Manual.
Procedure

1. Select the "Parameter" operating area.

2. Press the "OEM tool" softkey.

3. Position the cursor on a grinding tool.
## 13.8 Magazine

Tools are displayed with their magazine-related data in the magazine list. Here, you can take specific actions relating to the magazines and the magazine locations. Individual magazine locations can be location-coded or disabled for existing tools.

### Tool parameters

<table>
<thead>
<tr>
<th>Column heading</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td>Location</td>
<td>Magazine/location number</td>
</tr>
<tr>
<td></td>
<td>• Magazine location numbers</td>
</tr>
<tr>
<td></td>
<td>The magazine number is specified first, followed by the location number in the magazine.</td>
</tr>
<tr>
<td></td>
<td>If there is only one magazine, only the location number is displayed.</td>
</tr>
<tr>
<td></td>
<td>• Transfer location</td>
</tr>
<tr>
<td></td>
<td>• Loader</td>
</tr>
<tr>
<td></td>
<td>• Loading station</td>
</tr>
<tr>
<td></td>
<td>• Load position in the load magazine</td>
</tr>
<tr>
<td></td>
<td>The following icons can also be displayed for other magazine types (e.g. for a chain):</td>
</tr>
<tr>
<td></td>
<td>• Spindle location as an icon</td>
</tr>
<tr>
<td></td>
<td>• Locations for gripper 1 and gripper 2 (only when a spindle with dual gripper is used) as icons</td>
</tr>
<tr>
<td>Type</td>
<td>Tool type</td>
</tr>
<tr>
<td></td>
<td>Depending on the tool type (represented by an icon), certain tool offset data is enabled.</td>
</tr>
<tr>
<td></td>
<td>The icon identifies the position of the tool; this was selected when the tool was created.</td>
</tr>
<tr>
<td></td>
<td>You have the option of changing the tool position or the tool type by pressing the &lt;SELECT&gt; key.</td>
</tr>
<tr>
<td>Tool name</td>
<td>The tool is identified by the name and number of the sister tool (ST). You can enter the name as text or number.</td>
</tr>
<tr>
<td></td>
<td><strong>Note</strong>: The maximum length of tool names is 31 ASCII characters. The number of characters is reduced for Asian characters or Unicode characters. The following special characters are not permitted:</td>
</tr>
</tbody>
</table>
Further parameters

If you have set up unique cutting edge numbers, these are displayed in the first column.

<table>
<thead>
<tr>
<th>Column heading</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td>D no.</td>
<td>Unique cutting edge number</td>
</tr>
<tr>
<td>SN</td>
<td>Cutting edge number</td>
</tr>
</tbody>
</table>

Magazine list icons

<table>
<thead>
<tr>
<th>Icon/ marking</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td>Tool type</td>
<td></td>
</tr>
<tr>
<td>Red cross</td>
<td>The tool is disabled.</td>
</tr>
<tr>
<td>Yellow triangle pointing downward</td>
<td>The prewarning limit has been reached.</td>
</tr>
<tr>
<td>Yellow triangle pointing upward</td>
<td>The tool is in a special state. Place the cursor on the marked tool. A tooltip provides a short description.</td>
</tr>
<tr>
<td>Green frame</td>
<td>The tool is preselected.</td>
</tr>
<tr>
<td>Magazine/location number</td>
<td></td>
</tr>
<tr>
<td>Green double arrow</td>
<td>The magazine location is positioned at the change position.</td>
</tr>
<tr>
<td>Gray double arrow (configurable)</td>
<td>The magazine location is positioned at the load position.</td>
</tr>
<tr>
<td>Red cross</td>
<td>The magazine location is disabled.</td>
</tr>
</tbody>
</table>

Procedure

1. Select the "Parameter" operating area.

2. Press the "Magazine" softkey.

13.8.1 Positioning a magazine

You can position magazine locations directly on the loading point.
Procedure

1. The magazine list is opened.
2. Place the cursor on the magazine location that you want to position onto the load point.
3. Press the "Position magazine" softkey.
   The magazine location is positioned on the loading point.

Multiple load points
If you have configured several loading points for a magazine, then the "Loading Point Selection" window appears after pressing the "Position magazine" softkey.

Select the desired loading point in this window and confirm your selection with "OK" to position the magazine location at the loading point.

13.8.2 Relocating a tool
Tools can be directly relocated within magazines to another magazine location, which means that you do not have to unload tools from the magazine in order to load them into a different location.

When you are relocating a tool, the application automatically suggests an empty location. You may also directly specify an empty magazine location.

Machine manufacturer
Please refer to the machine manufacturer's specifications.

Procedure

1. The magazine list is opened.
2. Position the cursor on the tool that you wish to relocate to a different magazine location.
3. Press the "Relocate" softkey.
   The "... relocate from location ... to location ...", window is displayed. The "Location" field is pre-assigned with the number of the first empty magazine location.
4. Press the "OK" softkey to relocate the tool to the recommended magazine location.
   - OR -
   Enter the required magazine number in the "...magazine" field and the required magazine location number in "Location" field.
   Press the "OK" softkey.

   The tool is relocated to the specified magazine location.
13.8 Magazine

Several magazines
If you have set up several magazines, then the "...relocate from magazine... location... to..." window appears after pressing the "Relocate" softkey.
Select the desired magazine and location, and confirm your selection with "OK" to load the tool.

13.8.3 Deleting/unloading/loading/relocating all tools
You have the option of deleting all tools from the magazine list, unloading all tools from the magazine list, loading them to the magazine list, and relocating them in the magazine list at the same time.

Preconditions
The following requirements must be met so that the "Delete all," "Unload all," "Load all," or "Relocate all" softkey is displayed and is available:
- Magazine management is set up
- There is no tool in the buffer / in the spindle

Procedure
1. The magazine list opens.
2. Press the "Delete all" softkey.
   - OR -
   Press the "Unload all" softkey.
   - OR -
   Press the "Load all" softkey.
   - OR -
   Press the "Relocate all" softkey.

   A prompt is displayed asking whether you really want to delete, unload, load, or relocate all tools.
3. Press the "OK" softkey to go ahead and delete, unload, load, or relocate the tools.
   The tools are deleted, unloaded, loaded, or relocated in the displayed order, i.e. as specified by the sorting and the set filter.
4. Press the "Cancel" softkey if you wish to cancel the unloading operation.
13.9 Sorting tool management lists

When you are working with many tools, with large magazines or several magazines, it is useful to display the tools sorted according to different criteria. Then you will be able to find a specific tool more easily in the lists.

Procedure

1. Select the "Parameter" operating area.

2. Press the "Tool list", "Tool wear" or "Magazine" softkey.

3. Press the ">>" and "Sort" softkeys.

The lists are displayed sorted numerically according to magazine location.

4. Press the "Acc. to type" softkey to display the tools arranged by tool type. Identical types are sorted according to their radius.
   
   Press the "Acc. to name" softkey to display the tool names in alphabetical order.

   The sister tool numbers are used to sort tools with the same names.
   - OR -

   Press the "Acc. to T number" softkey to display the tools sorted numerically.

   The list is sorted according to the specified criteria.

Machine manufacturer

Please refer to the machine manufacturer's specifications.
Filtering the tool management lists

The filter function allows you to filter-out tools with specific properties in the tool management lists.

For instance, you have the option of displaying tools during machining that have already reached the prewarning limit in order to prepare the corresponding tools for equipping.

Filter criteria

- Only display the first cutting edge
- Only tools that are ready to use
- Only tools with active code
- Only tools that have reached the prewarning limit
- Only locked tools
- Only tools with remaining quantity of ... to ...
- Only tools with residual tool life of ... to ...
- Only tools with unloading marking
- Only tools with loading marking

Note

Multiple selection

You have the option of selecting several criteria. You will receive an appropriate message if conflicting filter options are selected.

You can configure OR logic operations for the various filter criteria.

Machine manufacturer

Please refer to the machine manufacturer's specifications.

References

A description of the configuration options is provided in
SINUMERIK Operate (IM9) / SINUMERIK 840D sl Commissioning Manual
Procedure

1. Select the "Parameter" operating area.

2. Press the "Tool list", "Tool wear" or "Magazine" softkey.

... 

3. Press the ">>" and "Filter" softkeys.
   The "Filter" window opens.

4. Activate the required filter criterion and press the "OK" softkey.
   The tools that correspond to the selection criteria are displayed in the list.
   The active filter is displayed in the window header.
13.11 Specific search in the tool management lists

There is a search function in all tool management lists, where you can search for the following objects:

- **Tools**
  - You enter a tool name. You can narrow down your search by entering a sister tool number.
  - You have the option of only entering a part of the name as search term.
  - You enter the D number and activate if necessary, the check box "Active D number".

- **Magazine locations or magazines**
  If only one magazine is configured, then the search is made according to the magazine location.
  If several magazines are configured, then it is possible to search a specific magazine location in a specific magazine or just to search in a specific magazine.

- **Empty locations**
  The empty location search is realized using the tool size. The tool size is defined by the number of half locations required to the right, left, top and bottom. All four directions are of significance for a box magazine. For a chain magazine, a disk-type or a turret, only the half locations to the right and left are of significance. The maximum number of half locations, which a tool can occupy is limited to 7.
  If the lists with the location type are used, then the empty location search is made using the location type and location size.
  You can enter the location type as numerical value or as text depending on the particular configuration.

**Machine manufacturer**

Please refer to the machine manufacturer's specifications.

**References**

For a description of configuration options, refer to the Commissioning Manual SINUMERIK Operate (IM9) / SINUMERIK 840D sl

**Procedure**

1. Select the "Parameter" operating area.
2. Press the "Tool list", "Tool wear" or "Magazine" softkey.

...
3. Press the ">>" and "Search" softkeys.

4. Press the "Tool" softkey if you wish to search for a specific tool.
   - OR -
   Press the "Magazine location" softkey if you wish to search for a specific magazine location or a specific magazine.
   - OR -
   Press the "Empty location" softkey if you wish to search for a specific empty location.

5. Press the "OK" softkey.
The search is started.

6. Press the "Search" softkey again if the tool that was found is not the tool that was being searched for.
The search term is kept and with "OK" you start the search for the next tool that corresponds to the entry.

7. Press the "Cancel" softkey to cancel the search.
13.12 Tool details

13.12.1 Displaying tool details

The following parameters of the selected tool can be displayed using softkeys in the "Tool Details" window.

- Tool data (Page 340)
- Grinding data (Page 341)
- Cutting edge data (Page 342)
- Monitoring data (Page 343)

Procedure

1. The tool list, the wear list, the OEM tool list or the magazine is open.

... 

2. Position the cursor to the desired tool.

3. If you are in the tool list or in the magazine, press the ">>" and "Details" softkey.

- OR -

If you are in the wear list or OEM tool list, press the "Details" softkey.

The "Tool Details" window is displayed.

The tool data is displayed in the list.

4. Press the "Grinding data" softkey if you want to display the grinding data.

5. Press the softkey "Cutting edge data" if you wish to display the cutting edge data.

6. Press the softkey "Monitoring data" if you wish to display the monitoring data.

13.12.2 Tool data

The "Tool Details" window provides the following data on the selected tool when the "Tool data" softkey is active.
### 13.12 Tool details

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td>Magazine location</td>
<td>The magazine number is specified first, followed by the location number in the magazine. If there is only one magazine, only the location number is displayed.</td>
</tr>
<tr>
<td>Tool name</td>
<td>The tool is identified by the name and the sister tool number. You can enter the name as text or number.</td>
</tr>
<tr>
<td>ST</td>
<td>Sister tool number (for sister tool strategy)</td>
</tr>
<tr>
<td>D quantity</td>
<td>Number of created cutting edges</td>
</tr>
<tr>
<td>D</td>
<td>Cutting edge number</td>
</tr>
<tr>
<td>Tool state</td>
<td>A: Activate tool</td>
</tr>
<tr>
<td></td>
<td>F: Tool enabled</td>
</tr>
<tr>
<td></td>
<td>X: Block tool</td>
</tr>
<tr>
<td></td>
<td>M: Measure tool</td>
</tr>
<tr>
<td></td>
<td>V: Reaching the prewarning limit</td>
</tr>
<tr>
<td></td>
<td>W: Tool being changed</td>
</tr>
<tr>
<td>Tool size</td>
<td>Standard: Tool does not require an additional location in a magazine.</td>
</tr>
<tr>
<td>Special size</td>
<td>Oversize: The tool occupies two half locations left, two half locations right, one half location top and one half location bottom in a magazine.</td>
</tr>
<tr>
<td>Left</td>
<td>Number of half locations to the left of the tool</td>
</tr>
<tr>
<td>Right</td>
<td>Number of half locations to the right of the tool</td>
</tr>
<tr>
<td>Tool OEM parameters 1 - 6</td>
<td>Freely available parameters</td>
</tr>
</tbody>
</table>

### 13.12.3 Grinding data

The "Tool Details" window provides the following data on the selected tool when the "Tool data" softkey is active.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td>Magazine location</td>
<td>The magazine number is specified first, followed by the location number in the magazine. If there is only one magazine, only the location number is displayed.</td>
</tr>
<tr>
<td>Tool name</td>
<td>The tool is identified by the name and the sister tool number. You can enter the name as text or number.</td>
</tr>
<tr>
<td>ST</td>
<td>Sister tool number (for replacement tool strategy)</td>
</tr>
<tr>
<td>D quantity</td>
<td>Number of created cutting edges</td>
</tr>
<tr>
<td>D</td>
<td>Cutting edge number</td>
</tr>
<tr>
<td>Minimum wheel radius</td>
<td>Specifies the minimum wheel radius</td>
</tr>
</tbody>
</table>
### 13.12 Tool details

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td>Actual wheel radius</td>
<td>Specifies the actual wheel radius.</td>
</tr>
<tr>
<td>Minimum wheel width</td>
<td>Specifies the minimum wheel width</td>
</tr>
<tr>
<td>Actual wheel width</td>
<td>Specifies the actual wheel width</td>
</tr>
<tr>
<td>Maximum speed</td>
<td>Specifies the maximum speed</td>
</tr>
<tr>
<td>Maximum peripheral velocity</td>
<td>Specifies the maximum peripheral velocity.</td>
</tr>
<tr>
<td>Angle of inclined wheel</td>
<td>Specifies the angle of inclined wheel</td>
</tr>
<tr>
<td>Spindle number</td>
<td>Specifies the spindle number</td>
</tr>
<tr>
<td>Parameter for calculating the radius</td>
<td>Selected parameter for the radius calculation</td>
</tr>
<tr>
<td>Chaining rule</td>
<td>This parameter defines which tool parameters of tool cutting edge 2 (D2) and tool cutting edge 1 (D1) have to be chained to one another.</td>
</tr>
</tbody>
</table>

### 13.12.4 Cutting edge data

The "Tool Details" window provides the following data on the selected tool when the "Cutting edge data" softkey is active.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Magazine location</td>
<td>The magazine number is specified first, followed by the location number in the magazine. If there is only one magazine, only the location number is displayed.</td>
</tr>
<tr>
<td>Tool name</td>
<td>The tool is identified by the name and the sister tool number. You can enter the name as text or number.</td>
</tr>
<tr>
<td>ST</td>
<td>Sister tool number (for replacement tool strategy)</td>
</tr>
<tr>
<td>D quantity</td>
<td>Number of created cutting edges</td>
</tr>
<tr>
<td>D</td>
<td>Cutting edge number</td>
</tr>
<tr>
<td>Tool type</td>
<td>Tool icon with type number and current cutting edge position</td>
</tr>
<tr>
<td>Cylindrical grinding</td>
<td></td>
</tr>
<tr>
<td>Geometry</td>
<td>Geometry data, length X</td>
</tr>
<tr>
<td>Wear</td>
<td>Tool wear for length X</td>
</tr>
<tr>
<td>Surface grinding</td>
<td></td>
</tr>
<tr>
<td>Geometry</td>
<td>Geometry data, length X</td>
</tr>
<tr>
<td>Wear</td>
<td>Tool wear for length X</td>
</tr>
<tr>
<td>Radius</td>
<td></td>
</tr>
<tr>
<td>Wear</td>
<td>Wear of the cutting edge radius</td>
</tr>
<tr>
<td>Ø</td>
<td></td>
</tr>
<tr>
<td>Geometry</td>
<td>Tool diameter</td>
</tr>
<tr>
<td>Parameter</td>
<td>Description</td>
</tr>
<tr>
<td>---------------------------</td>
<td>------------------------------------</td>
</tr>
<tr>
<td>Wear</td>
<td>Tool wear, diameter</td>
</tr>
<tr>
<td>Cutting edge OEM parameters 1 - 2</td>
<td></td>
</tr>
</tbody>
</table>

### 13.12.5 Monitoring data

The "Tool Details" window provides the following data on the selected tool when the "Monitoring data" softkey is active.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td>Magazine location</td>
<td>The magazine number is specified first, followed by the location number in the magazine. When there is only one magazine, only the location number is displayed.</td>
</tr>
<tr>
<td>Tool name</td>
<td>The tool is identified by the name and the sister tool number. You can enter the name as text or number.</td>
</tr>
<tr>
<td>ST</td>
<td>Sister tool number (for replacement tool strategy)</td>
</tr>
<tr>
<td>D quantity</td>
<td>Number of created cutting edges</td>
</tr>
<tr>
<td>D</td>
<td>Cutting edge number</td>
</tr>
<tr>
<td>Monitoring type</td>
<td>T - tool life</td>
</tr>
<tr>
<td></td>
<td>C - count</td>
</tr>
<tr>
<td></td>
<td>W - wear</td>
</tr>
<tr>
<td></td>
<td>The wear monitoring is configured via machine data. Please note the specifications of the machine manufacturer.</td>
</tr>
<tr>
<td>Actual value</td>
<td>Actual value for tool life, count or wear</td>
</tr>
<tr>
<td>Setpoint</td>
<td>Setpoint for tool life, count or wear</td>
</tr>
<tr>
<td>Prewarning limit</td>
<td>Specification of the tool life, the count or wear at which a warning is displayed.</td>
</tr>
<tr>
<td>Monitoring OEM parameters 1 - 8</td>
<td></td>
</tr>
</tbody>
</table>
13.13 Changing a tool type

Procedure

1. The tool list, the wear list, the OEM tool list or the magazine opens.

2. Position the cursor in the column "Type" of the tool that you wish to change.

3. Press the <SELECT> key.
   The "Tool Types - Favorites" window opens.

4. Select the desired tool type in the list of favorites or select by pressing the "Grinders 400-499" or "Spec. tool 700-900" softkey.

5. Press the "OK" softkey.
   The new tool type is accepted and the corresponding icon is displayed in the "Type" column.
13.14 Working with Multitool

Using a multitool you have the possibility of storing more than one tool at a magazine location. The multitool itself has two or more locations to accept tools. The tools are directly mounted on the multitool. The multitool is located at a location in the magazine.

Geometrical arrangement of the tools on the multitool
The geometrical arrangement of the tools is defined by the clearance between the locations on the multitool.

The clearance between the locations can be defined as follows:

- Using the multitool location number or
- using the angle of the multitool location

If angle is selected here, then the value of the angle must be entered for each multitool location.

Regarding loading and unloading in a magazine, the multitool is treated as a single unit.

13.14.1 Tool list for multitool

If you work with a multitool, the tool list is supplemented by the column for the multitool location number. As soon as the cursor lands on a multitool in the tool list, certain column headings change.

<table>
<thead>
<tr>
<th>Column header</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td>Location</td>
<td>Magazine/location number</td>
</tr>
<tr>
<td>MT loc.</td>
<td>Multitool location number</td>
</tr>
<tr>
<td>TYPE</td>
<td>Symbol for multitool</td>
</tr>
<tr>
<td>Multitool name</td>
<td>Name of the multitool</td>
</tr>
</tbody>
</table>

![Tool list with multitool in the spindle](image)

Figure 13-10 Tool list with multitool in the spindle
Procedure

1. Select the "Parameter" operating area.

2. Press the "Tool list" softkey.
   The "Tool list" window is opened.

13.14.2 Create multitool

The multitool can be selected in the list of special tool types.

![Selection list for special tools with multitool](image)

Procedure

1. The tool list is opened.

2. Position the cursor at the position where the tool is to be created.
   For this, you can select an empty magazine location or the NC tool storage outside the magazine.
   You may also position the cursor on an existing tool in the NC tool storage area. Data from the displayed tool will not be overwritten.

3. Press the "New tool" softkey.
   The "New Tool - Favorites" window opens.
4. Press the "Special tool 700-900" softkey.
5. Select the multitool and press the "OK" softkey. The "New Tool" window opens.
6. Enter the multitool name and define the number of multitool locations.
   If you wish to define the distance of the tools based on the angle, select the "Angle input" check box, and for each multitool location, enter the distance to the reference location as an angle.

<table>
<thead>
<tr>
<th>Multitool name</th>
<th>No. of locs.</th>
<th>Angle input</th>
<th>Multitool angle</th>
</tr>
</thead>
<tbody>
<tr>
<td>MULTITOOL3</td>
<td>3</td>
<td>✓</td>
<td>1 8.888</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>2 128.888</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>3 238.888</td>
</tr>
</tbody>
</table>

The multitool is created in the tool list.

**Note**

The tool creation sequence can be defined differently.

**Machine manufacturer**

Please observe the information provided by the machine manufacturer.

**13.14.3 Equipping multitool with tools**

**Precondition**

A multitool has been created in the tool list.

**Procedure**

1. The tool list opens.
2. Select the required multitool, position the cursor on an empty multitool location.
3. Press the "New tool" softkey.
4. Select the required tool from the relevant selection list, e.g. favorites.
Load multitool

1. Select the required multitool, position the cursor on an empty multitool location.
2. Press the "Load" softkey.
3. The "Load with..." window opens.
4. Select the required tool.

Load the tool into the multitool

1. Position the cursor on the tool that you want to load into the multitool.
2. Press the "Load" and "Multitool" softkeys.
3. The "Load on..." window opens.
4. Select the required multitool and the multitool location into which you wish to load the tool.

13.14.4 Removing a tool from multitool

If the multitool has been mechanically re-equipped with new tools, the old tools in the tool list must be removed from the multitool.

This is done by positioning the cursor on the line on which the tool is located that is to be removed. When it is unloaded, the tool is automatically saved in the tool list outside the magazine in the NC memory.

Procedure

1. The tool list is opened.
2. Position the cursor on the tool that you would like to unload from the multitool and press the "Unload" softkey.
   - OR -
   Position the cursor on the tool that you would like to remove from the multitool and delete and press the "Delete tool" softkey.
13.14.5 Deleting multitool

Procedure

1. The tool list opens.
2. Position the cursor on the multitool that you wish to delete.
3. Press the "Delete multitool" softkey.
   The multitool with all of the tools that are located in it is deleted.

13.14.6 Loading and unloading multitool

Procedure

1. The tool list is opened.

**Loading a multitool into the magazine**

2. Position the cursor on the multitool that you wish to load into the magazine.
3. Press the "Load" softkey.
   The "Load on" window opens.
   The "Locat." field contains the number of the first empty magazine location by default.
4. Press the "OK" softkey to load the multitool into the suggested empty location.
   - OR -
   Enter the location number you require and press the "OK" softkey.
   The multitool together with the tools in it is loaded to the specified magazine location.

**Loading a multitool into a magazine**

2. Position the cursor on the desired empty magazine location.
3. Press the "Load" softkey.
   The "Load with" window opens.
4. Select the desired multitool.
5. Press the "OK" softkey.
Unloading a multitool

2. Position the cursor on the multitool that you wish to unload from the magazine.

3. Press the "Unload" softkey.
   The multitool is removed from the magazine and is saved in the NC memory at the end of the tool list.

13.14.7 Positioning multitool

You can position a magazine. In this case, a magazine location is positioned to the loading point.

Multitools that are located in a spindle can also be positioned. The multitool is rotated to bring the affected multitool location into the machining position.

Procedure

1. The magazine list is opened.
   The multitool is in the spindle.

2. Position the cursor on the multitool location that you want to bring into the machining position.

3. Press the "Position multitool" softkey.

13.14.8 Relocating a multitool

Multitools can be directly relocated to another magazine location within magazines, which means that you do not have to unload multitools with the tools in question from the magazine in order to relocate them to a different location.

When you are relocating a multitool, the system automatically recommends an empty location. You may also directly specify an empty magazine location.

Procedure

1. Select the "Parameter" operating area.

2. Press the "Magazine" softkey.

3. Position the cursor at the multitool that you wish to relocate to a different magazine location.
4. Press the "Relocate" softkey.
The "... relocate from location ... to location ..." window is displayed. The "Location" field is pre-assigned with the number of the first empty magazine location.

5. Press the "OK" softkey to relocate the multitool to the suggested magazine location.
- OR -
Enter the required magazine number in the "...magazine" field and the required magazine location number in "Location" field.

**Note:**
Please observe the information provided by the machine manufacturer.

Press the "OK" softkey.
The multitool with the tools is relocated to the specified magazine location.

### 13.14.9 Reactivating a multitool

A multitool and tools located on the multitool can be disabled independently of one another.

If a multitool is disabled, the tools of the multitool can no longer be loaded for use by means of a tool change.

If only one tool on a multitool has a set monitoring function and the tool life or the workpiece count has expired, the tool and the multitool on which the tool is located are disabled. The other tools on the multitool are not disabled.

**Machine manufacturer**
Please observe the information provided by the machine manufacturer.

If several tools with monitoring are mounted on the multitool and the tool life or the workpiece count has expired for one tool, only this tool is disabled.

<table>
<thead>
<tr>
<th>TOOL</th>
<th>Tool wear</th>
<th>W2_Zwischenspeicher</th>
</tr>
</thead>
<tbody>
<tr>
<td>D no</td>
<td>Loc.</td>
<td>MT LO.</td>
</tr>
<tr>
<td>---</td>
<td>---</td>
<td>---</td>
</tr>
<tr>
<td>1</td>
<td>1</td>
<td>MULTITOOL_U57</td>
</tr>
<tr>
<td>1</td>
<td>2</td>
<td>SCHEIBER_R322</td>
</tr>
<tr>
<td>1</td>
<td>1</td>
<td>SCHEIBER_R332</td>
</tr>
<tr>
<td>1</td>
<td>1</td>
<td>3D_T_S88</td>
</tr>
</tbody>
</table>

**Reactivating**
If a tool with an expired tool life or workpiece count that is mounted on a multitool is reactivated, the tool life or workpiece count is set to the target value for this tool and the tool and the multitool are enabled again.
If a multitool is reactivated, on which tools with monitoring are mounted, the tool life or workpiece count are set to the target value for all tools on that multitool, irrespective of whether the tools are disabled or not.

**Preconditions**

In order to be able to reactivate a tool, the monitoring function must be activated and a setpoint must be stored.

**Procedure**

1. Select the "Parameter" operating area.
2. Press the "Tool wear" softkey.
3. Position the cursor at the multitool that is disabled and which you would like to reactivate.  
   - OR -  
   Position the cursor on the tool that you would like to activate again.
4. Press the "Reactivate" softkey.
   The value entered as the setpoint is entered as the new tool life or workpiece count.
   The tool and the multitool are then enabled again.

**Reactivating and positioning**

If the "Reactivate with positioning" function is configured, the magazine location at which the selected multitool is located is also positioned to the loading point. You can change the multitool.

**Reactivation of all monitoring types**

When the "Reactivation of all monitoring types" function is configured, all the monitoring types set in the NC for a tool are reset during reactivation.

**Machine manufacturer**

Please observe the information provided by the machine manufacturer.

**References**

HMI sl / SINUMERIK 840D sl Commissioning Manual
13.15 **Settings for tool lists**

In the "Settings" window you have the following options to set the view in the tool lists:

- **Display only one magazine in "Magazine sort"**
  - You can limit the display to one magazine. The magazine is displayed with the assigned buffer magazine locations and the tools not loaded.
  - You may set via a configuration if you want to jump to the next magazine by clicking softkey "Magazine selection" or if the dialog "Magazine selection" is switched over to any magazine.

- **Display only spindle in buffer.**
  In order to display only the spindle location during operation, the remaining locations of the buffer are hidden.

- **Permits tool in/out file**
  - When creating a new tool, the tool data can be loaded from a file.
  - When deleting or unloading a tool, the tool data can be backed up in a file.

- **Switch on adapter-transformed view**
  - Geometry lengths and the application offsets are displayed in a transformed manner in the tool list.
  - In the tool wear list, the wear length and the sum offsets are displayed in a transformed manner.

**Machine manufacturer**

Please refer to the machine manufacturer's specifications.

**References**

For further information about configuring the settings, please refer to the following references:
Commissioning Manual SINUMERIK Operate (IM9) / SINUMERIK 840D sl

**Procedure**

1. Select the "Parameter" operating area.

2. Press the "Tool list", "Tool wear" or "Magazine" softkey.
3. Press the "Continue" and "Settings" softkeys.

4. Activate the corresponding check box for the desired setting.
Managing programs

14.1 Overview

You can access programs at any time via the Program Manager for execution, editing, copying, or renaming.

Programs that you no longer require can be deleted to release their storage space.

<table>
<thead>
<tr>
<th>NOTICE</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Execution from USB-FlashDrive</strong></td>
</tr>
<tr>
<td>Direct execution from a USB-FlashDrive is not recommended.</td>
</tr>
<tr>
<td>There is no protection against contact problems, falling out, breakage through knocking or unintentional removal of the USB-FlashDrive during operation.</td>
</tr>
<tr>
<td>Disconnecting it during operation will result in the stopping of the machining and thus to the workpiece being damaged.</td>
</tr>
</tbody>
</table>

Storage for programs

Possible storage locations are:

- NC
- Local drive
- Network drives
- USB drives
- FTP drives
- V24

Software options

To display the "Local drive" softkey, you require the "Additional 256 MB HMI user memory on CF card of the NCU" option (not for SINUMERIK Operate on PCU50 or PC/PG).
Data exchange with other workstations

You have the following options for exchanging programs and data with other workstations:

- USB drives (e.g. USB-FlashDrive)
- Network drives
- FTP drives

Choosing storage locations

In the horizontal softkey bar, you can select the storage location that contains the directories and programs that you want to display. In addition to the "NC" softkey, via which the passive file system data can be displayed, additional softkeys can be displayed.

The "USB" softkey is only operational when an external storage medium is connected (e.g. USB-FlashDrive on the USB port of the operator panel).

Displaying documents

You can display documents on all drives of the program manager (e.g. in the local drive or USB) and via the data tree of the system data. Various data formats are supported:

- PDF
- HTML
  It is not possible to preview HTML documents.
- Various graphic formats (e.g. BMP or JPEG)
- DXF

Software options

You require the "DXF reader" option in order to display DXF files.

Note

FTP drive

It is not possible to preview documents on the FTP drive.

Structure of the directories

In the overview, the icons in the left-hand column have the following meaning:

- Directory
- Program

All directories have a plus sign when the program manager is called for the first time.
The plus sign in front of empty directories is removed after they have been read for the first time.

The directories and programs are always listed complete with the following information:

- **Name**
  The name can be a maximum of 24 characters long. Permissible characters include all upper-case letters (without accents), digits, and underscores.

- **Type**
  - Directory: DIR or WPD
  - Dressing program: DRS directory
  - Program: MPF
  - Subprogram: SPF
  - Initialization programs: INI
  - Job lists: JOB
  - Tool data: TOA
  - Magazine assignment: TMA
  - Zero points: UFR
  - R parameters: RPA
  - Global user data/definitions: GUD
  - Setting data: SEA
  - Protection zones: PRO
  - Sag: CEC

- **Size (in bytes)**

- **Date/time (of creation or last change)**

### Active programs

Selected, i.e. active programs are identified with a green icon.

<table>
<thead>
<tr>
<th>CHAN</th>
<th>Name</th>
<th>Type</th>
<th>Length</th>
<th>Date</th>
<th>Time</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>DREHENT9</td>
<td>WPO</td>
<td>8300</td>
<td>12/03/03</td>
<td>12:03:38 PM</td>
</tr>
<tr>
<td></td>
<td>GGG</td>
<td>WPO</td>
<td>11300</td>
<td>12/03/03</td>
<td>12:03:38 PM</td>
</tr>
<tr>
<td></td>
<td>JOBSHOP_MEHRAK</td>
<td>WPO</td>
<td>8300</td>
<td>12/03/03</td>
<td>13:18:27 PM</td>
</tr>
<tr>
<td></td>
<td>MEHRIC</td>
<td>WPO</td>
<td>11300</td>
<td>12/03/03</td>
<td>12:49:23 PM</td>
</tr>
<tr>
<td></td>
<td>MEHRKANAL</td>
<td>WPO</td>
<td>11300</td>
<td>12/03/03</td>
<td>12:47:20 PM</td>
</tr>
<tr>
<td></td>
<td>SIM CHESS KING</td>
<td>WPO</td>
<td>11300</td>
<td>12/03/03</td>
<td>3:48:14 PM</td>
</tr>
<tr>
<td></td>
<td>SIM CHESS_LADY_28</td>
<td>WPO</td>
<td>11300</td>
<td>12/03/03</td>
<td>3:48:14 PM</td>
</tr>
<tr>
<td></td>
<td>SIM CHESS_TOWER</td>
<td>WPO</td>
<td>11300</td>
<td>12/03/03</td>
<td>3:48:15 PM</td>
</tr>
<tr>
<td></td>
<td>SIM_ZYK_T_28</td>
<td>WPO</td>
<td>11300</td>
<td>12/03/03</td>
<td>3:48:17 PM</td>
</tr>
<tr>
<td></td>
<td>SPO05</td>
<td>WPO</td>
<td>11300</td>
<td>12/03/03</td>
<td>3:09:49 AM</td>
</tr>
<tr>
<td></td>
<td>UTI</td>
<td>MPF</td>
<td>205</td>
<td>12/03/03</td>
<td>3:22:48 PM</td>
</tr>
<tr>
<td></td>
<td>TEMIC</td>
<td>WPO</td>
<td>11300</td>
<td>12/03/03</td>
<td>3:48:33 PM</td>
</tr>
</tbody>
</table>
14.1.1 NC memory

The complete NC working memory is displayed along with all workpieces, main programs and subprograms as well as dressing programs.

You can create further subdirectories here.

Procedure

1. Select the "Program manager" operating area.

2. Press the "NC" softkey.

14.1.2 Local drive

Workpieces, main and subprograms as well as dressing programs that are saved in the user memory of the CF card or on the local hard disk are displayed.

For archiving, you have the option of mapping the structure of the NC memory system or to create a separate archiving system.

You can create any number of subdirectories here, in which you can store any files (e.g. text files with notes).

Software options

To display the "Local drive" softkey, you require the "Additional HMI user memory on CF card of the NCU" option (not for SINUMERIK Operate on PCU50 or PC/PG).

Procedure

1. Select the "Program manager" operating area.

2. Press the "Local drive" softkey.

14.1.3 Creating an NC directory on the local drive

On the local drive, you have the option of mapping the directory structure of the NC memory. This also simplifies the search sequence.
Creating Directories

1. The local drive is selected.

2. Position the cursor on the main directory.


4. In the "Name" entry field, enter "mpf.dir", "spf.dir" and "wks.dir" and press the "OK" softkey. The directories "Part programs", "Subprograms" and "Workpieces" are created below the main directory.

14.1.4 USB drives

USB drives enable you to exchange data. For example, you can copy to the NC and execute programs that were created externally.

<table>
<thead>
<tr>
<th>NOTICE</th>
</tr>
</thead>
<tbody>
<tr>
<td>Interruption of operation</td>
</tr>
<tr>
<td>Direct execution from the USB FlashDrive is not recommended, because machining can be undeniably interrupted, therefore resulting in workpiece damage.</td>
</tr>
</tbody>
</table>

Partitioned USB-FlashDrive (only 840D sl and TCU)

If the USB-FlashDrive has several partitions, these are displayed in a tree structure as a subtree (01,02,...).

For EXTCALL calls, enter the partition (e.g. USB:/02/... or //ACTTCU/FRONT/02/... or //ACTTCU/FRONT,2/... or //TCU/TCU1/FRONT/02/...)

You can also configure any partition (e.g. //ACTTCU/FRONT,3).
14.1 Overview

Procedure

1. Select the "Program manager" operating area.

2. Press the "USB" softkey.

Note
The "USB" softkey can only be operated when a USB-FlashDrive is inserted in the front interface of the operator panel.

14.1.5 FTP drive

The FTP drive offers you the following options - to transfer data, e.g. part programs, between your control system and an external FTP server.

You have the option of archiving any files in the FTP server by creating new directories and subdirectories.

Note
Selecting a program / execution
It is not possible to select a program directly on the FTP drive, and change to execution in the "Machine" operating area.

Precondition
User name and password have been set up in the FTP server.

Procedure

1. Select the "Program manager" operating area.

2. Press the "FTP" softkey.

When selecting the FTP drive for the first time, a login window is displayed.
3. Enter the user name and password and press the "OK" softkey to log into the FTP server.
The content of the FTP server with its folders is displayed.

4. Press the "Log out" softkey after the required data processing has been completed.
The connection to the FTP server is disconnected. In order to reselect the FTP drive, you must log on again.
14.2 Opening and closing the program

To view a program in more detail or modify it, open the program in the editor.

With programs that are in the NCK memory, navigation is already possible when opening. The program blocks can only be edited when the program has been opened completely. You can follow the opening of the program in the dialog line.

With programs that are opened via local network, USB FlashDrive or network connections, navigation is only possible when the program has been opened completely. A progress message box is displayed when opening the program.

---

**Note**

**Channel changeover in the editor**

When opening the program, the editor is opened for the currently selected channel. This channel is used to simulate the program.

If you change over a channel in the editor, this does not influence the editor. Only when closing the editor do you change into the other channel.

---

**Procedure**

1. Select the "Program manager" operating area.

2. Select the desired storage location and position the cursor on the program that you would like to edit.

3. Press the "Open" softkey.
   
   - OR -
   
   Press the <INPUT> key.

4. Press the <Cursor right> key.

5. Double-click the program.

   The selected program is opened in the "Editor" operating area.

4. Now make the necessary program changes.

5. Press the "NC Select" softkey to switch to the "Machine" operating area and begin execution.

   When the program is running, the softkey is deactivated.
14.2 Opening and closing the program

Closing the program

Press the ">>" and "Exit" softkeys to close the program and editor again.

- OR -

If you are at the start of the first line of the program, press the <Cursor left> key to close the program and the editor.

To reopen a program you have exited with "Close", press the "Program" key.

Note

A program does not have to be closed in order for it to be executed.
14.3 Executing a program

When you select a program for execution, the control switches automatically to the "Machine" operating area.

Program selection

Select the workpieces (WPD), main programs (MPF) or subprograms (SPF) by placing the cursor on the desired program or workpiece.

For workpieces, the workpiece directory must contain a program with the same name. This program is automatically selected for execution (e.g. when you select the workpiece SHAFT.WPD, the main program SHAFT.MPF is automatically selected).

If an INI file of the same name exists (e.g. SHAFT.INI), it will be executed once at the first part program start after selection of the part program. Any additional INI files are executed in accordance with machine data MD11280 $MN_WPD_INI_MODE.

MD11280 $MN_WPD_INI_MODE=0:
The INI file with the same name as the selected workpiece is executed. For example, when you select SHAFT1.MPF, the SHAFT1.INI file is executed upon <CYCLE START>.

MD11280 $MN_WPD_INI_MODE=1:
All files of type SEA, GUD, RPA, UFR, PRO, TOA, TMA and CEC which have the same name as the selected main program are executed in the specified sequence. The main programs stored in a workpiece directory can be selected and processed by several channels.

Machine manufacturer
Please refer to the machine manufacturer’s specifications.

Procedure

1. Select the "Program manager" operating area.

2. Select the desired storage location, and position the cursor on the workpiece/program that you would like to execute.

3. Press the "Select" softkey.

The control switches automatically into the "Machine" operating area.

- OR -

If the selected program is already opened in the "Program" operating area, press the "Execute NC" softkey.

Press the <CYCLE START> key.

Machining of the workpiece is started.
Note

Program selection from external media

If you execute programs from an external drive (e.g. network drive), you require the "Execution from external storage (EES)" software option.
14.4 Creating a directory / program / job list

14.4.1 File and directory names

The following rules are to be observed when assigning names to files and directories:

- All letters are permissible (with the exception of umlauts, special characters, language-specific special characters, Asian or Cyrillic characters)
- All digits
- Underscores (_).
- The name can be a maximum of 24 characters long

Note
To avoid problems with Windows applications, do not use the following terms as program names or directory titles:

- CON, PRN, AUX, NUL
- COM1, COM2, COM3, COM4, COM5, COM6, COM7, COM8, COM9
- LPT1, LPT2, LPT3, LPT4, LPT5, LPT6, LPT7, LPT8, LPT9

Please note that these terms, including those with extensions (e.g. LPT1.MPF, CON.INI) can lead to problems if they are transferred to a Windows environment by copying, archiving or uploading, for example.

14.4.2 Creating a new directory

Directory structures help you to manage your program and data transparently. At all storage locations, you can create subdirectories for this purpose in a directory.

In a subdirectory, in turn, you can create programs and then create program blocks for them.

Note
Restrictions
- Directory names must end in .DIR or .WPD.
- The maximum name length is 28 characters including the extension.
- The maximum path length for nested workpieces, including all supplementary characters, is 100 characters.
- These names are automatically converted to upper-case letters.

This limitation does not apply for work on USB/network drives.
Procedure

1. Select the "Program manager" operating area.

2. Select the desired storage medium, i.e. a local or USB drive.

3. If you want to create a new directory in the local network, place the cursor on the topmost folder and press the "New" and "Directory" softkeys. The "New Directory" window opens.

4. Enter the desired directory name and press the "OK" softkey.

14.4.3 Creating a new workpiece

You can set up various types of files such as main programs, initialization files, tool offsets, etc. in a workpiece.

Note

Workpiece directories
You have the option of nesting tool directories. You must note that the length of the call line is restricted. You will be informed if the maximum number of characters is reached when entering the workpiece name.

Procedure

1. Select the "Program manager" operating area.

2. Select the desired storage location and position the cursor on the folder, in which you would like to create a workpiece.

3. Press the "New" softkey. The "New Workpiece" window appears.

4. If necessary, select a template if any are available.

5. Enter the desired workpiece name and press the "OK" softkey.
14.4 Creating a directory / program / job list

14.4.4 Creating a new G code program

You can create G code programs and then render G code blocks for them in a directory/workpiece.

Procedure

1. Select the "Program manager" operating area.

2. Select the desired storage location and position the cursor on the folder, in which you would like to store the program.

3. Press the "New" softkey.

   The "New G Code Program" window appears.

4. If necessary, select a template if any are available.

5. Select the file type (MPF or SPF).

   If you are in the NC memory and have selected the file "Subprograms" or "Part programs", you can create only one subprogram (SPF) or main program (MPF) respectively.

6. Enter the desired program name and press the "OK" softkey.

   The program type is appropriately specified.

14.4.5 Creating a new dressing program

You can create dressing programs and then render G code blocks for them in a directory/workpiece.
Procedure

1. Select the "Program manager" operating area.

2. Select a storage location and place the cursor on the "Dressing programs" folder.

3. Press the "New" softkey.

The "New Dressing Program" window opens.

The "DRS" file type is set as default.

4. Enter the desired program name and press the "OK" softkey.

14.4.6 Storing any new file

In each directory or subdirectory you can create a file in any format that you specify.

Note

File extensions

In the NC memory, the extension must have 3 characters, and DIR or WPD are not permitted.

In the NC memory, you can create the following file types under a workpiece using the "Any" softkey.
Procedure

1. Select the "Program manager" operating area.

2. Select the desired storage location and position the cursor on the folder in which you would like to create the file.


4. Select a file type from the "Type" selection field (for example, "Definitions GUD") and enter the name of the file to be created when you have selected a workpiece directory in the NC memory. The file automatically has the selected file format.

   - OR -

   Enter a name and file format for the file to be created (e.g. My_Text.txt).

5. Press the "OK" softkey.

14.4.7 Creating a job list

For every workpiece, you can create a job list for extended workpiece selection. In the job list, you specify instructions for program selection in different channels.

Syntax

The job list contains the SELECT instructions.

SELECT <program> CH=<channel number> [DISK]
The SELECT instruction selects a program for execution in a specific NC channel. The selected program must be loaded into the working memory of the NC. The DISK parameter enables the selection of external execution (CF card, USB data carrier, network drive).

- **<Program>**
  Absolute or relative path specification of the program to be selected.
  Examples:
  - //NC/WCS.DIR/SHAFT.WPD/SHAFT1.MPF
  - SHAFT2.MPF

- **<Channel number>**
  Number of the NC channel in which the program is to be selected.
  Example:
  CH=2

- **[DISK]**
  Optional parameter for programs that are not in the NC memory and are to be executed "externally".
  Example:
  SELECT //remote/myshare/shaft3.mpf CH=1 DISK

**Comment**

Comments are identified in the job list by ";" at the start of the line or by round brackets.

**Template**

You can select a template from Siemens or the machine manufacturer when creating a new job list.

**Executing a workpiece**

If the "Select" softkey is selected for a workpiece, the syntax of the associated job list is checked and then executed. The cursor can also be placed on the job list for selection.

**Procedure**

1. Select the "Program manager" operating area.
2. Press the "NC" softkey, and in directory "Workpieces" place the cursor on the program for which you wish to create a job list.
3. Press the "New" and "Any" softkeys.
   The "Any New Program" window opens.
4. Select entry "Job list JOB" from the "Type" selection field and enter a name and press the "OK" softkey.
14.4.8 Creating a program list

You can also enter programs in a program list that are then selected and executed from the PLC.

The program list may contain up to 100 entries.

Machine manufacturer

Please refer to the machine manufacturer's specifications.

Procedure

1. Select the "Program manager" operating area.

2. Press the menu forward key and the "Program list" softkey.
   The "Prog.-list" window opens.

3. Place the cursor in the desired line (program number).

4. Press the "Select program" softkey.
   The "Programs" window opens. The data tree of the NC memory with workpiece, part program and subprogram directory is displayed.

5. Place the cursor on the desired program and press the "OK" softkey.
   The selected program is inserted in the first line of the list together with its path.
   - OR -
   Enter the program name directly in the list.
   If you are making entries manually, check that the path is correct (e.g. //NC/WKS.DIR/MEINPROGRAMM.WPD/MEINPROGRAMM.MPF).

   //NC and the extension (.MPF) may be added automatically.
   With multi-channel machines, you can specify in which channel the program is to be selected.

6. To remove a program from the list, place the cursor on the appropriate line and press the "Delete" softkey.
   - OR -
   To delete all programs from the program list, press the "Delete all" softkey.
14.5 Creating templates

You can store your own templates to be used for creating part programs and workpieces. These templates provide the basic framework for further editing.

You can use them for any part programs or workpieces you have created.

Storage location for templates

The templates used to create part programs or workpieces are stored in the following directories:

HMI Data/Templates/Manufacturer/Part programs or Workpieces
HMI Data/Templates/User/Part programs or Workpieces

Procedure

1. Select the "Start-up" operating area.

2. Press the "System data" softkey.

3. Position the cursor on the file that you wish to store as a template and press the "Copy" softkey.

4. Select the directory in which you want to store the data - "Part programs" or "Workpieces" - and press the "Paste" softkey.

Stored templates can be selected when a part program or a workpiece is being created.
14.6 Searching directories and files

You have the possibility of searching in the Program Manager for certain directories and files.

**Note**

**Search with place holders**

The following place holders simplify the search:

- "*": Replaces any character string
- "?": Replaces any character

If you use place holders, only directories and files are found that correspond exactly to the search pattern.

Without place holders, directories and files are found that contain the search pattern at an arbitrary position.

**Search strategy**

The search is made in all of the selected directories and their subdirectories.

If the cursor is positioned on a file, then a search is made from the higher-level directory.

**Note**

**Searching in opened directories**

Open the closed directories for a successful search.

**Procedure**

1. Select the "Program Manager" operating area.

2. Select the storage location in which you wish to perform the search and then press the ">>" and "Search" softkeys.
   The "Find File" window opens.

3. Enter the desired search term in the "Text" field.
   Note: When searching for a file with place holders, enter the complete name with extension (e.g. DRILLING.MPF).

4. When required, activate the "Observe upper and lower case" checkbox.

5. Press the "OK" softkey to start the search.

6. If a corresponding directory or file is found, then it is marked.
7. Press the “Continue search” and “OK” softkeys if the directory or the file does not correspond to the required result.

- OR -
Press the “Cancel” softkey when you want to cancel the search.
14.7 Displaying the program in the Preview.

You can show the content on a program in a preview before you start editing.

Procedure

1. Select the "Program manager" operating area.

2. Select a storage location and place the cursor on the relevant program.

3. Press the ">>" and "Preview window" softkeys.
   The "Preview: ..." window opens.

4. Press the "Preview window" softkey again to close the window.
14.8 Selecting several directories/programs

You can select several files and directories for further processing. When you select a directory, all directories and files located beneath it are also selected.

---

**Note**

**Selected files**

If you have selected individual files in a directory, then this selection is canceled when the directory is closed.

If the complete directory with all of the files included in it are selected, then this selection is kept when closing the directory.

---

**Procedure**

1. Select the "Program manager" operating area.

2. Choose the desired storage location and position the cursor on the file or directory from which you would like your selection to start.

3. Press the "Select" softkey. The softkey is active.

4. Select the required directories/programs with the cursor keys or mouse.

5. Press the "Select" softkey again to deactivate the cursor keys.

---

**Canceling a selection**

By reselecting an element, the existing selection is canceled.
### Selecting via keys

<table>
<thead>
<tr>
<th>Key combination</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td>![SELECT]</td>
<td>Renders or expands a selection. You can only select individual elements.</td>
</tr>
<tr>
<td>![SHIFT]</td>
<td>Renders a consecutive selection.</td>
</tr>
<tr>
<td>![DEL]</td>
<td>A previously existing selection is canceled.</td>
</tr>
</tbody>
</table>

### Selecting with the mouse

<table>
<thead>
<tr>
<th>Key combination</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td>Left mouse</td>
<td>Click on element: The element is selected. A previously existing selection is canceled.</td>
</tr>
<tr>
<td>Left mouse +</td>
<td>Expand selection consecutively up to the next click.</td>
</tr>
<tr>
<td>![SHIFT] Pressed</td>
<td>Expand selection to individual elements by clicking. An existing selection will expand to include the element you clicked.</td>
</tr>
</tbody>
</table>

---

Managing programs

14.8 Selecting several directories/programs
14.9 Copying and pasting a directory/program

To create a new directory or program that is similar to an existing program, you can save time by copying the old directory or program and only changing selected programs or program blocks.

The capability of copying and pasting directories and programs can also be used to exchange data with other systems via USB/network drives (e.g. USB FlashDrive).

Copied files or directories can be pasted at a different location.

**Note**
You can only paste directories on local drives and on USB or network drives.

**Note**
**Write rights**
If the current directory is write-protected for the user, then the function is not listed.

**Note**
When you copy directories, any missing endings are added automatically.
All letters (except accented characters), numbers, and underscores are permitted when assigning names. The names are automatically converted to upper-case letters, and extra dots are converted to underscores.

**Example**
If the name is not changed when copying, a copy is created automatically:

MYPROGRAM.MPF is copied to MYPROGRAM__1.MPF. The next time it is copied, it is changed to MYPROGRAM__2.MPF, etc.

If the files MYPROGRAM.MPF, MYPROGRAM__1.MPF, and MYPROGRAM__3.MPF already exist in a directory, MYPROGRAM__2.MPF is created as the next copy of MYPROGRAM.MPF.

**Procedure**

1. Select the "Program Manager" operating area.
2. Choose the desired storage location and position the cursor on the file or directory which you would like to copy.
3. Press the "Copy" softkey.
4. Select the directory in which you want to paste your copied directory/program.
5. Press the "Paste" softkey.

An appropriate note is displayed if a directory/program with the same name exists in this directory. You are requested to assign a new name, otherwise the directory/program is assigned a name by the system.

If the name contains illegal characters or is too long, a prompt will appear for you to enter a permissible name.

6. Press the "OK" or "Overwrite all" softkey if you want to overwrite existing directories/programs.

- OR -
Press the "No overwriting" softkey if you do not want to overwrite already existing directories/programs.

- OR -
Press the "Skip" softkey if the copy operation is to be continued with the next file.

- OR -
Enter another name if you want to paste the directory/program under another name and press the "OK" softkey.

---

**Note**

**Copying files in the same directory**

You cannot copy files to the same directory. You must copy the file under a new name.
14.10 Deleting a program/directory

Delete programs or directories from time to time that you are no longer using to maintain a clearer overview of your data management. Back up the data beforehand, if necessary, on an external data medium (e.g. USB FlashDrive) or on a network drive.

Please note that when you delete a directory, all programs, tool data and zero point data and subdirectories that this directory contains are deleted.

Procedure

1. Select the "Program manager" operating area.

2. Choose the desired storage location and position the cursor on the file or directory that you would like to delete.

3. Press the ">>" and "Delete" softkeys.
   A prompt appears as to whether you really want to delete the file or directory.

4. Press the "OK" softkey to delete the program/directory.

   - OR -

   Press the "Cancel" softkey to cancel the process.
14.11 Changing file and directory properties

Information on directories and files can be displayed in the "Properties for ..." window. Information on the creation date is displayed near the file's path and name. You can change names.

Changing access rights

Access rights for execution, writing, listing and reading are displayed in the "Properties" window.

- Execute: Is used for the selection for execution
- Write: Controls the changing and deletion of a file or a directory

For NC files, you have the option to set the access rights from keyswitch 0 to the current access level, to be set separately for each file.

If an access level is higher than the current access level, it cannot be changed.

For external files (e.g. on a local drive), the access rights are displayed to you only if settings have been executed for these files by the machine manufacturer. They cannot be changed via the "Properties" window.

Settings for the access rights to directories and files

Via a configuration file and MD 51050, access rights of the directories and file types of the NC and user memory (local drive) can be changed and pre-assigned.

References

A detailed description of the configuration can be found in the following documentation:

SINUMERIK Operate Commissioning Manual

Procedure

1. Select the program manager.

2. Choose the desired storage location and position the cursor on the file or directory whose properties you want to display or change.

3. Press the ">>" and "Properties" softkeys. The "Properties from ..." window appears.
4. Enter any necessary changes.  
   **Note**: You can save changes via the user interface in the NC memory.

5. Press the "OK" softkey to save the changes.
14.12 Creating drives

14.12.1 Overview

Up to 21 connections to so-called logical drives (data carriers) can be configured. These drives can be accessed in the "Program manager" and "Startup" operating areas.

The following logical drives can be set up:

- USB interface
- Network drives
- CompactFlash card
- CompactFlash card of the NCU, only for SINUMERIK Operate in the NCU (for 840D sl)
- Local hard disk of the PCU, only for SINUMERIK Operate on the PCU (for 840D sl)

Software option – for 840D sl

In order to use the CompactFlash card as a data carrier, you require the option "Additional HMI user memory on CF card of NCU" (not for SINUMERIK Operate on PCU/PC).

Software option – for 828D

You will need the "Manage network drives" option to manage additional drives via Ethernet.

Note

The USB interfaces of the NCU are not available for SINUMERIK Operate and can therefore not be configured (for 840D sl).

14.12.2 Setting up drives

The "Set Up Drives" window is available in the "Start-up" operating area for configuring the softkeys in the Program Manager.

Note

Reserved softkeys

Softkeys 4, 7 and 16 are not available to be freely configured.

Machine manufacturer

Please refer to the machine manufacturer's specifications.
The created configuration data is stored in the "logdrive.ini" file. This file is located in the /user/sinumerik/hmi/cfg directory.

**General information**

<table>
<thead>
<tr>
<th>Entry</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td>Drives 1 - 24</td>
<td></td>
</tr>
<tr>
<td>Type</td>
<td></td>
</tr>
<tr>
<td>No drive</td>
<td>No drive defined</td>
</tr>
<tr>
<td>NC program memory</td>
<td>Access to the NC memory</td>
</tr>
<tr>
<td>USB local</td>
<td>Access to the USB interface of the active op-</td>
</tr>
<tr>
<td></td>
<td>erator unit</td>
</tr>
<tr>
<td>USB global</td>
<td>All of the TCUs in the plant network can ac-</td>
</tr>
<tr>
<td></td>
<td>cess the USB memory medium.</td>
</tr>
<tr>
<td>NW Windows</td>
<td>Network drive in Windows systems.</td>
</tr>
<tr>
<td>NW Linux</td>
<td>Network drive in Linux systems.</td>
</tr>
<tr>
<td>Local drive</td>
<td>Local drive. Hard disk or user memory on the</td>
</tr>
<tr>
<td></td>
<td>CompactFlash card.</td>
</tr>
<tr>
<td>FTP</td>
<td>Access to an external FTP server.</td>
</tr>
<tr>
<td></td>
<td>The drive cannot be used as global part pro-</td>
</tr>
<tr>
<td></td>
<td>gram memory.</td>
</tr>
<tr>
<td>User cycles</td>
<td>Access to the user cycle directory of the Com-</td>
</tr>
<tr>
<td></td>
<td>pactFlash card</td>
</tr>
<tr>
<td>Manufacturer cycles</td>
<td>Access to the manufacturer cycle directory of</td>
</tr>
<tr>
<td></td>
<td>the CompactFlash card.</td>
</tr>
<tr>
<td>Drive Windows</td>
<td>Access to a local PCU/PC directory.</td>
</tr>
</tbody>
</table>

**Specifications for USB**

<table>
<thead>
<tr>
<th>Entry</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Device</td>
<td>Names of the TCU to which the USB storage medium is connected, e.g. tcu1.</td>
</tr>
<tr>
<td></td>
<td>The NCU must already know the TCU name.</td>
</tr>
<tr>
<td>Connection</td>
<td>USB interface that is located at the front of the operator panel.</td>
</tr>
<tr>
<td>Front</td>
<td>X203/X204</td>
</tr>
<tr>
<td>X203/X204</td>
<td>USB interface X203/X204 that is located at the rear of the operator panel.</td>
</tr>
<tr>
<td>X61/X62</td>
<td>For SIMATIC Thin Client, the USB interfaces are X61 and X62.</td>
</tr>
<tr>
<td>X212/X213</td>
<td>TCU20.2/20.3</td>
</tr>
<tr>
<td>X20</td>
<td>OP 08T</td>
</tr>
<tr>
<td>X60.P1/P2/P3/P4</td>
<td>PCU</td>
</tr>
<tr>
<td>Symbolic</td>
<td>Symbolic drive name.</td>
</tr>
</tbody>
</table>

**Additional parameters under Details**
### Specifications for local drives

<table>
<thead>
<tr>
<th>Entry</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Partition</td>
<td>Partition number on the USB storage medium, e.g. 1 or all. If a USB hub is being used, then specify the USB port of the hub.</td>
</tr>
<tr>
<td>USB path</td>
<td>Path to the USB hub. <strong>Note:</strong> This value is not currently evaluated.</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Entry</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Symbolic</td>
<td>Symbolic drive name. Assignment of the names under Details</td>
</tr>
</tbody>
</table>

**Additional parameters under Details**

<table>
<thead>
<tr>
<th>Use drive as:</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>LOCAL_DRIVE</td>
<td>The activation of the checkbox assigns the symbolic name to the drive. If an assignment exists already for the drive, no change can be made. All checkboxes are active as preassignment.</td>
</tr>
<tr>
<td>CF_CARD</td>
<td></td>
</tr>
<tr>
<td>SYS_DRIVE</td>
<td></td>
</tr>
</tbody>
</table>

### Specifications for network drives

<table>
<thead>
<tr>
<th>Entry</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Computer name</td>
<td>Logical name of the server or the IP address.</td>
</tr>
<tr>
<td>Release name</td>
<td>Name, under which the network drive was released. Only for network drives in Windows systems.</td>
</tr>
<tr>
<td>Path</td>
<td>Start directory. The path is specified relative to the released directory.</td>
</tr>
<tr>
<td>User name</td>
<td>Enter the user name and the corresponding password for which the directory is enabled on the server. The password is displayed in encoded form as string of &quot;*&quot; characters and is stored in the &quot;logdrive.ini&quot; file.</td>
</tr>
<tr>
<td>Password</td>
<td></td>
</tr>
<tr>
<td>Symbolic</td>
<td>Symbolic drive name. Maximum 12 characters can be entered (letters, digits, underscore). The names NC, GDIR and FTP are reserved. They are also used to label the softkey if a softkey text is not specified.</td>
</tr>
</tbody>
</table>

**Managing programs**

14.12 Creating drives
### Specifications for FTP

<table>
<thead>
<tr>
<th>Entry</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Computer name</td>
<td>Logical name of the FTP server or the IP address.</td>
</tr>
<tr>
<td>Path</td>
<td>Start directory on the FTP server. The path is specified relative to the home directory.</td>
</tr>
<tr>
<td>User name</td>
<td>User names and the associated password for login to the FTP server.</td>
</tr>
<tr>
<td>Password</td>
<td>The password is displayed in encoded form as a string of &quot;*&quot; characters and is stored in the &quot;logdrive.ini&quot; file.</td>
</tr>
</tbody>
</table>

**Additional parameters under Details**

| Port       | Interface for the FTP connection. The default port is 21.                 |
|Disconnect  | After a disconnect timeout, the FTP connection is disconnected. The timeout can be between 1 and 150 s. 10 s is the default setting. |

### Additional specifications when using the "Execution from external storage (EES)" function

**Machine manufacturer**

Please refer to the machine manufacturer's specifications.

<table>
<thead>
<tr>
<th>Entry</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Enable drive</td>
<td>Only for &quot;Drive Windows (PCU)&quot; type. The drive is enabled in the network. A user name is required. The checkbox must be activated if the local drive serves as global part program memory.</td>
</tr>
<tr>
<td>Global part program memory</td>
<td>Only for local drives, network drives and global USB drives. The checkbox indicates that all system nodes have access to the configured logical drive. The nodes can directly execute part programs from the drive. The setting can only be changed under Details.</td>
</tr>
<tr>
<td>Use this drive for EES program execution</td>
<td>Only for USB drives. Allows a local USB storage medium to be used to execute programs using EES.</td>
</tr>
</tbody>
</table>

**Additional parameters under Details**
## Managing programs

### 14.12 Creating drives

<table>
<thead>
<tr>
<th>Entry</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Windows user name</td>
<td>Only for USB drives, local drives and local directories</td>
</tr>
<tr>
<td>Windows password</td>
<td>User name and the associated password for release of the configured drive. The specifications from the “Global Settings” window are used as default setting.</td>
</tr>
<tr>
<td>Global part program memory</td>
<td>Only for local drives, network drives and global USB drives</td>
</tr>
<tr>
<td></td>
<td>The checkbox defines whether all system nodes have access to the configured logical drive. Only one drive can be selected as global part program memory (GDIR). If another drive has already been defined as GDIR and the checkbox is activated, the original setting is deleted.</td>
</tr>
</tbody>
</table>

### Specifications for the configured softkey

<table>
<thead>
<tr>
<th>Entry</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Access level</td>
<td>Assign access rights to the connections: From access level 7 (keyswitch position 0) to access level 1 (manufacturer). The particular assigned access level applies to all operating areas.</td>
</tr>
<tr>
<td>Softkey text</td>
<td>Two lines are available as labeling text for the softkey. %n is accepted as a line separator. If the first line is too long, then a line break is automatically inserted. If a space is present, it is taken as a line separator. For language-dependent softkey texts, the text ID is entered, which is used to search in the text file. If nothing is specified in the entry field, then the symbolic drive name is used as softkey text.</td>
</tr>
<tr>
<td>Softkey icon</td>
<td>No icon is displayed on the softkey.</td>
</tr>
<tr>
<td>sk_usb_front.png</td>
<td>File names of the icon displayed on the softkey.</td>
</tr>
<tr>
<td>sk_local_drive.png</td>
<td></td>
</tr>
<tr>
<td>sk_network_drive_ftp.png</td>
<td></td>
</tr>
<tr>
<td>Text file</td>
<td>slpmdialog</td>
</tr>
<tr>
<td>Text context</td>
<td>SIPmDialog</td>
</tr>
</tbody>
</table>
Procedure

1. Select the "Start-up" operating area.

2. Press the "HMI" and "Log. drive" softkeys.
The "Set Up Drives" window opens.

3. Select the softkey that you want to configure.

4. To configure softkeys 9 to 16 or softkeys 17 to 24, click the ">> level" softkey.

5. To allow entry fields to be edited, press the "Change" softkey.

6. Select the data for the corresponding drive or enter the necessary data.

7. Press the "Details" softkey if you want to enter additional parameters.
Press the "Details" softkey to return to the "Set Up Drives" window.

8. Press the "OK" softkey.
The entries are checked.
A window with the appropriate message opens if the data is incomplete or incorrect. Acknowledge the message with "OK" softkey.

9. If you press the "Cancel" softkey, then all of the data that has not been activated is rejected.

9. Restart the control in order to activate the configuration and to obtain the softkeys in the "Program Manager" operating area.

Entering the default settings for drive release

Note
This function is available only on Windows systems when the "Execution from external storage (EES)" software option is activated.

1. Select the "Start-up" operating area.

2. Press the "HMI" and "Log. drive" softkeys.
The "Set Up Drives" window opens.
3. Press the "Glob. settings" softkey.

4. Enter the user name and the associated password for the configured drives to be released.

5. Press the "OK" softkey.

   The specifications are transferred as default setting for the Windows release.

   If you press the "Cancel" softkey, then all of the data that has not been activated is rejected.
14.13 EXTCALL

The EXTCALL command can be used to access files on a local drive, USB data carriers or network drives from a part program.

The programmer can set the source directory with the setting data SD $SC42700 EXT_PROG_PATH and then specify the file name of the subprogram to be loaded with the EXTCALL command.

Supplementary conditions

The following supplementary conditions must be taken into account with EXTCALL calls:

- You can only call files with the MPF or SPF extension via EXTCALL from a network drive.
- The files and paths must comply with the NCK naming conventions (max. 25 characters for the name, 3 characters for the identifier).
- A program is found on a network drive with the EXTCALL command if
  - with SD $SC42700 EXT_PROG_PATH the search path refers to the network drive or a directory contained on the network drive. The program must be stored directly on that level, no subdirectories are searched.
  - without SD $SC42700 the correct location of the program is specified in the EXTCALL call itself by means of a fully qualified path that can also point to a subdirectory of the network drive.
- For programs that were generated on external storage media (Windows system) observe upper- and lower-case syntax.

Note

Maximum path length for EXTCALL

The path length must not exceed 112 characters. The path comprises the contents of the setting data (SD $SC42700) and the path data for EXTCALL call from the part program.

Examples of EXTCALL calls

The setting data can be used to perform a targeted search for the program.

- Call of USB drive on TCU (USB storage device on interface X203), if SD42700 is empty: e.g. EXTCALL "/TCU/TCU1/X203,1/TEST.SPF"
- OR -
  Call of USB drive on TCU (USB storage device on interface X203), if SD42700 "/TCU/TCU1/X203,1" contains: EXTCALL "TEST.SPF"

- Call of the USB front connection (USB-FlashDrive), if SD $SC 42700 is empty: e.g. EXTCALL "/ACTTCU/Front,1/TEST.SPF"
- OR -
  Call of the USB front connection (USB-FlashDrive), if SD42700 "/ACTTCU/Front,1" contains: EXTCALL "TEST.SPF"
• Call of network drive, if SD42700 is empty: e.g. EXTCALL "/computer name/enabled drive/TEST.SPF"
  - OR -
  Call of the network drive, if SD $SC42700 "/Computer name/enabled drive" contains:
  EXTCALL "TEST.SPF"

• Use of the HMI user memory (local drive):
  – On the local drive, you have created the directories part programs (mpf.dir),
    subprograms (spf.dir) and workpieces (wks.dir) with the respective workpiece
    directories (.wpd):
    SD42700 is empty: EXTCALL "TEST.SPF"
    The same search sequence is used on the CompactFlash card as in the NCK part
    program memory.
  – On the local drive, you have created your own directory (e.g. my.dir):
    Specification of the complete path: e.g. EXTCALL "/user/sinumerik/data/prog/my.dir/
    TEST.SPF"
    A search is performed for the specified file.

**Note**

Abbreviations for local drive, CompactFlash card and USB front connection

As abbreviation for the local drive, the CompactFlash card and the USB front connection
you can use the abbreviation LOCAL_DRIVE:, CF_CARD: and USB: (e.g. EXTCALL
"LOCAL_DRIVE:/spf.dir/TEST.SPF").

Alternatively, you can also use the abbreviations CF_Card and LOCAL_DRIVE.

---

**Software options**

To display the "Local drive" softkey, you require option "Additional HMI user
memory on CF card of the NCU" (not for SINUMERIK Operate on PCU50 / PC).

---

**NOTICE**

Possible interruption when executing from USB FlashDrive

Direct execution from a USB-FlashDrive is not recommended.

There is no protection against contact problems, falling out, breakage through knocking or
unintentional removal of the USB-FlashDrive during operation.

Disconnecting it during operation will result in immediate stopping of the machining and, thus,
to the workpiece being damaged.

---

**Machine manufacturer**

Processing EXTCALL calls can be enabled and disabled.

Please refer to the machine manufacturer's specifications.
14.14 Execution from external storage (EES)

The "Execution from external storage" function allows you to directly execute any size of part program from an appropriately configured drive. The behavior is the same as that for execution from the NC part program memory without the restrictions that apply to "EXTCALL".

Software option
You require the "CNC user memory extended" software option in order to use this function in the user memory (100 MB) of the CompactFlash card.

Software option
In order to use this function without restrictions, e.g. for a network drive or a USB drive, you require the "Execution from external storage (EES)" software option.

Note
Not possible to teach-in program
When an EES program has been selected, it is not possible to teach-in programs.

Machine manufacturer
Please observe the information provided by the machine manufacturer.

You have the option of processing the G code programs saved on the configured external drives as usual in the editor.

When executing the G code programs, you obtain a current block display, as usual. You can edit the programs directly in the Reset state.

In addition to the current block display, you can also show a basic block display. You can make corrections with the "Program correction" function, as usual.
14.15 Backing up data

14.15.1 Generating an archive in the Program Manager

You have the option of archiving individual files from the NC memory and the local drive.

Archive formats

You have the option of saving your archive in the binary and punched tape format.

Save target

The archive folder of the system data in the "Startup" operating area as well as USB and network drives are available as save target.

Procedure

1. Select the "Program Manager" operating area.

2. Select the storage location for the file/files to be archived.

3. In the directories, select the required file from which you want to create an archive.
   - OR -
   If you want to back up several files or directories, press the "Select" softkey.
   Make the selection using the cursor keys or mouse.

4. Press the ">>" and "Archive" softkeys.

5. Press the "Generate archive" softkey.
   The "Generate Archive: Select archiving" window opens.

6. Position the cursor to the required storage location, press the "Search" softkey, enter the required search term in the search dialog and press the "OK" softkey if you wish to search for a specific directory or subdirectory.
   **Note:** The place holders "***" (for any character string) and "??" (for any character) make it easier for you to perform a search.
   - OR -
   Select the required storage location, press the "New directory" softkey, enter the required name in the "New directory" window and press the "OK" softkey to create a directory.
7. Press "OK".
The "Generate Archive: Name" window opens.

9. Select the format (e.g. archive ARC (binary format) for 840 sl or archive ARD for 828D), enter the desired name and press the "OK" softkey.
A message informs you if archiving was successful.

14.15.2 Generating an archive via the system data

If you only want to backup specific data, then you can select the desired files directly from the data tree and generate an archive.

**Archive formats**
You have the option of saving your archive in the binary and punched tape format.
You can display the content of the selected files (XML, ini, hsp, syf files, programs) using a preview.
You can display information about the file, such as path, name, date of creation and change, in a Properties window.

**Precondition**
The access rights depend on the relevant areas and range from protection level 7 (key switch position 0) to protection level 2 (password: Service).

**Storage locations**
- CompactFlash card under
  /user/sinumerik/data/archive, or
  /oem/sinumerik/data/archive
- All configured logical drives (USB, network drives)

**Software option**
In order to save archives on the CompactFlash Card in the user area you require the "Additional HMI user memory on CF card of NCU" option.

**NOTICE**

Possible data loss when using USB flash drives
USB-FlashDrives are not suitable as persistent memory media.
Procedure

1. Select the "Startup" operating area.

2. Press the "System data" softkey.
   The data tree opens.

3. In the data tree, select the required files from which you want to generate an archive.
   - OR -
   If you want to back up several files or directories, press the "Select" softkey.
   Make the selection using the cursor keys or mouse.

4. If you press the ">>" softkey, further softkeys are displayed on the vertical bar.

5. Press the "Preview window" softkey.
   The contents of the selected file are displayed in a small window.
   Press the "Preview window" softkey again to close the window.

6. Press the "Properties" softkey.
   Information about the selected file is displayed in a small window.
   Press the "OK" softkey to close the window.

7. Press the "Search" softkey.
   Enter the required search term in the search dialog and press the "OK" softkey if you wish to search for a specific directory or subdirectory.
   **Note:** The place holders "*" (for any character string) and "?" (for any character) make it easier for you to perform a search.

8. Press the "Archive" and "Generate archive" softkeys.
   The "Generate Archive: Select Storage Location" window opens.
   The "Archive" folder with the subfolders "User" and "Manufacturer" as well as the storage media (e.g. USB) are displayed.

9. Select the required location for archiving and press the "New directory" softkey to create a suitable subdirectory.
   The "New Directory" window opens.

10. Enter the required name and press the "OK" softkey.
    The directory is created below the selected folder.

11. Press the "OK" softkey.
    The "Generate Archive: Name" window opens.
12. Select the format (e.g. archive ARC (binary format) for 840D sl or archive ARD for 828D), enter the desired name and press the "OK" softkey to archive the file/files.

A message informs you if archiving was successful.

13. Press the "OK" softkey to confirm the message and end the archiving operation.

An archive file in the .ARC (840D sl) or .ARD (828D) format type is created in the selected directory.

14.15.3 Reading in an archive in the Program Manager

In the "Program Manager" operating area, you have the option of reading in archives from the archive folder of the system data as well as from configured USB and network drives.

**Software option**

In order to read-in user archives in the "Program Manager" operating area, you require the option "Additional HMI user memory on CF Card of NCU" (not for 840D sl / SINUMERIK Operate on PCU50 / PC).

**Procedure**

1. Select the "Program Manager" operating area.

2. Press the "Archive" and "Read in archive" softkeys.

   The "Read in archive: Select archive" window opens.

3. Select the archive storage location and position the cursor on the required archive.

   **Note:** When the option is not set, the folder for user archives is only displayed if the folder contains at least one archive.

   - OR -

   Press the "Search" softkey and in the search dialog, enter the name of the archive file with file extension (*.arc) for 840D sl or with file extension (*.ard) for 828D if you wish to search for a specific archive and press the "OK" softkey.

4. Press the "OK" or "Overwrite all" softkey to overwrite existing files.

...
5. Press the "Cancel" softkey to cancel the read-in process.

14.15.4 Read in archive from system data

If you want to read in a specific archive, you can select this directly from the data tree.

Procedure

1. Select the "Startup" operating area.
2. Press the "System data" softkey.
3. In the data tree below the "Archive" directory, in the "User" folder, select the file that you wish to read in.
4. Press the "Read in" softkey.
5. Press the "OK" or "Overwrite all" softkey to overwrite existing files.

- OR -
Press the "Do not overwrite" softkey if you do not want to overwrite already existing files.

- OR -
Press the "Skip" softkey if the read-in operation is to be continued with the next file.

The "Read In Archive" window opens and a progress message box appears for the read-in process.

You will then obtain a "Read error log for archive" in which the skipped or overwritten files are listed.
You will then obtain a "Read error log for archive" in which the skipped or overwritten files are listed.

6. Press the "Cancel" softkey to cancel the read-in process.
14.16 Setup data

14.16.1 Backing up setup data

Apart from programs, you can also save tool data and zero point settings.

You can use this option, for example, to backup the required tools and zero point data for a specific G code program. If you want to execute this program at a later point in time, you will then have quick access to the relevant settings.

Even tool data that you have measured on an external tool setting station can be copied easily into the tool management system using this option.

---

**Note**

**Backing up setup data from part programs**

Setup data from part programs can only be backed up if they have been saved in the "Workpieces" directory.

For part programs, which are located in the "Part programs" directory, "Save setup data" is not listed.

---

## Backing up data

<table>
<thead>
<tr>
<th>Data</th>
<th>Tool data</th>
<th>Magazine assignment</th>
<th>Zero points</th>
<th>Basic zero points</th>
<th>Directory</th>
<th>File name</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>• No</td>
<td>• Yes</td>
<td>• No</td>
<td>• No</td>
<td>The directory is displayed, in which the selected program is located.</td>
<td>Here you have the option of changing the suggested file names.</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>• Complete tool list</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>• Yes</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>• No</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>• The selection box &quot;Basis zero point&quot; is hidden</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>• All</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

---

**Note**

**Magazine assignment**

You can only read out the magazine assignments if your system provides support for loading and unloading tool data to and from the magazine.
Procedure

1. Select the "Program Manager" operating area.

2. Position the cursor on the program whose tool and zero point data you wish to back up.

3. Press the ">>" and "Archive" softkeys.

4. Press the "Setup data" softkey.
   The "Backup setup data" window opens.

5. Select the data you want to back up.

6. When required, change the specified name of the originally selected program here in the "File name" field.

7. Press the "OK" softkey.
   The setup data will be set up in the same directory in which the selected program is stored.
   The file is automatically saved as INI file.

Note

Program selection
If a main program as well as an INI file with the same name are in a directory, when selecting the main program, initially, the INI file is automatically started. In this way, unwanted tool data can be changed.

Machine manufacturer
Please refer to the machine manufacturer's specifications.

14.16.2 Reading-in set-up data

When reading-in, select which of the backed-up data you require:

- Tool data
- Magazine assignment
• Zero points
• Basic zero point

**Tool data**

Depending on which data you have selected, the system behaves as follows:

- **Complete tool list**
  First, all tool management data are deleted and then the saved data are imported.

- **All tool data used in the program**
  If at least one of the tools to be read in already exists in the tool management system, you can choose between the following options.

  - **Overwrite all**
    Select the "Replace all" softkey to import all tool data. Any existing tools will now be overwritten without a warning prompt.
    - OR -
    Press the "Do not overwrite" softkey if existing tools must not be overwritten. Already existing tools are skipped, without you receiving any queries.
    - OR -
    Press the "Skip" softkey if already existing tools are not to be overwritten. For an already existing tool, you receive a query.

**Selecting loading point**

For a magazine, if more than one loading point was set-up, using the "Select loading point" softkey, you have the option of opening a window in which you can assign a loading point to a magazine.

**Procedure**

1. Select the "Program Manager" operating area.

2. Position the cursor on the file with the backed-up tool and zero point data (*.INI) that you wish to re-import.

3. Press the <Cursor right> key
   - OR -
   Double-click the file.
   The "Read-in setup data" window opens.
4. Select the data (e.g. magazine assignment) that you wish to read-in.

5. Press the "OK" softkey.
14.17 **Backing up parameters**

In addition to the programs, you can also save R-parameters and global user variables. You can use this option, for example, to back up the required arithmetic parameters and user variables for a specific program. If you want to execute this program at a later point in time, you will then have quick access to the relevant data.

**Note**

**Backing up parameters from part programs**

Parameters from part programs can only be backed up if they have been saved in the "Workpieces" directory.

For part programs that are located in the "Part programs" or "Subprograms" directory, "Save parameters" is not listed.

---

**Backing up data**

Which data is offered for backup depends on the machine configuration:

<table>
<thead>
<tr>
<th>Data</th>
<th>Options</th>
</tr>
</thead>
<tbody>
<tr>
<td>R parameters</td>
<td>No, Yes - all channel-specific arithmetic parameters</td>
</tr>
<tr>
<td>Global R parameters</td>
<td>No, Yes - all global arithmetic parameters</td>
</tr>
<tr>
<td>UGUD parameters</td>
<td>No, Yes - all channel-specific variables of the user</td>
</tr>
<tr>
<td>Global UGUD parameters</td>
<td>No, Yes - all global variables of the user</td>
</tr>
<tr>
<td>MGUD parameters</td>
<td>No, Yes - all channel-specific variables of the machine manufacturer</td>
</tr>
<tr>
<td>Global MGUD parameters</td>
<td>No, Yes - all global variables of the machine manufacturer</td>
</tr>
</tbody>
</table>

Directory: The directory is displayed, in which the selected program is located.

File name: Here you have the option of changing the suggested file names.

For multi-channel machines, the parameters of the active channel are always backed up.

**Job lists**

If you select Back up parameters for a job list, the parameters of all programs that it contains are backed up.

The name of the job list does not agree with the names of the programs it contains. To nevertheless permit the unique assignment of the parameter files, they are always assigned the same name as the associated program. You cannot change these file names.
**Procedure**

1. Select the "Program Manager" operating area.

2. Select the drive on which the program is saved.

3. Position the cursor on the program whose parameters you want to back up.

4. Press the ">>" and "Archive" softkeys.

5. Press the "Save parameters" softkey.
   The "Save parameters" window appears.

6. Select the data you want to back up.

7. Press the <CHANNEL> key or click on the channel display if you want to change the active channel.

- OR -

8. Change the specified name of the originally selected program in the "File name" field as required.

9. Press the "OK" softkey.
   The parameters are saved in the same directory in which the selected program is stored.
   The R-parameters (*.RPA) and the user variables (*.GUD) are saved in separate files.

**Note**

**Program selection**

If a directory contains a main program as well as an RPA file or a GUD file with the same name, initially these files are automatically started when the main program is selected. Tool data or parameters may accidentally be changed as a result.
Managing programs

14.17 Backing up parameters

Machine manufacturer
Please refer to the machine manufacturer’s instructions.
14.18 V24

14.18.1 Reading-in and reading-out archives via a serial interface

You have the option of reading out and reading in archives in the "Program manager" operating area as well as in the "Startup" operating area via the V24 serial interface.

Availability of the V24 serial interface

If you want to change the availability of the V24 interface, you can adjust the following parameters in file "slpmconfig.ini":

<table>
<thead>
<tr>
<th>Parameters</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>[V24]</td>
<td>Describes the section in which the relevant setting parameters are located.</td>
</tr>
<tr>
<td>useV24</td>
<td>Setting for the availability of the V24 serial interface</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>true</td>
<td>Interface and softkeys are available (default)</td>
</tr>
<tr>
<td>false</td>
<td>Interface and softkeys are not available</td>
</tr>
</tbody>
</table>

Storage of file "slpmconfig.ini"

The template of the file "slpmconfig.ini" for SINUMERIK Operate is stored in the following directory:

<Installation path>/siemens/sinumerik/hmi/template/cfg

Copy the file to one of the following directories:

<Installation path>/user/sinumerik/hmi/cfg
<Installation path>/oem/sinumerik/hmi/cfg

Note

If you want to achieve a better overview of the changes you have made yourself, simply delete the unchanged parameters from the file copy "slpmconfig.ini".

Reading-out archives

The files to be sent (directories, individual files) are zipped in an archive (*.arc). If you send an archive (*.arc), this is sent directly without being additionally zipped. If you have selected an archive (*.arc) together with an additional file (e.g. directory), then these are zipped into a new archive and are then sent.
Reading-in archives

Use interface V24 if you want to read in archives. They are transferred and then subsequently unzipped.

Note
Reading in commissioning archives
When you read in a commissioning archive via the V24 interface, then this is immediately activated.

Externally processing the punched tape format
If you wish to externally process an archive, then generate this in the punched tape format.

Procedure

1. Select the "Program manager" operating area, and press the "NC" or "Local. drive" softkey.

- OR -
Select the "Startup" operating area and press the "System data" softkey.

Reading-out archives

2. Select the directories or the files that you wish to send to V24.

3. Press the ">>" and "Archive" softkeys.

4. Press the "V24 send" softkey.

- OR -
Press the "V24 receive" softkey if you wish to read-in files via V24.

### 14.18.2 Setting V24 in the program manager

<table>
<thead>
<tr>
<th>V24 setting</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td>Protocol</td>
<td>The following protocols are supported for transfer via the V24 interface:</td>
</tr>
<tr>
<td></td>
<td>• RTS/CTS (default setting)</td>
</tr>
<tr>
<td></td>
<td>• Xon/Xoff</td>
</tr>
<tr>
<td>Transfer</td>
<td>Data transfer using a secured protocol (ZMODEM protocol):</td>
</tr>
<tr>
<td></td>
<td>• Normal (default setting)</td>
</tr>
<tr>
<td></td>
<td>• secure</td>
</tr>
<tr>
<td></td>
<td>For the selected interface, secure data transfer is set in conjunction</td>
</tr>
<tr>
<td></td>
<td>with handshake RTS/CTS.</td>
</tr>
<tr>
<td>Baud rate</td>
<td>Transfer rate: up to 115 kbaud data transfer rate. The baud rate that can</td>
</tr>
<tr>
<td></td>
<td>be used depends on the connected device, the cable length and the</td>
</tr>
<tr>
<td></td>
<td>general electrical conditions.</td>
</tr>
<tr>
<td></td>
<td>• 110</td>
</tr>
<tr>
<td></td>
<td>• ....</td>
</tr>
<tr>
<td></td>
<td>• 19200 (default)</td>
</tr>
<tr>
<td></td>
<td>• ...</td>
</tr>
<tr>
<td></td>
<td>• 115200</td>
</tr>
<tr>
<td>Archive format</td>
<td>• Punched tape format (default setting)</td>
</tr>
<tr>
<td></td>
<td>• Binary format (PC format)</td>
</tr>
<tr>
<td>V24 settings (details)</td>
<td></td>
</tr>
<tr>
<td>Interface</td>
<td>• COM1</td>
</tr>
<tr>
<td>Parity</td>
<td>Parity bits are used for error detection: The parity bits are added to</td>
</tr>
<tr>
<td></td>
<td>the coded characters to make the number of positions set to &quot;1&quot; an</td>
</tr>
<tr>
<td></td>
<td>uneven number (uneven parity) or to an even number (even parity).</td>
</tr>
<tr>
<td></td>
<td>• None (default setting)</td>
</tr>
<tr>
<td></td>
<td>• Odd</td>
</tr>
<tr>
<td></td>
<td>• Even</td>
</tr>
<tr>
<td>Stop bits</td>
<td>Number of stop bits for asynchronous data transfer.</td>
</tr>
<tr>
<td></td>
<td>• 1 (default)</td>
</tr>
<tr>
<td></td>
<td>• 2</td>
</tr>
<tr>
<td>Data bits</td>
<td>Number of data bits for asynchronous data transfer.</td>
</tr>
<tr>
<td></td>
<td>• 5 bits</td>
</tr>
<tr>
<td></td>
<td>• ...</td>
</tr>
<tr>
<td></td>
<td>• 8 bits (default setting)</td>
</tr>
<tr>
<td>XON (hex)</td>
<td>Only with protocol: Xon/Xoff</td>
</tr>
<tr>
<td>XOFOFF (hex)</td>
<td>Only with protocol: Xon/Xoff</td>
</tr>
</tbody>
</table>
### V24 setting

<table>
<thead>
<tr>
<th>V24 setting</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td>Wait for XON for start receive V24</td>
<td>Only with protocol: Xon/Xoff</td>
</tr>
<tr>
<td>End of data transfer (hex)</td>
<td>Only for punched tape format</td>
</tr>
<tr>
<td></td>
<td>Stop with end of data transfer character</td>
</tr>
<tr>
<td></td>
<td>The default setting for the end of data transfer character is (HEX) 1A</td>
</tr>
<tr>
<td>Time monitoring (sec)</td>
<td>Time monitoring</td>
</tr>
<tr>
<td></td>
<td>For data transfer problems or at the end of data transfer (without end of data transfer character) data transfer is interrupted after the specified number of seconds.</td>
</tr>
<tr>
<td></td>
<td>The time monitoring is controlled by a time generator (clock) that is started with the first character and is reset with each transferred character.</td>
</tr>
<tr>
<td></td>
<td>The time monitoring can be set (seconds).</td>
</tr>
</tbody>
</table>

### Procedure

1. Select the "Program Manager" operating area.
2. Press softkey "NC" or "Local Drive".
3. Press the ">>" and "Archive" softkeys.
4. Select the "V24 settings" softkey.
   Window "Interface: V24" is opened.
5. The interface settings are displayed.
6. Press the "Details" softkey if you wish to view and process additional settings for the interface.
15.1 Displaying messages

PLC and part program messages may be issued during machining.
These message will not interrupt the program execution. Messages provide information with regard to a certain behavior of the cycles and with regard to the progress of machining and are usually kept beyond a machining step or until the end of the cycle.

Overview of messages
You can display all issued messages.
The message overview contains the following information:

- Date
- Message number
  is only displayed for PLC messages
- Message text

Procedure

1. Select the "Diagnostics" operating area.
2. Press the "Messages" softkey.
The "Messages" window appears.
15.2 Displaying alarms

If the machine develops a fault in operation, an alarm is generated and machining is possibly interrupted.

The error text that is displayed together with the alarm number gives you more detailed information on the error cause.

You have the possibility of saving all of the relevant diagnostic data to a ZIP file, which you can subsequently send to the hotline for analysis.

⚠️ CAUTION

Dangers for persons and machines

Carefully check the system, based on the description of the active alarm(s). Resolve the cause of the alarms. Then acknowledge the alarms in the specified way.

Failure to observe this warning will place your machine, workpiece, saved settings and possibly even your own safety at risk.

Alarm overview

You can display all upcoming alarms and acknowledge them.

The alarm overview contains the following information:

- Date and time
- Cancel criterion
  - The delete criterion specifies the key or softkey that can be used to acknowledge the alarm.
- Alarm number
- Alarm text

Procedure

1. Select the "Diagnostics" operating area.

2. Press the "Alarm list" softkey.

   The "Alarms" window appears.

   All pending alarms are displayed.

   The "Hide SI alarms" softkey is displayed if safety alarms are pending.

3. Press the "Hide SI alarms" softkey if you do not wish to display SI alarms.

4. Press the "Save diag. data" softkey if the cause of the alarm is unknown.

   The function collects all available LOG files of the operating software and saves them to the following directory:

   \user\sinumerik\didac\out_<Date-Time>.7z

5. If there is a problem with the system, you can send the ZIP file to the SINUMERIK hotline to help with the analysis of the problem.
Cancel alarms

In the "Cancel" column it is symbolized how you delete the pending alarms from the alarm list.

6. Position the cursor on an alarm.
7. If an NCK-POWER ON alarm is displayed, turn the unit off and back on (main switch), or press NCK-POWER ON.
   - OR -
   If an NC-Start alarm is displayed, press the <NC-Start> key.
   - OR -
   If a RESET alarm is displayed, press the <RESET> key.
   - OR -
   If a Cancel alarm is displayed, press the <ALARM CANCEL> key or press the "Cancel Alarm delete" softkey.

- OR -

- OR -
If an HMI alarm is displayed, press the "Delete HMI alarm" softkey.

- OR -
If a dialog alarm of the HMI is displayed, press the <RECALL> key.

- OR -
If a PLC alarm is displayed, press the key provided by the machine manufacturer.

- OR -
If a PLC alarm of the type SQ is displayed, press the "Acknowl. alarm" softkey.

The softkeys are activated when the cursor is on the corresponding alarm.

Acknowledgement symbols

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td><img src="symbol" alt="NCK POWER ON" /></td>
<td>NCK POWER ON</td>
</tr>
<tr>
<td><img src="symbol" alt="NC start" /></td>
<td>NC start</td>
</tr>
<tr>
<td><img src="symbol" alt="RESET alarm" /></td>
<td>RESET alarm</td>
</tr>
<tr>
<td><img src="symbol" alt="Cancel alarm" /></td>
<td>Cancel alarm</td>
</tr>
<tr>
<td><img src="symbol" alt="HMI alarm" /></td>
<td>HMI alarm</td>
</tr>
</tbody>
</table>
### Symbol, Meaning

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td><img src="image" alt="Symbol" /></td>
<td>Dialog alarms of the HMI</td>
</tr>
<tr>
<td><img src="image" alt="Symbol" /></td>
<td>PLC alarm</td>
</tr>
<tr>
<td><img src="image" alt="Symbol" /></td>
<td>PLC alarm of the SQ type (alarm number from 800000)</td>
</tr>
<tr>
<td><img src="image" alt="Symbol" /></td>
<td>Safety alarms</td>
</tr>
</tbody>
</table>

**Machine manufacturer**

Please refer to the machine manufacturer's instructions.
15.3 Displaying an alarm log

A list of all the alarms and messages that have occurred so far are listed in the "Alarm Log" window.

Up to 500 administered, incoming and outgoing events are displayed in chronological order.

Machine manufacturer
Please refer to the machine manufacturer's specifications.

Procedure

1. Select the "Diagnostics" operating area.

2. Press the "Alarm log" softkey.

   The "Alarm Log" window opens.
   All of the coming and going events - that have occurred since the HMI was started - are listed.

3. Press the "Display new" softkey to update the list of displayed alarms/messages.

4. Press the "Save Log" softkey.

   The log that is currently displayed is stored as text file alarmlog.txt in the system data in directory /user/sinumerik/hmi/log/alarm_log.
15.4  **Sorting, alarms, faults and messages**

If a large number of alarms, messages or alarm logs are displayed, you have the option of sorting these in an ascending or descending order according to the following criteria:

- Date (alarm list, messages, alarm log)
- Number (alarm list, messages)

As a consequence, for every extensive lists, you can obtain the required information faster.

**Procedure**

1. Select the "Diagnostics" operating area.
2. Press the "Alarm list", "Messages" or "Alarm log" softkey to display the requested messages and interrupts.
3. Press the "Sort" softkey.

   The list of entries is sorted in descending order according to date, i.e. the most recent information is at the beginning of the list.
4. Press the softkey "Ascending" to sort the list in the ascending order.

   The most recent event is shown at the end of the list.
5. Press the "Number" softkey if you wish to sort the alarm list or the list with messages according to numbers.
6. Press the "Descending" softkey to display the list in decreasing/descending order again.
15.5 Creating screenshots

You can create screenshots of the current user interface.

Each screenshot is saved as a file and stored in the following folder:

```
/user/sinumerik/hmi/log/screenshot
```

Procedure

Ctrl + P  Press the "Ctrl + P" key combination.

A screenshot of the current user interface is created in .png format.

The file names assigned by the system are in ascending order from "SCR_SAVE_0001.png" to "SCR_SAVE_9999.png". You can create up to 9,999 screenshots.

Copy file

1. Select the "Setup" operating area.
2. Press the "System data" softkey.
3. Open the folder specified above, and select the required screenshots.
4. Press the "Copy" softkey.
   - OR -
   Press the "Cut" softkey.
5. Open the required archive directory, e.g. on a USB flash drive and press the "Paste" softkey.

Note

WinSCP (840D sl only)

You can also copy the screenshots to a Windows PC using "WinSCP".

Note

Open files (840D sl only)

Open the files in SINUMERIK Operate to view the screenshots. On a Windows PC, you can open the files using a graphic program such as "Office Picture Manager".
15.6 PLC and NC variables

15.6.1 Displaying and editing PLC and NC variables

Changes can only be made to the NC/PLC variables with the appropriate password.

---

**WARNING**

Incorrect parameterization

Changes in the states of NC/PLC variables have a considerable influence on the machine. Incorrect configuration of the parameters can endanger life and cause damage to the machine.

---

In the “NC/PLC Variables” window, enter the NC system variables and PLC variables that you want to monitor or change in the list:

- **Variable**
  Address for NC/PLC variable.
  Incorrect variables have a red background and are displayed with a # character in the value column.

- **Comment**
  Any comment on the variable.
  The columns can be displayed and hidden.

- **Format**
  Specify the format in which the variable is to be displayed.
  The format can be specified (e.g. floating point).

- **Value**
  Displays the actual value of the NC/PLC variables.

---

### PLC variables

<table>
<thead>
<tr>
<th>Inputs</th>
<th>Outputs</th>
<th>Bit memory</th>
<th>Times</th>
<th>Counters</th>
<th>Data</th>
</tr>
</thead>
<tbody>
<tr>
<td>Input bit (Ex), input byte (EBx), input word (EWx), input double word (EDx)</td>
<td>Output bit (Ax), output byte (ABx), output word (AWx), output double word (ADx)</td>
<td>Memory bit (Mx), memory byte (MBx), memory word (MWx), memory double word (MDx)</td>
<td>Time (Tx)</td>
<td>Counter (Cx)</td>
<td>Data block (DBx): Data bit (DBXx), data byte (DBBx), data word (DBWx), data double word (DBDx)</td>
</tr>
<tr>
<td>Input bit (Ix), input byte (IBx), input word (IWx), input double word (IDx)</td>
<td>Output bit (Qx), output byte (QBx), output word (QWx), output double word (QDx)</td>
<td></td>
<td></td>
<td>Counter (Cx)</td>
<td>Data block (VBx): Data bit (V BXx), data byte (VBBx), data word (VBWx), data double word (VBDx)</td>
</tr>
</tbody>
</table>

---
**Formats**

<table>
<thead>
<tr>
<th>Format</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>B</td>
<td>Binary</td>
</tr>
<tr>
<td>H</td>
<td>Hexadecimal</td>
</tr>
<tr>
<td>D</td>
<td>Decimal without sign</td>
</tr>
<tr>
<td>+/-D</td>
<td>Decimal with sign</td>
</tr>
<tr>
<td>F</td>
<td>Floating point (for double words)</td>
</tr>
<tr>
<td>A</td>
<td>ASCII character</td>
</tr>
</tbody>
</table>

**Notation examples**

Permissible notation for variables:

- PLC variables: EB2, A1.2, DB2.DBW2, VB32000002
- NC variables:
  - NC system variables: Notation $AA_IM[1]
  - User variables / GUD: Notation GUD/MyVariable[1,3]
  - OPI notation: /CHANNEL/PARAMETER/R[u1,2]

**Note**

If the PLC user program writes a string into an NC/PLC variable, the string is only displayed correctly if the variable is parameterized on the NC side as a field variable of type "A" (ASCII).

**Example of a field variable**

<table>
<thead>
<tr>
<th>Variable</th>
<th>Format</th>
</tr>
</thead>
<tbody>
<tr>
<td>DBx.DBBy[&lt;number&gt;]</td>
<td>A</td>
</tr>
</tbody>
</table>

**Inserting variables**

The start value for "Filtering/searching" of variables differs. For example, to insert the variable $R[0]$, enter the following start value:

- The start value is 0 if you filter according to "System variables".
- The start value is 1 if you filter according to "All (no filter)". In this case, all signals are displayed and shown in the OPI notation.

The GUD from the machine data is only displayed in the Search window for the variable selection when the associated definition file has been activated. Otherwise, the sought variables must be entered manually, e.g. GUD/SYG_RM[1]
The following machine data is representative for all variable types (INT, BOOL, AXIS, CHAR, STRING): MD18660 $MN_MM_NUM_SYNACT_GUD_REAL[1].

**Note**

**Display of NC/PLC variables**

- System variables can be dependent on the channel. When the channel is switched over, the values from the selected channel are displayed.
  You have the option of having the variables displayed channel-specifically, e.g. $R1:CHAN1 and $R1:CHAN2. The values of channel 1 and channel 2 are displayed, no matter in which channel you are.
- For user variables (GUD), it is not necessary to make a specification according to global or channel-specific GUD. The first element of a GUD array starts with index 0 as for NC variables.
- Using the tooltip, you can display the OPI notation for NC system variables (except for GUD).

**Servo variables**

Servo variables can only be selected and displayed at "Diagnostics" → “Trace”.

### Changing and deleting values

1. Select the "Diagnostics" operating area.

2. Press the "NC/PLC variables" softkey.
   The "NC/PLC Variables" window opens.

3. Position the cursor in the "Variable" column and enter the required variable.

4. Press the <INPUT> key.
   The operand is displayed with the value.

5. Press the "Details" softkey.
   The "NC/PLC Variables: Details" window opens. The information for "Variable", "Comment" and "Value" is displayed in full length.

6. Position the cursor in the "Format" field and select the required format with <SELECT>.

7. Press the "Display comments" softkey.
   The "Comments" column is displayed. You have the option of creating comments or editing existing comments.
   Press the "Display comments" softkey once again to hide the column.

8. Press the "Change" softkey if you would like to edit the value.
   The "Value" column can be edited.
9. Press the "Insert variable" softkey if you wish to select a variable from a list of all existing variables and insert this. The "Select Variable" window opens.

10. Press the "Filter/search" softkey to restrict the display of variables (e.g. to mode group variables) using the "Filter" selection box and/or select the desired variable using the "Search" input box.

11. Press the "Delete all" softkey if you would like to delete all the entries for the operands.

12. Press the "OK" softkey to confirm the changes or the deletion.

- OR -

Press the "Cancel" softkey to cancel the changes.

---

### Editing the variable list

You can edit the variable list using the "Insert line" and "Delete line" softkeys.

- **Insert line**
  - If you press the softkey, a new line inserted before the line marked by the cursor.
  - You can only use the "Insert line" softkey if there is at least one empty line at the end of the variable list.
  - The softkey is deactivated if there is no empty line.

- **Delete line**
  - If you press the "Delete line" softkey, the line marked by the cursor is deleted.
  - An empty line will be added at the bottom of the variable list.

---

### Changing operands

Depending on the type of operand, you can increment or decrement the address by 1 place at a time using the "Operand +" and "Operand -" softkeys.

---

**Note**

**Axis names as index**

For axis names, the "Operand +" and "Operand -" softkeys do not act as index, e.g. for $AA.IM[X1].


15.6.2 Saving and loading screen forms

You have the option of saving the configurations of the variables made in the "NC/PLC variables" window in a screen form that you reload again when required.

Editing screen forms

If you change a screen form that has been loaded, then this is marked using with * after the screen form name.

The name of a screen form is kept in the display after switching-off.

Procedure

1. You have entered values for the desired variables in the "NC/PLC variables" window.
2. Press the ">>" softkey.
3. Press the "Save screen" softkey.
   The "Save screen: Select archiving" window opens.
4. Position the cursor on the template folder for variable screen forms in which your actual screen form should be saved and press the "OK" softkey.
   The "Save screen: Name" window opens.
5. Enter the name for the file and press the "OK" softkey.
   A message in the status line informs you that the screen form was saved in the specified folder.
   If a file with the same name already exists, they you will receive a prompt.
6. Press the "Load screen" softkey.
   The "Load screen" window opens and displays the sample folder for the variable screen forms.
7. Select the desired file and press the "OK" softkey.
   You return to the variable view. The list of all of the predefined NC and PLC variables is displayed.
15.7 Version

15.7.1 Displaying version data

The following components with the associated version data are specified in the "Version data" window:

- System software
- PLC basic program
- PLC user program
- System expansions
- OEM applications
- Hardware

Information is provided in the "Nominal version" column as to whether the versions of the components deviate from the version supplied on the CompactFlash Card.

- The version displayed in the "Actual version" column matches the version of the CF card.
- The version displayed in the "Actual version" column does not match the version of the CF card.

You may save the version data. Version data saved as text files can be further processed as required or sent to the hotline in the event of an error.

Procedure

1. Select the "Diagnostics" operating area.
2. Press the "Version" softkey.
3. Select the component for which you would like more information.
4. Press the "Details" softkey, in order to receive more exact information on the components displayed.
15.7.2 Save information

All the machine-specific information of the control is combined in a configuration via the user interface. You then have the option of saving the machine-specific information on the drives that have been set up.

Procedure

1. Select the "Diagnostics" operating area.

2. Press the "Version" softkey.
   It takes some time to call the version display. While the version data is being determined a progress message box and the appropriate text are displayed in the dialog line.

3. Press the "Save" softkey.
   The "Save version information: Select Archive" window opens. The following storage locations are offered depending on the configuration:
   - Local drive
   - Network drives
   - USB
   - Version data (archive: Data tree in the "HMI data" directory)

4. Then press the "New directory" softkey if you wish to create your own directory.

5. Press the "OK" softkey. The directory is created.

6. Press the "OK" softkey again to confirm the storage location.
   The "Save version information: Name" window opens.

7. Specify the desired settings.
   - "Name:" input field
     the file name is pre-assigned with <Machine name/no.>++<CF-card number> . "_config.xml" or "_version.txt" is automatically attached to the file names.
   - "Comment:" input field
     You can enter a comment that is stored with the configuration data.
   - Version data (.TXT)
     Activate the checkbox if you wish to output the pure version data in the text format.
   - Configuration data (.XML)
     Activate the checkbox if you wish to output the configuration data in the XML format.
     The configuration file contains the data you entered under Machine identity, the license requirements, the version information and the logbook entries.

8. Press the "OK" softkey to start the data transfer.
15.8 Logbook

15.8.1 Overview

The logbook provides you with the machine history in an electronic form. If service is carried out on the machine, this can be electronically saved. This means that it is possible to obtain a picture about the “History” of the control and to optimize service.

Editing the logbook

You can edit the following information:

- Editing information on the machine identity
  - Machine name/No.
  - Machine type
  - Address data

- Make logbook entries (e.g. "filter replaced")

- Deleting logbook entries

Note

Deleting logbook entries

Up to the 2nd commissioning, you have the option to delete all of the entered data up to the time of the first commissioning.

Output of the logbook

You have the possibility of exporting the logbook by generating a file using the "Save version" function in which the logbook is contained as section.

15.8.2 Displaying and editing the logbook

Procedure

1. Select the "Diagnostics" operating area.
2. Press the "Version" softkey.
3. Press the "Logbook" softkey.
   The "Machine logbook" window opens.
### Editing end customer data

4. You have the option of changing the address data of the end customer using the "Change" softkey.

- OR -

Using the “Clear” softkey, you can delete all logbook entries.

All entries, except the date of the first commissioning, are deleted. Softkey "Clear" is deactivated.

---

**Note**

**Deleting logbook entries**

As soon as the 2nd commissioning has been completed, the “Clear” softkey to delete the logbook data is no longer available.

---

### 15.8.3 Making a logbook entry

Using the "New logbook entry" window to make a new entry into the logbook.

Enter your name, company and department and a brief description of the measure taken or a description of the fault.

---

**Note**

**Setting line breaks**

If you wish to make line breaks in the "Fault diagnostics/measure" field, use the key combination <ALT> + <INPUT>.

The date and entry number are automatically added.

**Sorting the entries**

The logbook entries are displayed numbered in the "machine logbook" window.

More recent entries are always added at the top in the display.

---

**Procedure**

1. The logbook is opened.

2. Press the "New entry" softkey.
   
   The "New logbook entry" window opens.

3. Enter the required data and press the "OK" softkey.
   
   You return to the "Machine logbook" window and the entry is displayed below the machine identity data.
Note
Deleting logbook entries
Up to the completion of the 2nd commissioning, you have the option to delete the logbook entries up to the time of the first commissioning using the "Clear" softkey.

Searching for a logbook entry
You have the option for searching for specific entries using the search function.

1. The "Machine logbook" window is opened.
2. Press the "Find" softkey.
3. Enter the desired term in the search form. You can make a search according to date/time, company name/department or according to fault diagnostics/measure.
   The cursor is positioned on the first entry that corresponds to the search term.
4. Press the "Continue search" softkey if the entry found is not the one that you are looking for.

Additional search option
Press the "Go to Beginning" softkey to start the search at the latest entry.
Press the "Go to End" softkey to start the search at the oldest entry.
15.9 Remote diagnostics

15.9.1 Setting remote access

You can influence the remote access to your control in the "Remote diagnostics (RCS)" window.

You set the rights for all remote operating types in this window. The selected rights are defined from the PLC and using the setting at the HMI.

The HMI can restrict the rights specified from the PLC, but however, cannot extend the rights beyond the PLC rights.

If the settings made permit access from outside, then this is still dependent on a manual or automatic confirmation.

Rights for remote access

The "Specified by PLC" field shows the access rights for remote access or remote monitoring specified from the PLC.

Machine manufacturer

Please refer to the machine manufacturer's instructions.

In the "Selected in the HMI" selection box, you have the possibility of setting rights for remote control:

- Do not permit remote access
- Permit remote monitoring
- Permit remote control

Depending on the combination of the settings in the HMI and in the PLC the valid status as to whether access is permitted or not is shown in the "Resulting from this" line.

Settings for the confirmation dialog box

If the settings made for "Specified from the PLC" and "Selected in the HMI" permit access from outside, then this is however, still dependent on either a manual or automatic confirmation.

As soon as a remote access is permitted, at all of the active operating stations, a query dialog box is displayed for the operator at the active operating station to either confirm or reject an access.

For the case that there is no local operation, then the control behavior can be set for this particular scenario. You define how long this window is displayed and whether, after the confirmation has expired, the remote access is automatically rejected or accepted.

Display of the state

- Remote monitoring active
- Remote control active
If remote access is active, using these icons you will be informed in the status line as to whether a remote access is presently active or whether only monitoring is permitted.

### Procedure

1. Select the "Diagnostics" operating area.
2. Press the "Remote diag." softkey.
   The "Remote diagnostics (RCS)" window is opened.
3. Press the "Change" softkey.
   The "Selected in the HMI" is activated.
4. If you desire remote control, select the entry "Permit remote control".

In order that remote control is possible, the entry "Allow remote operation" must be specified in the fields "Specified by PLC" and "Selected in HMI".

5. Enter new values in the group "Behavior for remote access confirmation" if you wish to change the behavior for confirming remote access.
6. Press the "OK" softkey.
   The settings are accepted and saved.

### References

A description of the configuration options can be found in the following reference:

SINUMERIK Operate Commissioning Manual

15.9.2 Permit modem

You can permit remote access to your control via a teleservice adapter IE connected at X127.

**Machine manufacturer**

Please refer to the machine manufacturer's instructions.

**Software option**

You need the "Access MyMachine /P2P" option to display the "Allow modem" softkey.
15.9 Remote diagnostics

15.9.3 Request remote diagnostics

Using the "Request remote diagnostics" softkey, from your control you have the option of actively requesting remote diagnostics with your machinery construction OEM. Access via modem must be enabled if the access is to be made possible via a modem.

**Machine manufacturer**

Please refer to the machine manufacturer's instructions.

When requesting remote diagnostics, you obtain a window with the corresponding pre-assigned data and values of the ping service. If required, you can ask your machine manufacturer for this data.

<table>
<thead>
<tr>
<th>Data</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td>IP address</td>
<td>IP address of the remote PC</td>
</tr>
<tr>
<td>Port</td>
<td>Standard port that is intended for remote diagnostics</td>
</tr>
<tr>
<td>Send duration</td>
<td>Duration of the request in minutes</td>
</tr>
<tr>
<td>Send interval</td>
<td>Cycle in which the message is sent to the remote PC in seconds</td>
</tr>
<tr>
<td>Ping send data</td>
<td>Message for the remote PC</td>
</tr>
</tbody>
</table>

**Procedure**

1. The "Remote diagnostics (RCS)" window is opened.
2. Press the "Allow modem" softkey.
   Access to the control via modem is enabled so that a connection is established.
3. To block access again, press the "Allow modem" softkey again.

1. The "Remote diagnostics (RCS)" window is opened.
2. Press the "Request remote diagnostics" softkey.
   The "Request remote diagnostics" softkey is displayed.
3. Press the "Change" softkey if you would like to edit the values.
4. Press the "OK" softkey.
   The request is sent to the remote PC.
15.9.4 Exit remote diagnostics

Procedure

1. The "Remote diagnostics (RCS)" is opened and it is possible that remote monitoring or remote access is active.

2. Block the modem access if access via modem is to be blocked.

   - OR -

   In the "Remote Diagnostics (RCS)" window, reset the access rights to "Permit no remote access".
Alarm, fault, and system messages

15.9 Remote diagnostics
Teaching in a program

16.1 Overview

The "Teach in" function can be used to edit programs in the "AUTO" and "MDA" modes. You can create and modify simple traversing blocks.

You traverse the axes manually to specific positions in order to implement simple machining sequences and make them reproducible. The positions you approach are applied.

In the "AUTO" teach-in mode, the selected program is "taught".

In the "MDA" teach-in mode, you teach to the MDA buffer.

External programs, which may have been generated offline, can therefore be adapted and modified according to your specific requirements.

Note

Not possible to teach-in program

When an EES program has been selected, it is not possible to teach-in programs.

General sequence

1. Activate teach-in mode.

2. Insert a block.
   - To do this, position the cursor at the desired point in the program and insert an empty line.
   - Press the relevant softkey "Teach position", "Rapid traverse G01", "Straight line G1", or "Circle interpolation position CIP" and "Circle end position CIP".
   - OR -

3. Change an existing block.
   - To do this, mark the desired program block, and press the corresponding softkey "Teach in position ", "Rapid traverse G01", "Straight line G1", or "Circle interpolation point CIP" and "Circle end point CIP".
   - You can only overwrite a block with a block of the same type.

4. Traverse the axes.

5. Press the "Accept" softkey to teach-in the modified or newly created program block.

Note

Teach in multiple blocks

All defined axes are "taught in" in the first teach-in block. In all additional teach-in blocks, only axes modified by axis traversing or manual input are "taught in".

If you exit teach-in mode, this sequence begins again.
16.1 Overview

**Note**

**Selection of axes and parameters for teach-in**

You can select the axes to be included in the teach-in block in the "Settings" window.

You also specify here whether motion and transition parameters are offered for teach-in.

**Operating mode or operating area switchover**

If you switch to another operating mode or operating area while in teach-in mode, the position changes will be canceled and teach-in mode will be cleared.
16.2 Select teach in mode

Change to Teach in mode to adapt the current program.

Requirement

"AUTO" mode: The program to be edited is selected.

"MDI" mode: The program to be edited is loaded into the MDI buffer.

Procedure

1. Select the "Machine" operating area.

2. Press the <AUTO> or <MDA> key.

3. Press the <TEACH IN> key.

4. Press the "Teach prog." softkey.
16.3 Processing a program

16.3.1 Inserting a block

The cursor must be positioned on an empty line.

The windows for pasting program blocks contain input and output fields for the actual values in the WCS. Depending on the default setting, selection fields with parameters for motion behavior and motion transition are available.

When first selected, the input fields are empty, unless axes had already been traversed before the window was selected.

All data from the input/output fields are transferred to the program with the "Accept" softkey.

Procedure

1. Teach-in mode is active.
2. Position the cursor at the desired point in the program. If an empty row is not available, insert one.
3. Press the softkeys "Rap. tra. G0", "Straight line G1", or "Circ. interm. pos. CIP" and "Circ. end pos. CIP".
   The relevant windows with the input fields are displayed.
4. Traverse the axes to the relevant position.
5. Press the "Accept" softkey. A new program block will be inserted at the cursor position.
   - OR -
   Press the "Cancel" softkey to cancel your input.

16.3.2 Editing a block

You can only overwrite a program block with a teach-in block of the same type.

The axis values displayed in the relevant window are actual values, not the values to be overwritten in the block.

Note

If you wish to change any variable in a block in the program block window other than the position and its parameters, then we recommend alphanumerical input.
Procedure

1. Teach-in mode is active.

2. Select the program block to be edited.

3. Press the relevant softkey "Teach position, "Rap. tra. G0", "Straight line G1", or "Circ. intern. pos." CIP", and "Circ. end pos. CIP".

   The relevant windows with the input fields are displayed.

4. Traverse the axes to the desired position and press the "Accept" softkey.

   The program block is taught with the modified values.

   - OR -

   Press the "Cancel" softkey to cancel the changes.

16.3.3 Selecting a block

You have the option of setting the interrupt pointer to the current cursor position. The next time the program is started, processing will resume from this point.

With teach-in, you can also change program areas that have already been executed. This automatically disables program processing.

You must press reset or select a block to resume the program.

Procedure

1. Teach-in mode is active.

2. Place the cursor on the desired program block.

3. Press the "Block selection" softkey.

16.3.4 Deleting a block

In teach-in mode, you can delete both a teach-in block and a program block entirely.
Teaching in a program
16.3 Processing a program

Procedure

1. Teach-in mode is active.
2. Select the program block to be deleted.
3. Press the ">>" and "Delete block" softkeys.
   The program block on which the cursor is positioned is deleted.
16.4 Teach sets

Teach in position
You traverse the axes and write the current actual values directly into a positioning block.

Teach-in rapid traverse G0
You traverse the axes and teach-in a rapid traverse block with the approached positions.

Teach in straight G1
You traverse the axes and teach-in a machining block (G1) with the approached positions.

Teach in circular interpolation CIP
Enter the intermediate and end positions for the circle interpolation CIP. You teach-in each of these separately in a separate block. The order in which you program these two points is not specified.

Note
Make sure that the cursor position does not change during teach-in of the two positions.

You teach-in the intermediate position in the "Circle intermediate position CIP" window.
You teach-in the end position in the "Circle end position CIP" window.
The intermediate or interpolation point is only taught-in with geometry axes. For this reason, at least 2 geometry axes must be set up for the transfer.

Teach-in A-spline
For Akima-spline interpolation, you enter interpolation points that are connected by a smooth curve.
Enter a starting point and specify a transition at the beginning and end.
You teach-in each interpolation point via "Teach in of position".

Software option
You require the "Spline-Interpolation" option for A Spline interpolation.

Machine manufacturer
Please observe the information provided by the machine manufacturer.
Procedure

1. Teach-in mode is active.

2. Press the ">>" and "ASPLINE" softkeys.
   The "Akima-spline" window opens with the input fields.

3. Traverse the axes to the required position and, if necessary, set the transition type for the starting point and end point.

4. Press the "Accept" softkey.
   The new program block is inserted at the cursor-position.
   - OR -
   Press the "Cancel" softkey to cancel your input.

16.4.1 Input parameters for teach-in blocks

Parameters of axes for teach in

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>X</td>
<td>Axis position in the X direction</td>
</tr>
<tr>
<td>Y</td>
<td>Axis position in the Y direction</td>
</tr>
<tr>
<td>Z</td>
<td>Axis position in the Z direction</td>
</tr>
<tr>
<td>I</td>
<td>Coordinate of the circle intermediate points in the X direction</td>
</tr>
<tr>
<td>J</td>
<td>Coordinate of the circle intermediate points in the Y direction</td>
</tr>
<tr>
<td>K</td>
<td>Coordinate of the circle intermediate points in the Z direction</td>
</tr>
</tbody>
</table>

Feedrate (only for G1 and circle end position CIP)

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Description</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>F F</td>
<td>Feedrate</td>
<td>mm/rev</td>
</tr>
<tr>
<td></td>
<td></td>
<td>mm/min</td>
</tr>
</tbody>
</table>

Transition modes

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>G60</td>
<td>Exact stop</td>
</tr>
<tr>
<td>G64</td>
<td>Smoothing</td>
</tr>
<tr>
<td>G641</td>
<td>Programmable corner rounding</td>
</tr>
<tr>
<td>G642</td>
<td>Axis-specific corner rounding</td>
</tr>
</tbody>
</table>
### Parameter Description

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>G643</td>
<td>Block-internal corner rounding</td>
</tr>
<tr>
<td>G644</td>
<td>Axis dynamics corner rounding</td>
</tr>
<tr>
<td>G645</td>
<td>Smoothing</td>
</tr>
</tbody>
</table>

### Motion types

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>CP</td>
<td>Path-synchronous</td>
</tr>
<tr>
<td>PTP</td>
<td>Point-to-point</td>
</tr>
<tr>
<td>PTPG0</td>
<td>Only G0 point-to-point</td>
</tr>
</tbody>
</table>

### Transitional behavior of the spline curve

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Start</td>
<td>Transitional behavior at the beginning</td>
</tr>
<tr>
<td></td>
<td>• BAUTO - automatic calculation</td>
</tr>
<tr>
<td></td>
<td>• BNAT - curvature is zero or natural</td>
</tr>
<tr>
<td></td>
<td>• BTAN - tangential</td>
</tr>
<tr>
<td>End</td>
<td>Transitional behavior at the beginning</td>
</tr>
<tr>
<td></td>
<td>• EAUTO - automatic calculation</td>
</tr>
<tr>
<td></td>
<td>• ENAT - curvature is zero or natural</td>
</tr>
<tr>
<td></td>
<td>• ETAN - tangential</td>
</tr>
</tbody>
</table>
16.5 Settings for teach-in

In the "Settings" window, you define which axes are to be included in the teach-in block and whether motion-type and continuous-path mode parameters are to be provided.

Procedure

1. Teach-in mode is active.

2. Press the ">>" and "Settings" softkeys.
   The "Settings" window opens.

3. Under "Axes to be taught" and "Parameters to be taught", select the check boxes for the desired settings.

4. Press the "Accept" softkey to confirm the settings.
### 17.1 Functions

The "Ctrl-Energy" function provides you with the following options to improve the energy utilization of your machine.

#### Ctrl-E Analysis: Measuring and evaluating the energy consumption

Acquiring the actual energy consumption is the first step to achieving better energy efficiency. The energy consumption is measured and displayed at the control using the SENTRON PAC multi-function device.

Depending on the configuration and connection of the SENTRON PAC, you have the possibility of either measuring the power of the whole machine or only a specific load.

Independent of this, the power is determined directly from the drives and displayed.

#### Ctrl-E Profiles: Control of energy saving states of the machine

To optimize the energy consumption, you have the option of defining energy saving profiles and saving them. For instance, your machine has a basic and a more sophisticated energy-saving mode – or under certain conditions, automatically switches itself off.

These defined energy states are saved as profiles. At the user interface, you have the possibility of activating these energy-saving profiles (e.g. the so-called tea break key).

---

**Note**

**Ctrl-E Deactivating profiles**

Disable Ctrl-E profiles before a series commissioning in order to prevent the NCU unintentionally shutting down.

---

**Machine manufacturer**

Please observe the information provided by the machine manufacturer.

---

**Note**

**Calling of the function via shortcut key**

Press keys <CTRL> + <E> to call the "Ctrl Energy" function.
17.2 Ctrl-E analysis

17.2.1 Displaying energy consumption

The SINUMERIK Ctrl-Energy entry screen provides an easy-to-interpret overview of the energy consumption of the machine. To display the values and the graphical representation, a Sentron PAC must be connected and a long-term measurement configured.

This shows a consumption display with the following bar chart:

- Current power display
- Measurement of the current energy consumption
- Comparison measurement for the energy consumption

![Figure 17-1 Ctrl-Energy entry screen with display of the current energy consumption](image)

**Display in the "Machine" operating area**

The first row of the status display shows the current power status of the machine.

<table>
<thead>
<tr>
<th>Display</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td><img src="image" alt="Red Bar" /></td>
<td>A red bar indicates that the machine is not operating productively.</td>
</tr>
<tr>
<td><img src="image" alt="Dark-Green Bar" /></td>
<td>A dark-green bar in the positive direction indicates that the machine is operating productively and consuming energy.</td>
</tr>
<tr>
<td><img src="image" alt="Light-Green Bar" /></td>
<td>A light-green bar in the negative direction indicates that the machine is feeding energy back into the power supply system.</td>
</tr>
</tbody>
</table>

**References**

Information on the configuration is provided in the following reference:

Procedure

1. Select the "Parameter" operating area.

2. Press the menu forward key and then the "Ctrl-Energy" softkey.

- OR -

Press the <Ctrl> + <E> keys.

The "SINUMERIK Ctrl-Energy" window opens.

17.2.2 Displaying the energy analyses

You can obtain a detailed overview of the energy usage in the "Ctrl-E analyse" window. You obtain the usage display for the following components:

- Sum of the axes
- Sum of the units, if auxiliary units are configured in the PLC
- Sentron PAC
- Sum of the machine

Detailed display of the energy usage

Further, you also have the option of listing the usage values for all drives and where relevant, all auxiliary units.

References

Information on the configuration is provided in the following reference:

Procedure

1. You are in the “SINUMERIK Ctrl-Energy” entry window.

2. Press the "Ctrl-E analysis" softkey. The "Ctrl-E Analysis" window opens. You obtain the summed usage values for all of the components.

3. Press the "Details", softkey to display the energy usage of individual drives and auxiliary units.

17.2.3 Measuring and saving the energy consumption

For the currently selected axes, auxiliary units, SentronPAC or the complete machine you have the option of measuring and recording the energy consumption.

Measurement of the energy consumption by part programs

The energy consumption of part programs can be measured. Single drives are taken into account for the measurement.

You specify in which channel the start and stop of the part program should be initiated and how many repetitions you want to measure.

Save measurements

Save the measured consumption values so that you can subsequently compare the data.

Note

Up to 3 data sets are saved. The oldest data set is automatically overwritten if there are more than three measurements.

Measurement duration

The measurement duration is limited. The measurement is terminated if the maximum measurement time has been reached. The corresponding message is output in the dialog line.

Machine manufacturer

Please observe the information provided by the machine manufacturer.

Precondition

You have pressed the “Ctrl-E analyse” softkey and the “Ctrl-E analyse” window has been opened.
Procedure

1. Press the "Start measurement" softkey.
The "Setting Measurement: Select Device" window opens.

2. Select the desired device in the list, possibly activate the "Measure part program" checkbox, enter the number of repetitions, select the required channel, and press the "OK" softkey.
The trace is started.

3. Press the "Stop measurement" softkey.
The measurement is terminated.

4. Press the "Save measurement" softkey to save the consumption values of the actual measurement.

The selection of the axis to be measured depends on the configuration.

References

Information on the configuration is provided in the following reference:

17.2.4 Tracking measurements

You have the option of graphically displaying the actual and saved measurement curves.

Precondition

You have pressed the "Ctrl-E analyse" softkey and the "Ctrl-E analyse" window has been opened.

Procedure

1. Press the "Graphic" softkey.
The actual measurement is displayed as a blue measurement curve in the "Ctrl-E analyse" window.

2. Press the "Saved measurements" softkey to display the measurements last saved.
In addition, measurement curves with 3 different colors paths are displayed together with the measurement time.

3. Press the "Saved measurements" softkey again if you only want to see the actual measurement.
17.2.5 Tracking usage values

You have the option of displaying the actual and saved usage values in a detailed table.

<table>
<thead>
<tr>
<th>Display</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td>Start of the measurement</td>
<td>Shows the time at which the measurement was started by pressing the</td>
</tr>
<tr>
<td></td>
<td>“Start measurement” softkey.</td>
</tr>
<tr>
<td>Duration of the measurement [s]</td>
<td>Shows the measurement duration in seconds until the &quot;Stop measurement&quot;</td>
</tr>
<tr>
<td></td>
<td>softkey is pressed.</td>
</tr>
<tr>
<td>Device</td>
<td>Displays the selected measured component.</td>
</tr>
<tr>
<td></td>
<td>● Manual (fixed value, z.B. base load, defined in the PLC)</td>
</tr>
<tr>
<td></td>
<td>● Sentron PAC</td>
</tr>
<tr>
<td></td>
<td>● Sum of the units (if defined in the PLC)</td>
</tr>
<tr>
<td></td>
<td>● Sum of the axes</td>
</tr>
<tr>
<td></td>
<td>● Total, machine</td>
</tr>
<tr>
<td>Supplied energy [kWh]</td>
<td>Displays the energy drawn by the selected measured component in kilo-</td>
</tr>
<tr>
<td></td>
<td>watts per hour.</td>
</tr>
<tr>
<td>Regenerated energy [kWh]</td>
<td>Shows the regenerated energy of the selected measured component in</td>
</tr>
<tr>
<td></td>
<td>kilowatts per hour.</td>
</tr>
<tr>
<td>Energy totals [kWh]</td>
<td>Shows the total of all measured drive values or the total of all axes</td>
</tr>
<tr>
<td></td>
<td>as well as fixed value and Sentron PAC.</td>
</tr>
</tbody>
</table>

Display in the "Ctrl-E analysis" window: Table"

Precondition

1. You have pressed the "Ctrl-E analyse" softkey and the "Ctrl-E analyse" window has been opened.
2. You have already saved measurements.

Procedure

Press the "Graphic" and "details" softkeys.

In the "Ctrl-E analysis" window: Details", measurement data and usage values of the last three saved measurements – and possibly an actual measurement – are displayed in a table.

17.2.6 Comparing usage values

You have the option of comparing usage values (power drawn and power fed back) of actual and saved measurements.
Precondition

1. You have pressed the "Ctrl-E analyse" softkey and the "Ctrl-E analyse" window has been opened.
2. You have already saved measurements.

Procedure

1. Press the "Graphic" softkey.
   The power drawn and the recovered power of the actual measurement are displayed in a bar diagram.
3. Press the "Saved measurements" softkey to compare the last 3 measurements that were saved.
4. Press the "Saved measurements" softkey again if you only want to see the actual comparison.

17.2.7 Long-term measurement of the energy consumption

The long-term measurement of energy consumption is performed in the PLC and saved. The values from times in which the HMI is not active are also recorded.

Measured values

The infeed and regenerative power values as well as the sum of the power are displayed for the following periods:

- Current and previous day
- Current and previous month
- Current and previous year

Precondition

SENTRON PAC is connected.
Procedure

1. The "Ctrl-E Analysis" window is open.

2. Press the "Long time measurement" softkey.
   The "SINUMERIK Ctrl-Energy Analysis Long-term Measurement" window opens.
   The results of the long-term measurement are displayed.

3. Press the "Back" softkey to terminate the long-term measurement.
17.3 Ctrl-E profiles

17.3.1 Using the energy-saving profile

In the "Ctrl-E profiles" window, you can display all of the defined energy-saving profiles. Here, you have the option of directly activating or inhibiting a required energy-saving profile, or re-enabling profiles.

SINUMERIK Ctrl-Energy energy-saving profiles

<table>
<thead>
<tr>
<th>Display</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td>Energy-saving profile</td>
<td>All energy-saving profiles are listed.</td>
</tr>
<tr>
<td>active in [min]</td>
<td>The remaining time until the defined profile is reached is displayed.</td>
</tr>
</tbody>
</table>

Note

Disable all energy-saving profiles

For example, in order not to disturb the machine while measurements are being made, select "Disable all".

Once the pre-warning time of a profile has been reached, an alarm window that shows the remaining time is displayed. Once the energy-saving mode has been reached, then an appropriate message is displayed in the alarm line.

Predefined energy-saving profiles

<table>
<thead>
<tr>
<th>Energy-saving profile</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td>Simple energy-saving mode (machine standby)</td>
<td>Machine units that are not required are either throttled or shut down. When required, the machine is immediately ready to operate again.</td>
</tr>
<tr>
<td>Full energy-saving mode (NC standby)</td>
<td>Machine units that are not required are either throttled or shut down. Wait times are incurred at the transition into the ready to operate state.</td>
</tr>
<tr>
<td>Maximum energy-saving mode (auto shut-off)</td>
<td>The machine is completely switched-off. Longer wait times are incurred at the transition into the ready to operate state.</td>
</tr>
</tbody>
</table>

Machine manufacturer

The selection and function of the displayed energy-saving profiles can differ. Please observe the information provided by the machine manufacturer.
References

Information on the configuration of the energy-saving profiles is provided in the following reference:


Procedure

1. Select the "Parameter" operating area.

2. Press the menu forward key and then the "Ctrl-Energy" softkey.

- OR -

Press the <CTRL> + <E> keys.

3. Press the "Ctrl-E profile" softkey.

The "Ctrl-E Profile" window opens.

4. Position the cursor on the required energy-saving profile and press the "Activate immediately" softkey if you wish to directly activate this state.

5. Position the cursor on the required energy-saving profile and press the "Disable profile" softkey if you wish to disable this state.

The profile is inhibited and does not become active. The energy-saving profile is grayed-out and displayed without any time information. The labeling of the "Disable profile" softkey changes to "Enable profile". Press the "Enable profile" softkey in order to withdraw the energy-saving profile disable.

6. Press the "Disable all" softkey in order to disable all states.

All the profiles are disabled and cannot become active. The labeling of the "Disable all" softkey changes to "Enable all".

6. Press the "Enable all" softkey to withdraw the disable for all profiles.
18.1 Easy XML

The "Create user dialogs" function allows you to design customer- and application-specific HMI user interfaces with an XML-based script language.

This script language makes it possible to display machine-specific menus and dialog forms in the <CUSTOM> operating area on the HMI.

These scripts can also be executed from an NC program with the MMC(...) instruction.

**Machine manufacturer**
Please observe the information provided by the machine manufacturer.

**Usage**

The defined script instructions offer the following properties:

1. Display dialogs containing the following elements:
   - Softkeys
   - Variables
   - Texts and Help texts
   - Graphics and Help displays

2. Call dialogs by:
   - Pressing the (start) softkeys

3. Restructure dialogs dynamically:
   - Edit and delete softkeys
   - Define and design variable fields
   - Insert, exchange, and delete display texts (language-dependent or language-neutral)
   - Insert, exchange, and delete graphics

4. Initiate operations in response to the following actions:
   - Displaying dialogs
   - Inputting values (variables)
   - Selecting a softkey
   - Exiting dialogs

5. Data exchange between dialogs
6. Variables
   – Read (NC, PLC, drive and user variables)
   – Write (NC, PLC, drive and user variables)
   – Combine with mathematical, comparison or logic operators

7. Execute functions:
   – Subprograms
   – File functions
   – PI services

8. Apply protection levels according to user classes

9. Controlling dialog content with part program instructions

Calling user dialogs
If the "xmldial.xml" configuration file is stored in the /oem/sinumerik/hmi/appl directory, start the user dialogs by pressing the <CUSTOM> key.

The configured softkeys are displayed when the <CUSTOM> operating area is called. You open and operate the configured dialogs via the softkeys.

Note
After the initial copying of the file to the directory, a RESET of the control is required.

Note
Scripts that originate from the SINUMERIK 802D sl and 808 can be executed without change in the 828D.

References
A description for configuring your own dialogs is available in the following reference:

SINUMERIK 828D Commissioning Manual;
Easy XML Programming Manual
18.2 Easy Extend

18.2.1 Overview

Easy Extend enables machines to be retrofitted with additional devices, which are controlled by the PLC or that require additional NC axes (such as bar loaders, swiveling tables or milling heads), at a later point in time. These additional devices are easily commissioned, activated, deactivated or tested with Easy Extend.

Communication

The communication between the operator component and the PLC is performed via a PLC user program. The sequences to be executed for the installation, activation, deactivation and testing of a device are stored in a statement script.

Available devices and device states are displayed in a list. The view of the available devices can be controlled for users according to their access rights.

The subsequent chapters are selected for example only and are not available in every statement list.

Machine manufacturer

Please observe the information provided by the machine manufacturer.

Up to 64 devices can be managed.

References

SINUMERIK 828D Commissioning Manual

18.2.2 Enabling a device

The available device options may be protected by a password.

Machine manufacturer

Please observe the information provided by the machine manufacturer.
18.2 Easy Extend

Procedure

1. Select the "Parameter" operating area.

2. Press the menu forward key and then the "EasyExtend" softkey.
   A list of the connected devices is displayed.

3. Press the "Enable function" softkey.
   The "Enabling of the Devices Option" window opens.

4. Enter the option code and press the "OK" softkey.
   A tick appears in the appropriate checkbox in the "Function" column and
   the function is enabled.

### 18.2.3 Activating and deactivating a device

<table>
<thead>
<tr>
<th>Status</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td>✔</td>
<td>Device activated</td>
</tr>
<tr>
<td>🔴</td>
<td>System waiting for PLC checkback signal</td>
</tr>
<tr>
<td>❌</td>
<td>Device faulty</td>
</tr>
<tr>
<td>⚠</td>
<td>Interface error in the communication module</td>
</tr>
</tbody>
</table>

Procedure

1. Easy Extend is opened.

2. You can select the desired device in the list with the <Cursor down> or <Cursor up> keys.

3. Position the cursor on the device option for which the function has been unlocked and press the "Activate" softkey.
   The device is marked as activated and can now be used.

4. Select the desired activated device and press the "Deactivate" softkey to
   switch the device off again.
18.2.4 Initial commissioning of additional devices

Normally, the device has already been commissioned by the machine manufacturer. You have the option of commissioning the devices subsequently, for example when retrofitting additional devices. The "Start-up" softkey has been declared as Manufacturer data class (M).

Procedure

1. Select the "Parameter" operating area.
2. Press the menu forward key and then the "Easy Extend" softkey.
4. Press the "Comm. start-up" softkey to start the commissioning. Before starting, a complete data backup is made to which you can resort in an emergency.
5. Press the "Cancel" softkey if you want to abort commissioning prematurely.
6. Press the "Restore" softkey to load the original data.
7. Press the "Device function test" softkey to test the machine manufacturer's intended function.

18.2.5 Commissioning Easy Extend

The additional devices are managed with an Easy XML script.

References

For further information about configuring, please refer to:
SINUMERIK 828D Commissioning Manual
18.3 SINUMERIK Integrate Run MyScreens

"Run MyScreens" allows you to design your own user interface for machine manufacturer- or user-specific functional expansions and implement a user-specific layout.

You also have the possibility to modify or replace configured Siemens or machine-manufacturer user interfaces.

With newly created user interfaces, you can process part programs, for example. Dialogs are designed directly on the control system.

Software option

To expand the number of dialogs, you require one of the following software options:

- SINUMERIK 828D/840D sl, SINUMERIK Integrate Run MyScreens
- SINUMERIK 840D sl, SINUMERIK Integrate Run MyScreens + Run MyHMI
- SINUMERIK 840D sl, SINUMERIK Integrate Run MyHMI / 3GL
- SINUMERIK 840D sl, SINUMERIK Integrate Run MyHMI / WinCC

Machine manufacturer

Please observe the information provided by the machine manufacturer.

Usage

Defined script instructions permit the following functions:

1. Display dialogs containing the following elements:
   - Softkeys
   - Variables
   - Texts and Help texts
   - Graphics and Help displays

2. Call dialogs by:
   - Pressing the (start) softkeys

3. Restructure dialogs dynamically:
   - Edit and delete softkeys
   - Define and design variable fields
   - Insert, exchange, and delete display texts (language-dependent or language-neutral)
   - Insert, exchange, and delete graphics

4. Initiate operations in response to the following actions:
   - Displaying dialogs
   - Inputting values (variables)
   - Selecting a softkey
   - Exiting dialogs
5. Data exchange between dialogs

6. Variables
   - Read (NC, PLC and user variables)
   - Write (NC, PLC and user variables)
   - Combine with mathematical, comparison or logic operators

7. Execute functions:
   - Subprograms
   - File functions
   - PI services

8. Apply protection levels according to user classes

References

A description for configuring your own dialogs is available in the following reference:

SINUMERIK 828D Commissioning Manual;

SINUMERIK Integrate Run MyScreens (BE2) Programming Manual; SINUMERIK 840D sl / 828D
18.3 SINUMERIK Integrate Run MyScreens
19.1 Overview

The mobile SINUMERIK HT 8 handheld terminal combines the functions of an operator panel and a machine control panel. You are thus provided the possibility of monitoring, operating, teaching and programming in immediate proximity to the machine.

Operation

The 7.5" TFT color display provides touch operation. Membrane keys are featured for traversing the axes, numeric input, control of the cursor, and for machine control panel functions (e.g. start and stop).

The HT 8 is equipped with an emergency stop button and two 3-stage acknowledgment buttons. You can also connect an external keyboard.

References

For more information about connection and startup of the HT 8, see the following references:
Customer keys
The four customer keys can be assigned arbitrarily and can be configured customer-specifically.

Machine manufacturer
Please observe the information provided by the machine manufacturer.

Integrated machine control panel
An MCP is integrated in the HT 8. It is comprised of keys (e.g. start and stop) as well as softkeys emulating keys.

A description of individual keys can be obtained in the chapter "Controls on the machine control panel".

Note
PLC interface signals that are triggered via the softkeys of the machine control panel menus are edge triggered.

Acknowledgment button
The HT 8 has two acknowledgment buttons. You are therefore provided the opportunity of activating the acknowledgement function for operator actions which require approval (e.g. showing the traversing keys) with either hand.

Acknowledgment buttons are available for the following button positions:
- Released (no activation)
- Acknowledgment (center position) - acknowledgment for channels 1 and 2 is on the same switch
- Panic (completely pushed through)

Traversing keys
To traverse the axes of your machine with the HT 8 traversing keys, "JOG" or "MDI" mode and the "REF. POINT" or "TEACH IN" functions must have been selected. You have to press the acknowledgment button depending on the setting.

Virtual keyboard
A virtual keyboard is available for the easy entry of values.
Changing the channel

- You are able to switch the channel by touch in the status display:
  - In the Machine operating area (large status display), by touch operation of the channel display in the status display.
  - In the other operating areas (no status display), by touch operation of the channel display in the screen headers (yellow field).
- The “1… n CHANNEL” softkey is available in the machine control panel menu that can be reached via the user menu key "U".

Operating area switchover

You display the operating area menu by touching the display symbol for the active operating area in the status display.

Handwheel

The HT 8 is available with a hand wheel.

Further information about the connection can be found in the Equipment Manual SINUMERIK 840D sl Operating Components - Handheld Units.
19.2 Traversing keys

The traversing keys are not labeled. However, you can display a label for the keys in place of the vertical softkey bar.

Labeling of the traversing keys is displayed for up to six axes on the touch panel by default.

Machine manufacturer
Please refer to the machine manufacturer's specifications.

Showing and hiding

You can link the showing and hiding of the label to activation of the enabling button, for example. In this case, the traversing keys are displayed when you press the enabling button.

If you release the enabling button, the traversing keys are hidden again.

Machine manufacturer
Please refer to the machine manufacturer's specifications.

All existing vertical and horizontal softkeys are covered or hidden, i.e. other softkeys cannot be used.
19.3 Machine control panel menu

You select keys from the machine control panel which are reproduced by the software by touch operation of the relevant softkeys.

See Section “Controls on the machine control panel” for a description of the individual keys.

Note

PLC interface signals that are triggered via the softkeys of the machine control panel menus are edge triggered.

Showing and hiding

The user menu key “U” displays the CPF softkey bar (vertical softkey bar) and the user softkey bar (horizontal softkey bar).

Press the menu forward key to extend the horizontal user softkey bar. There are 8 additional softkeys available.

You use the “Back” softkey to hide the menu bar again.
Softkeys on the machine control panel menu

Available softkeys:

"Machine" softkey    Select the "Machine" operating area
"[VAR]" softkey      Select the axis feedrate in the variable increment
"1… n CHANNEL" softkey    Changing the channel
"Single Block" softkey    Switch single block execution on/off
"WCS MCS" softkey    Switch between WCS and MCS
"Back" softkey      Close window

Note
The window will automatically disappear when changing regions areas with the <MENU SELECT> key.
19.4 Virtual keyboard

The virtual keyboard is used as the input device for touch operator panels.

Open the virtual keyboard by double-clicking on an input-enabled operator control (program editor, editing fields). It is possible to position the virtual keyboard anywhere within the user interface.

You can choose between a full keyboard and a downsized keyboard featuring the numeric keypad only. With the full keyboard, it is possible to switch between English keyboard layout and the keyboard layout which corresponds with the actual language set for the respective country.

Procedure

1. Position the cursor on the desired entry field.
2. Click the input field.
   The virtual keyboard is displayed.
3. Enter your values via the virtual keyboard.
4. Press the <INPUT> key.

   - OR -
   Position the cursor on another operator element.
   The value is accepted and the virtual keyboard is closed.

Positioning of the virtual keyboard

Press and hold the open area to the left of the icon for "Close window" with the stylus or a finger. Move the keyboard to the desired position.
Special keys on the virtual keyboard

1. "Tilde" key
   - Changes the sign in a numerical entry field.
   - A tilde character is inserted in a text box (e.g. program editor).

2. "Eng" key
   Toggles the keyboard assignment between the English keyboard assignment and the keyboard assignment for the current language setting.

3. Area for positioning the virtual keyboard.

4. "Num" key
   Reduces the virtual keyboard to the number block.

Number block of the virtual keyboard

Press the "ABC" key to return to the full keyboard.
19.5 Calibrating the touch panel

It is necessary to calibrate the touch panel upon first connection to the controller.

**Note**

**Recalibration**

If the operation is not exact, then redo the calibration.

**Procedure**

1. Press the menu back key and the <MENU SELECT> key at the same time to start the TCU service screen.

2. Touch the "Calibrate TouchPanel" button. The calibration process will be started.

3. Follow the instructions on the screen and touch the three calibration points one after the other. The calibration process has terminated.

4. Touch the horizontal softkey "1" or the key with the number "1" to close the TCU service screen.
19.5 Calibrating the touch panel
Appendix

A.1 840D sl / 828D documentation overview
### Appendix

#### A.1 840D sl / 828D documentation overview

<table>
<thead>
<tr>
<th>General documentation</th>
<th>User documentation</th>
<th>Manufacturer/Service Documentation</th>
<th>Information/training</th>
<th>Electronic documentation</th>
</tr>
</thead>
<tbody>
<tr>
<td>Advertising brochure</td>
<td>Operating Manual</td>
<td>Equipment Manual</td>
<td>Training documentation</td>
<td>DOConCD</td>
</tr>
<tr>
<td>- SINUMERIK 840D sl</td>
<td>- Turning</td>
<td>- NCU</td>
<td>- Basic milling with</td>
<td>Industry Online</td>
</tr>
<tr>
<td>- SINUMERIK 828D</td>
<td>- Milling</td>
<td>- Commissioning Manual</td>
<td>ShopMill</td>
<td>Support (SIO2)</td>
</tr>
<tr>
<td>- SINUMERIK 828D BASIC</td>
<td>- Grinding</td>
<td>- Service Manual</td>
<td>- Basic turning with</td>
<td>Industry Mall</td>
</tr>
<tr>
<td>Catalog NC 62</td>
<td>Programming Manual</td>
<td>Commissioning Manual</td>
<td>ShopTurn</td>
<td></td>
</tr>
<tr>
<td>- SINUMERIK 828D</td>
<td>- Job planning</td>
<td>- Tool and mold making</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Catalog NC 82</td>
<td>- Measuring cycles</td>
<td>- Safety Integrated</td>
<td></td>
<td></td>
</tr>
<tr>
<td>- SINUMERIK 840D sl</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Catalog PM 21</td>
<td>Programming Manual</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>- SIMOTION, SINAMICS S120</td>
<td>- ISO turning</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Intelligent machine</td>
<td>- Alarms</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>tool operation</td>
<td>Diagnostics Manual</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>- S120</td>
<td>- Alarms</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>System Manual</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Ctrl-Energy</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

#### Grinding

Operating Manual, 06/2019, A5E44903521B AB
"SINUMERIK Operate Gen. 2"
   Multitouch panel, 63
"SINUMERIK Operate Generation 2" user interface
   Touch operator controls, 69
"SINUMERIK Operate Generation 2" user interface, 63
   Function key block, 68
   Virtual keyboard, 70

A
ABC keyboard, 78
Acknowledgment button, 462
Actual-value display, 37
Adapter-transformed view, 353
Advanced Surface, 199
Alarm log
   display, 415
   Sorting, 416
Alarms
   Backing up log data, 412
   cancel, 413
   Displaying, 412
   Sorting, 416
Any file, 369
Archive
   Generate in the program manager, 394
   generating in the system data, 395
   Punched tape format, 394
   read in archive from system data, 398
   reading-in in the Program Manager, 397
Auxiliary functions
   H functions, 200
   M functions, 200
Axes
   Defined increment, 138
   Direct positioning, 140
   Referencing, 86
   Repositioning, 153
   Traversing, 138
   Variable increment, 139

B
B axis
   Calibrating the swivel axis, 286
Backing up
   Data - in the Program Manager, 394
   Data - via the system data, 395
   Parameter, 404
   Setup data, 400
Base offset, 98
Basic blocks, 149
Binary format, 394
Block
   Search, 154
   Searching - interruption point, 157
Block search
   Mode, 158
   Program interruption, 157
   Search target parameters, 157, 158
   Search target specification, 156
   Using, 154

C
Calibrating the swivel axis
   B axis, 286
Camera
   Streaming, 58
   Channel switchover, 91
   Chuck dimensions, 120
   Cleaning mode, 57
   Coarse and fine offsets, 98
   Code carrier connection, 319
   Collision avoidance, 289
   Displaying the machine model (kinematic chains), 291
   Machine operating area, 292
   Settings, 292
   Configured stop, 161
   Context-sensitive online help, 59
   Contour call - CYCLE62
   Function, 246
   Parameter, 247
   Coordinate system
      Switching over, 93
   Counter-spindle, 121
Create
   User dialog, 453
Creating
   Dressing program, 368
   Multitool, 346
   Program block, 170
Ctrl-Energy
   Comparing usage values, 448
   Displaying measured curves, 447
   Displaying usage values, 448
   Energy analysis, 444, 445
   Energy-saving profiles, 451
   Functions, 443
   Measuring the energy consumption, 446
   Saved measuring curves, 447, 448
Cutting edge numbers, 311
CYCLE4071
   External programming, 254
CYCLE4072
   External programming, 256
CYCLE4073
   External programming, 260
CYCLE4074
   External programming, 261
CYCLE4075
   External programming, 264
CYCLE4077
   External programming, 267
CYCLE4078
   External programming, 271
CYCLE4079
   External programming, 273
CYCLE435 - Set dresser coordinate system
   External programming, 248
CYCLE495 - form-truing
   External programming, 250
CYCLE62- contour call
   Function, 246
   Parameter, 247
Cylinder error compensation, 123
Cylindrical grinding
   Measuring a tool, 106

Dictionary
   Importing, 52
Directory
   Copying, 379
   Creating, 366
   Deleting, 381
   Highlight, 377
   Inserting, 379
   Naming convention, 366
   Properties, 382
   Selecting, 377
Display Manager
   Operator controls, 81
Dressing program
   Creating, 368
   Program directory, 357
DRF (handwheel offset), 160
Drives
   Logical drive, 384
   Setting up, 384
DRY (dry run feed), 160
Duplo number, (See sister tool number)
DXF file
   Changing the section, 179
   Cleaning, 177
   close, 177
   Delete Element, 183
   Delete range, 182
   Increasing / decreasing the section, 178
   Machining plane, 181
   Open, 177
   Rotating the drawing, 179
   Saving, 183
   Select contour and accept, 184
   Select the machining range, 182
   Snap radius, 181
   Specifying a reference point, 184
   Zero point, 184
DXF-Reader, 176

E
Easy Extend
   Activate/deactivate device, 456
   Additional devices, 455
   Enabling a device, 455
   First commissioning of the additional devices, 457
   Initial commissioning, 457
Editor
   Calling, 165
   Settings, 172
EES (execution from external storage), 387
Energy analysis
   Details, 445
   Displaying, 445
   Long time measurement, 449
Energy consumption
   Displaying, 444
   Measuring, 446
Energy-saving profiles, 451
Equipping tools
   Multitool, 347
   EXTCALL call, 391

F
Feed data
   Actual value window, 39
Finger gestures, 65
Form-truing - CYCLE495
   External programming, 250
FTP drive, 360
Function
   REF POINT, 89
   REPOS, 89
   Single block, 90
   TEACH IN, 90
Function key block
   "SINUMERIK Operate Generation 2" user interface, 68

G
G Code program
   Creating, 368
G functions
   Display all G groups, 199
   Displaying selected G groups, 197
   Mold making, 199
GCC (G code converter), 161
Global R parameters, 188
Global user variables, 191
Gloves, 64

H
Handheld terminal 8, 461
Handwheel
   Assigning, 126
Highlight
   Directory, 377
   Program, 377

HT 8
   Acknowledgment button, 462
   Overview, 461
   Touch Panel, 469
   Traversing keys, 464
   User menu, 465
   Virtual keyboard, 467

I
IME
   Chinese characters, 49
   Korean characters, 53
Initial commissioning
   Easy Extend, 457
   Interruption point
      Approaching, 157

J
Job list, 370

K
Key combinations - Simulation
   Changing the section, 225
   Enlarging/reducing a graphic, 223
   Feedrate, 217
   Moving the graphic, 224
   Override, 217
   Rotating the view, 224
   Single block mode, 218

L
Layer selection, 177
Loading
   Multitool, 349
Local drive
   Creating an NC directory, 358
Logbook
   Delete entries, 426
   Displaying, 425
   Edit the address data, 425
   Entry search, 427
   Making an entry, 426
   Output, 424
   Overview, 425
   Long time measurement
      Energy analysis, 449
M
Machine control panel
in the sidescreen, 77
Operator controls, 30
Machine model, 289
Machine-specific information, 424
Machining times
Delete, 175
Display in block display, 40, 148
Representation, 228
Magazine
Deleting tools, 334
Loading tools, 334
Positioning, 332
Relocating tools, 334
Unloading tools, 334
Magazine list, 331
Magazine management, 305
Main spindle, 121
Manual mode, 133
Settings, 141
Traversing axes, 138
MDA
Deleting a program, 131
MDI
Executing a program, 130
Loading a program, 128
Saving a program, 129
Measurement
Tool, 315
Measuring
Workpiece zero, 117
Measuring a tool
Cylindrical grinding, 106
Surface grinding, 110
Messages
Displaying, 411
Sorting, 416
Mode groups, 91
MRD (Measuring Result Display), 161
Multi-channel view, 295
"Machine" operating area, 296
OP015, OP019, 299
Settings, 301
Multitool, 345
Creating, 346
Delete, 349
Equipping tools, 347
Loading, 349
Parameters in the tool list, 345
Positioning, 350
Reactivating, 351
Relocating, 350
Removing tool, 348
Unloading, 350
Multitouch panel
SINUMERIK Operate Gen. 2, 63
Widescreen format, 71
N
Navigation bar
Sidescreen, 71
NC/PLC variables
Changing, 420
displaying, 418
New contour
Function - Milling, 237
Parameter - Milling, 238
O
Online help
Context-sensitive, 59
Operating areas, 19
Changing, 42
Selecting, 42
Operating mode
AUTO, 90
Changing, 42
JOG, 89, 133
MDI, 90
REPOS, 89
Operator controls
Display Manager, 81
Machine control panel, 30
Operator panel fronts, 20
P
Pages, 71
Panel
cleaning, 57
Parameter
Back ing up, 404
Calculating, 44
Changing, 44
Entering, 43
Positioning
Multitool, 350
Index

Preview
  Program, 376

Program
  Closing, 362
  Copying, 379
  Correcting, 152
  Creating with cycle support, 233
  Deleting, 381
  Editing, 165
  Executing, 364
  Highlight, 377
  Inserting, 379
  Naming convention, 366
  Open, 362
  Preview, 376
  Properties, 382
  Renumbering blocks, 170
  Replacing text, 167
  Running-in, 146
  Searching for a program position, 165
  Selecting, 145, 377
  Teach-in, 433

Program block
  Copying and inserting, 168
  Current, 40, 148
  Delete, 168
  insert, 168
  Numbering, 169
  Search, 165
  Selecting, 168

Program blocks, 170

Program control
  activate, 161
  Modes of operation, 160

Program correction, 152

Program level, 150

Program list, 372

Program Manager, 355
  Searching for directories and files, 374

Program runtime, 205

Programmed stop 1, 160

Programmed stop 2, 160

Programs
  Managing, 355

Properties
  Directory, 382
  Program, 382

Protection levels
  Softkeys, 56

PRT (no axis motion), 160

Q
  Quantity, 325

R
  R parameters, 189
    Backing up, 404
  Reactivating
    Multitool, 351
    Tool, 327
  Reference, 86
  Relocating
    Multitool, 350
    Tool, 333
  Remote access
    Allowing, 429
    Setting, 428
  Remote diagnostics, 428
    Exit, 431
    requesting, 430
  Removing tools
    Multitool, 348
  Repositioning, 153
  RG0 (reduced rapid traverse), 160
  Run MyScreens, 458

S
  Save
    Parameter, 404
    Setup data, 400
  SB (single blocks), 160
  SB1, 146
  SB2, 146
  SB3, 146
  Screen layout, 68
  Screenshots
    Copying, 417
    Creating, 417
    Open, 417
  Search
    in the Program Manager, 374
  Search mode, 158
  Searching
    Logbook entry, 427
  Selecting
    Directory, 377
    Program, 377
Set dresser coordinate system - CYCLE435
   External programming, 248
Setting actual values, (See setting zero offsets)
   Setting zero offsets, 391
   Settings
      Collision avoidance, 292
      Editor, 172
      For automatic operation, 207
      For manual operation, 141
      Multi-channel view, 301
      Teach-in, 442
      Tool lists, 353
Setup data
   Backing up, 400
   Reading in, 401
Sidescreen
   ABC keyboard, 77
   MCP, 77
   Navigation bar, 71
   Operator controls, 71
   Overview, 71
   Pages, 77
   Requirements, 71
   Show, 73
   Standard widgets, 73
   Widgets, 71
Simulation
   3D view, 220
   Before machining, 213
   Canceling, 213
   Changing the feedrate, 217
   Changing the graphic section, 225
   Dressing view, 219
   Face view (cylindrical grinding), 220
   Further side views (surface grinding), 220
   Machine space, 221
   Moving the graphic, 224
   Non-modal, 218
   Overview, 209
   Program control, 217
   Rotating the graphic, 224
   Side view, 219
   Starting, 213
   Stopping, 213
   Tool paths, 222
   Views, 219
Simultaneous recording
   3D view, 220
   Before machining, 215
   Changing the graphic section, 225
   Dressing view, 219
   During machining, 216
   Face view (cylindrical grinding), 220
   Further side views (surface grinding), 220
   Machine space, 221
   Moving the graphic, 224
   Overview, 209
   Rotating the graphic, 224
   Side view, 219
   Starting - before machining, 215
   Tool paths, 222
   Views, 219
Single block
   Coarse (SB1), 146
   Fine (SB3), 146
SINUMERIK Operate Gen. 2
   Screen layout, 68
Sister tool number, 310
Skip blocks, 161
SKP (skip blocks), 161
Special characters, 21
Spindle chuck data
   Parameter, 122
   Store chuck dimensions, 120
Spindle data
   Actual value window, 40
   Spindle speed limitation, 120
   Standard widget
      Actual values, 73
      Alarms, 74
      Axle load, 75
      Camera, 77
      Service life, 76
      Tool, 75
      Variables, 75
      Zero point, 74
Startup, 85
Status display, 35
Supplement
   User interface, 458
Surface grinding
   Measuring a tool, 110
Switching over
   Coordinate system, 93
Synchronized actions
   Displaying status, 203

T

   Tailstock, 122
   Teach-in, 433
   Changing blocks, 436
   Circle intermediate position CIP, 439
   Continuous-path mode, 441
   Deleting blocks, 437
General sequence, 433
Inserting a position, 439
Inserting blocks, 436
Motion type, 440
Parameter, 440
Rapid traverse G0, 439
Selecting, 435
Selecting a block, 437
Settings, 442
Traversing block G1, 439
Templates
Creating, 373
Storage locations, 373
Tool
Change type, 344
Delete, 316
Details, 340
Dimensioning, 308
Loading, 317
Measurement, 315
Reactivating, 327
Relocating, 333
Unloading, 317
Tool data
Actual value window, 39
Backing up, 400
reading in, 401
Tool life, 325
Tool list, 310
Tool lists
Settings, 353
Tool management, 303
List filtering, 336
Sorting lists, 335
Tool parameters, 308
Tool types, 306
Tool wear, 325
Tool wear list
Open, 324
Touch operator controls
Cancel alarms, 69
Changing the channel, 69
Touch Panel
calibration, 469
Transformed view, 353

V
Variable screen forms, 422
Virtual keyboard
"SINUMERIK Operate Generation 2" user
interface, 70
HT 8, 467
Virtual keys
ABC keyboard, 71
MCP keys, 71

W
Wear, 325
Wear, summed offset, 325
Widgets, 71
Work offsets
Active WO, 98
Delete, 104
Grinding frames, 101
Overview, 99
Seat offsets, 101
Working area limitation, 119
Workpiece, 367
Starting machining, 143
Stopping machining, 143, 144
Workpiece counter, 205
Workpiece zero
Automatic measurement, 117
Manual measurement, 117

U
Unique cutting edge numbers
Unique, 311
Unit of measurement
switching, 93

Unloading
Multitool, 350
USB drive, 359
User agreement, 87
User dialog
Creation, 453
User interface
Supplementing, 458
User variables, 187
Activating, 195
Back up, 404
Channel GUD, 192
Defining, 195
Global GUD, 191, 195
Global R parameters, 188
Local LUD, 193
Program PUD, 194
R parameters, 189
Searching, 194
Z
Zero offsets
  Displaying details, 102
  Overview, 97
  Settable ZO, 101
  Setting, 95
Zero point
  DXF file, 184
Zero point settings
  Backing up, 400
  reading in, 401