Graphic Programming System
Software Version 6
Drilling/Boring and Milling
Part 2: Operating/Programming Functions

User’s Guide 07.97 Edition

User Documentation
SINUMERIK® Documentation

Printing history

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

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

Status code in the "Remarks" column:

A . . . . New documentation.
B . . . . Unrevised reprint with new Order No.
C . . . . Revised edition with new status.
If factual changes have been made on the page since the last edition,
this is indicated by a new edition coding in the header on that page.

<table>
<thead>
<tr>
<th>Edition</th>
<th>Order No.</th>
<th>Remarks</th>
</tr>
</thead>
<tbody>
<tr>
<td>11.95</td>
<td>6FC5198–6AB00–0BP0</td>
<td>A</td>
</tr>
<tr>
<td>07.97</td>
<td>6FC5198–6AB00–0BP2</td>
<td>C</td>
</tr>
</tbody>
</table>

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

This publication was produced with Interleaf V 5.4

The reproduction, transmission or use of this document or its contents is not permitted without express written authority. Offenders will be liable for damages. All rights, including rights created by patent grant or registration of a utility model or design, are reserved.

© Siemens AG 1995 All rights reserved.

Order No. 6FC5198–6AB00–0BP0
Printed in the Federal Republic of Germany
Preliminary Remarks

The documentation for your Graphic Programming System is broken down into three Guides.

- “Graphic Programming System User’s Guide Part 2: Operating/Programming Functions”
- “Graphic Programming System Environment Description”

This User’s Guide describes the Graphic Programming System WOP for drilling and milling technologies.

This “Graphic Programming System User’s Guide Part 2” describes

- The functions of the individual operations and provides
- Information regarding important points to watch out for when programming with the Graphic Programming System.

This Part 2 assumes that you are familiar with the User’s Guide “Part 1: Programming Examples”.

This User’s Guide describes the Graphic Programming System (WOP) for drilling and milling technologies.

Note

Before using the functions provided by the Graphic Programming System in practice, you should have worked through the programming example in Part 1 and read Part 2 of the User’s Guide carefully.

This User’s Guide does not contain a detailed description of all functions available in the system and of all possible operator inputs.

The graphic programming system features a powerful help function, which is available in almost any operating state.
This “Part 2: Operating/Programming Functions” document is structured as follows:

Section 1: Introduction

This section provides you with an overview.

Section 2: Help Functions

Section 2 describes how to call up directly on the screen descriptions relating to the functions of the softkey menus and drawings for the various parameters as an additional information service.

Section 3: Operation

Section 3 describes the basic principles of operating the graphic programming system and its display.

Section 4: Programming Functions

Section 4 provides guidelines concerning the functions of the graphic programming system. Some of the functions are described in detail.

Section 5: Appendix

The appendix provides an overview of the menu trees of the graphic programming system functions. The User’s Guide describes these functions briefly in the form of a table.

These descriptions are identical with the help texts of the softkey menus.

In addition, the types of workholders and tools are described with the help of a sketch and some parameters.

Further notes

Key symbols

The operator actions described here always assume that you are using the keyboard on the operator panel.

The key symbols in the examples correspond to the symbols on the operator panel.

If you are using the full keyboard for operating the Graphic Programming System, please refer to Section 3 which explains the differences between the key symbols.

System texts

System texts such as softkey texts, names of interactive screenforms or parameter names in interactive screen forms are always written in bold type.

Operator inputs

Operator inputs such as parameter inputs in interactive screenforms are presented in “inverted commas”.
1 Introduction

Starting from the workpiece sketch, you use graphics to define:

- The tools
- The blank contour
- The finished part contour
- Machining

and a part program in DIN code is generated automatically.

You can simulate the programmed traversing movements and display these on the screen.

The part program can be edited at a later time.

You program in different planes, which you define as surfaces.

You always program the surfaces
- in the XY plane, and
- the depth in the Z axis

without reference to a machine.

The system calculates the geometry and machining information at the machining level of the machine only when the Create part program function is activated.

Axis designation

The following axis designations apply to surfaces that are perpendicular to the design plane:

![Diagram showing finished part and surfaces](image)

Bild 1.1 Finished part showing surfaces

---

1) Only if multi-side machining option is available
2 Help Function

The function described below provides you with help in almost every operating situation.
By pressing the help key, you activate the help system which is stored in the programming system.

Help in the Graphic Programming System

- Help texts concerning the current horizontal softkey functions are available if the first softkey of the horizontal softkey menu shows a green “i”.
- You can obtain graphic help for the various input fields in almost all interactive screenforms. They are not marked specially.

Help in the Graphic Programming System

If no interactive screen form is displayed, the help system describes the functions of the horizontal softkeys after this key has been pressed.

Bild 2.1 Graphic help for a help screen

The designations F1 to F7 (see Fig. 2.1) refer to the softkeys of the horizontal menu, e.g.:
- F1 = Point,
- F7 = Edit.
If a **black triangle** appears next to the help text of a softkey, you can press this key to open a further explanatory help screen.

**Close help screen**

Press this key to close the help screen.

**Graphic aid for interactive screenforms**

The graphic help for the individual input fields (see Fig. 2.2) is obtained by pressing the **RECALL** key.

Press the **Help key** to display the individual images again.

![Bild 2.2 Graphic help](image)
3 Operation

This manual mainly describes the operator control concepts and the operator control and display elements of the “Graphic Programming System” (GPS) package.

To practice, we advise you to program a workpiece as given in an example (see “User’s Guide Part 1: Programming Examples”).

The functions of the keys during graphic programming are explained below.

The following sections describe the keys used for

- Interactive screenforms
- Crosshair control and
- Selection

The Graphic Programming System assists operation by supplying information.

Information for operation is displayed below the graphics field.
### 3.1 Interactive screenform control

For programming the workpiece or the tools, parameters are necessary. The Graphic Programming System inserts screenforms in the graphic area.

You are supplied with input fields for entering parameters. You enter these values in interactive mode.

The figure below shows an example.

#### Interactive screenform name

Every interactive screenform has a name which is displayed in the first line. In this example, the name is **Manual milling**.

#### Parameters

Every interactive screenform contains parameters. A parameter consists of a name and an input field for its values.

In this example, the **Total depth** is one such parameter.

#### Input field

An interactive screenform allows parameters to be input. Input fields are emphasized by colors. The values are input through the alpha numeric keyboard. In this example, these are the fields with the values “0.000”.

#### Continuation form

A continuation form allows several entries to be made for a parameter. The existence of a continuation form is indicated by a **black triangle**.

A continuation form can either be an interactive screenform or a selection form.
Text selection field  A text selection field can be recognized by a black triangle pointing to the left.

In this selection field you can toggle between the individual default texts or make a selection in a selection form.

Selection form  A selection form is used for selecting certain specific inputs. The possibilities are given by the system.

In our example, an approach strategy for contour milling.

| direct  (SM1)          |
| over contour normal (SM2) |
| over contour tang. (SM3) |
| Tang. quart.circle (SM4) |
| Tang. circle centre (SM5) |
| Tang. semicircle (SM6)   |

Bild 3.2  Selection form
### Key functions for interactive screenform control

<table>
<thead>
<tr>
<th>Operator panel 840C</th>
<th>Full keyboard</th>
<th>Function in interactive screenforms</th>
</tr>
</thead>
<tbody>
<tr>
<td><img src="home.png" alt="Home key" /></td>
<td><img src="home.png" alt="Home" /></td>
<td>With the <strong>Home key</strong>, you put the cursor in the first input field and accept the parameters that have been input.</td>
</tr>
<tr>
<td><img src="end.png" alt="End key" /></td>
<td><img src="end.png" alt="End" /></td>
<td>With the <strong>End key</strong> you go from input field to input field and accept the parameters that have been input.</td>
</tr>
<tr>
<td><img src="control.png" alt="Control keys" /></td>
<td><img src="4left6right.png" alt="4 left 6 right" /></td>
<td>With these keys, you control the cursor within a field when inputting parameters.</td>
</tr>
<tr>
<td><img src="navigation.png" alt="Navigation keys" /></td>
<td><img src="updown.png" alt="Up 8 Down 2" /></td>
<td>With these keys, you move the cursor from input field to input field and accept the parameters that have been input.</td>
</tr>
<tr>
<td><img src="backspace.png" alt="Backspace key" /></td>
<td><img src="backspace.png" alt="Backspace" /></td>
<td>The <strong>CLEAR key</strong> deletes the character to the left of the cursor.</td>
</tr>
<tr>
<td><img src="pagedown.png" alt="Page Down key" /></td>
<td><img src="pagedown.png" alt="PgDn" /></td>
<td>With this key, you move <strong>forwards</strong> from input field to input field and accept the parameters that have been input.</td>
</tr>
<tr>
<td><img src="pageup.png" alt="Page Up key" /></td>
<td><img src="pageup.png" alt="PgUp" /></td>
<td>If a <strong>black triangle</strong> can be seen to the right of the input field, you open a continuation form with this key.</td>
</tr>
<tr>
<td><img src="function.png" alt="Function keys" /></td>
<td><img src="..." alt="..." />.png</td>
<td>You use the <strong>alphanumeric keys</strong> to enter parameters.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>The <strong>RECALL key</strong> ends interactive screenform input and the system ignores all inputs. The <strong>RECALL key</strong> closes the help screens.</td>
</tr>
<tr>
<td></td>
<td><img src="enter.png" alt="Enter" /></td>
<td>With the <strong>INPUT key</strong> you end screenform inputs and accept all entries.</td>
</tr>
</tbody>
</table>
### Operator panel 840C

<table>
<thead>
<tr>
<th>Full keyboard</th>
<th>Function in interactive screenforms</th>
</tr>
</thead>
<tbody>
<tr>
<td>Shift</td>
<td>With the <strong>Help key</strong>, you call up a screen with help text for each horizontal softkey menu. This contains explanatory texts relating to the various softkey functions. If an interactive screenform is displayed on the screen, you obtain graphical help for the parameters on pressing this key.</td>
</tr>
<tr>
<td>F1</td>
<td></td>
</tr>
<tr>
<td>F3</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Control</th>
<th>If there is an interactive screenform on the screen and the cursor is located in an input field, you call up a <strong>Pocket calculator function</strong> with this key.</th>
</tr>
</thead>
<tbody>
<tr>
<td>Control</td>
<td>With this key combination or key you combine the work schedule steps to form a block under the function <strong>Program steps</strong>.</td>
</tr>
<tr>
<td>N</td>
<td>If the cursor is positioned on a text selection field, you can insert the texts one after the other into this field using this key combination or key. This field is indicated by a <strong>black triangle</strong>.</td>
</tr>
<tr>
<td>N</td>
<td></td>
</tr>
</tbody>
</table>
3.2 Selection control

When working with the graphic programming system you will notice that for some functions operator guidance is provided directly by the keys described below.

The following options are available

- Selection with crosshair,
- Direct selection with cursor keys, with operator guidance
- Selection directly with the cursor keys, without operator guidance.

Information line

The graphic programming system guides you through the programming steps. The operations that you have to carry out are given in an information line below the graphic area.

An exception to this is the function Oriented geometry. This function uses the information line also as a log line.

Selection with the crosshair

With certain functions (such as: Construction geometry/Construct. elements/Line) the programming system changes over to the operator guidance mode in which you need the crosshair.

To select geometry elements on the screen, use the control elements Crosshair.

The diagram below shows an example of this.

The operator guidance in the information line tells you to take certain action. The instruction “Select first reference element” refers to the crosshair.

Bild 3.3 Crosshair control
Key functions

<table>
<thead>
<tr>
<th>Operator panel 840C</th>
<th>Full keyboard</th>
<th>Function with crosshair control</th>
</tr>
</thead>
<tbody>
<tr>
<td><img src="image" alt="Operator panel" /></td>
<td><img src="image" alt="Full keyboard" /></td>
<td><img src="image" alt="Function with crosshair control" /></td>
</tr>
<tr>
<td><img src="image" alt="Function with crosshair control" /></td>
<td><img src="image" alt="Function with crosshair control" /></td>
<td><img src="image" alt="Function with crosshair control" /></td>
</tr>
</tbody>
</table>

- When selecting a geometry element with contour, you need not position the crosshair exactly. The programming system automatically finds the element that is closest to the crosshair.
- When selecting a point element, you must position the crosshair exactly.
Selection with the cursor keys and operator guidance

When working with the Graphic Programming System, you will find that in certain functions (e.g. when milling) the operator guidance runs immediately over the keys described below. These keys select or deselect contours or elements that already exist.

The figure below shows an example of this.

The operator guidance message in the information line tells you what to do.

Selected contours or elements are marked in color.

Key functions

<table>
<thead>
<tr>
<th>Operator panel 840C</th>
<th>Full keyboard</th>
<th>Function with selection control</th>
</tr>
</thead>
<tbody>
<tr>
<td>↑</td>
<td>8 ↑</td>
<td>Select the contour with these keys.</td>
</tr>
<tr>
<td>↓</td>
<td>2 ↓</td>
<td>The selected element or point is accepted by pressing the INPUT key.</td>
</tr>
<tr>
<td>←</td>
<td>Enter 43</td>
<td>By pressing the INPUT key twice, you activate the function that you wanted for the contour (e.g. Face roughing).</td>
</tr>
<tr>
<td>→</td>
<td>Enter 43</td>
<td>With the RECALL key, you end the function and all actions are lost.</td>
</tr>
</tbody>
</table>

Selected contours or elements are marked in color.
Selection with the cursor keys without operator guidance

The function Oriented geometry (see Section 4.6.1) displays the individual elements programmed by you in the information line below the graphic area.

An example is given in Fig. 3.5.

![Diagram](image)

**Bild 3.5  Selection control with cursor keys without operator guidance**

The selected construction elements are displayed in a color.

**Key functions**

<table>
<thead>
<tr>
<th>Operator panel</th>
<th>Full keyboard</th>
<th>Function in interactive screen-form control</th>
</tr>
</thead>
<tbody>
<tr>
<td><img src="image" alt="Left Arrow" /></td>
<td>6 →</td>
<td>With these cursors you move the cursor within the information line. If a contour element can be displayed it is marked in the graphic area.</td>
</tr>
<tr>
<td><img src="image" alt="Right Arrow" /></td>
<td>4 ←</td>
<td></td>
</tr>
<tr>
<td><img src="image" alt="Home" /></td>
<td></td>
<td>You can move the cursor to the starting point with the <strong>Home key</strong>.</td>
</tr>
<tr>
<td><img src="image" alt="End" /></td>
<td></td>
<td>You can move the cursor to the last element with the <strong>End key</strong>.</td>
</tr>
</tbody>
</table>
3.3 Other key functions

The operator control elements described below apply in all operating situations.

<table>
<thead>
<tr>
<th>Operator panel 840C</th>
<th>Full keyboard</th>
<th>Function</th>
</tr>
</thead>
<tbody>
<tr>
<td><img src="image" alt="MACHINE AREA key" /></td>
<td><img src="image" alt="MACHINE AREA key" /></td>
<td>With the <strong>MACHINE AREA</strong> key, you switch over from any menu into the machine area. The operating status of the window now in the background remains as it is.</td>
</tr>
<tr>
<td><img src="image" alt="ETC key" /></td>
<td><img src="image" alt="ETC key" /></td>
<td>You extend the horizontal softkey functions with the <strong>ETC key</strong>.</td>
</tr>
<tr>
<td><img src="image" alt="Area switchover key" /></td>
<td><img src="image" alt="Area switchover key" /></td>
<td>With the <strong>Area switchover key</strong>, you can put the five areas into the horizontal softkey menu at any time.</td>
</tr>
</tbody>
</table>
4 Programming Functions

Section 4 contains
- Information about the WOP file environment
- Programming notes on the functions on the MMC programming level
- Some descriptions of examples of the following WOP functions:
  - Tools
  - Material
  - Geometry
  - Machining
  - Program steps and
- A description of the calculator function.

We recommend you refer to the “User’s Guide Part 1: Programming Examples”.
Starting from a sketch of a workpiece, this Guide documents the individual operations from the definition of the geometry to the creation of the part program.
This “User’s Guide Part 2: Operating/Programming Functions” assumes that you have worked through this example and are familiar with the functions of the example.

4.1 File environment

While you are programming, the Graphic Programming System uses, changes and creates the following:
- Empirical value file (uses)
- Part programs (creates)
- Program steps as a work schedule (creates)
- User magazines (uses, changes, creates)
- Master data catalog (uses, changes)
4.2 Empirical values

Purpose

The empirical value file is an ASCII file made available to the user (programmer) in which settings and program sequences can be stored by setting variables. The file is executed when the system powers up, thus initiating all entered variables. Any changes made to the empirical value file therefore do not have an effect until the system is initialized.

Notes in programming

In addition to the empirical value file there is also a configuring file.

Configuring file

The configuring file is an ASCII file available to the machine manufacturer in which all the machine-dependent data are configured. The file is executed when the system powers up, thus initiating all entered variables. Any changes made to the configuring file therefore do not have an effect until the system is initialized (e.g. with or without multiple surface machining).

Empirical value file

You can edit the empirical value file.

The values for positions, lengths and distances are always entered in "metric".

Anything not located between two vertical lines is interpreted as a comment and is ignored. All instructions in "|" are executed.

For example: Part program no. |V1978 = "1"| right.

Operating sequences

The relevant environment description describes how you

- call up and
- edit

an empirical value file.

You alter the value assignment behind the "=" sign.

For example: |V831 = 2|
**Activating altered empirical values**

If an interactive screenform is displayed on the screen and the cursor is located in an input field you can

Call the calculator function with the **toggle key** and change the empirical value file or configuring file by entering variables, e.g.: \[ V1362 = \ldots \].

In this case, the set variables apply immediately and are not reset until the graphic programming system is terminated.

If you have changed the empirical value file using the editor you can activate the variables by

- performing a cold restart (power on reset) or –
- activating the softkey functions **Read new** or **Read, original**.

The empirical values in a file are listed individually below.
ERFAHRUNGSWERTE und Maskenvorbelegungen für FRAESMACHINEN
/EMPIRICAL VALUES and Screen form defaults for MILLING MACHINES

Programmanfang /Beginning of program:

Programnummern Vorbesetzung /Program number default
[v1978=“1”]

Maschinennamen /Machine name
[v1981=“840C WOP-M”]
[V928=100] mm

Einschaltstellung Ebene /Flexible Plane Selection:
1 = G17 immer bei Fräsen /G17 fixed for Milling!
[V812=1]

Einschaltstellung Nullpunkt–Verschiebung /Zero Offset effective after control power on
1 = G53, 2 = G54, 3 = G55, 4 = G56, 5 = G57
[V831=2]

Wegbedingung für Programmanfang /Type of move at the beginning of program
0 = G00, 1 = G01
[V926=0]

Programmende /End of Program:

Ref.–Position anfahren /Travel to machine reference position
2 = nein /no, 1 = ja /yes
[V859=2]

WKZ –Wechselposition anfahren /Travel to tool change position
2 = nein /no, 1 = ja /yes
[V858=1]

Werkzeug ablegen /Change Tool
2 = nein /no, 1 = ja /yes
[V849=2]

1) The parameters have been taken from the empirical value file of the NC (current date June, 26 1997).
Ein–Ausgabesystem Bedienoberfläche
/Input output system user interface:

Oberfläche: metrisches oder Inch – System
/User interface: Metric or Inch System
 1 = metrisch => Einheit Schnittgeschw. in m/min
 2 = Inch => ft/min
|V1249=1|

NC–Code Ausgabe /NC–code output (1 Inch = 25.4 mm)
 1 = Ausgabe in mm /Output in mm (metric system)
 25.4 = Ausgabe in Inch /Output in inch
|V1412=1|

Standard–Format für NC–Sätze /Standard format for NC blocs
 1 = 3 Nachkommastellen /3 fractional digits
 2 = 4 Nachkommastellen /4 fractional digits
 3 = 5 Nachkommastellen /5 fractional digits
|V1403=1|

Funktionen vor Bearbeitung
/Functions before machining operation:

Positioniergeschwindigkeit /Axis positioning speed
 1 = Eilgang /Rapid traverse
 2 = Vorschub /Programmed feedrate
|V1105=1|

Zeitpunkt der Funktionsausgabe
/Time of execution of auxiliary functions (M,S,T,H)
 1 = keine Ausgabe /no output
 2 = vor Anfahren /before travel
 3 = am Startpunkt /at start position
|V1107=2|

Vorbelegung NPV /Default Zero Offset
 1 = aus /None
 2 = 1. NPV /1st ZO – G54
 3 = 2. NPV /2nd ZO – G55
 4 = 3. NPV /3rd ZO – G56
 5 = 4. NPV /4th ZO – G57
|V1108=2|
Funktionen nach Bearbeitung
/Functions after machining operation:

Positioniergeschwindigkeit /Axis positioning speed
    1 = Eilgang /Rapid traverse
    2 = Vorschub /Programmed feedrate
[V1110=1]

Zeitpunkt der Funktionsausgabe
/Time of execution of auxiliary functions (M,S,T,H)
    1 = keine Ausgabe /no output
    2 = nach Abfahren /after travel
[V1112=2]

Vorbelegung NPV /Default zero offset
    1 = aus /None
    2 = 1. NPV /1st ZO – G54
    3 = 2. NPV /2nd ZO – G55
    4 = 3. NPV /3rd ZO – G56
    5 = 4. NPV /4th ZO – G57
[V1113=2]

Werkzeugeingabe Bohrer+Fräser /Drilling+Milling tool:

Länge Halter /Holder length
[V1326=67] [mm]

Durchmesser Halter /Holder diameter
[V1327=50] [mm]

Durchmesser Schaft /Shaft diameter
[V1328=30] [mm]

Werkzeugauswahl /Chuck definition:

Magazinausgabe
    0 = wie Stammdaten (zweistufig, erst Klasse, dann einzelne Werkzeuge)
    /As per Siemens data (two step, first class, individual tools)
    1 = direkt (einstufig) /1 = direct (one step)
[V1383=1]
Bohren allgemein /Drilling:

Ausgabesteuerung Bohrbearbeitung /Output control drilling
1 = aufgelöste Einzelsätze /Single blocks
2 = Steuerungszyklus /Control cycle
| V1179=2 |

Ausgabesteuerung Zyklus /Output control cycle
1 = als G–Funktion /As G–function
2 = als Unterprogramm /As subroutine
| V1180=1 |

Rückzugsabstand inkrementell zur Bearbeitung /Incremental retraction distance from the machined surface
| V1117=8 | [mm] CR |

Sicherheitsabstand inkrementell zur Bearbeitung /Incremental safety clearance from machined surface
| V1118=5 | [mm] CT |

A–Mass Verrechnung (Bohrerspitze) /A–dimension calculation (tip of drill)
1 = ja /yes  0 = nein /no
| V1181=1 | Bohren u.Zentrieren / Drilling and center
| V1182=1 | Bohren u.Plansenken / Drilling and spotface
| V1206=1 | Tieflochbohren / Deep hole drilling
| V1194=1 | Gewindebohren /Tapping
| V1196=1 | Ausbohren 1 /Bore 1
| V1198=1 | Ausbohren 2 /Bore 2
| V1199=1 | Ausbohren 3 /Bore 3
| V1200=1 | Ausbohren 4 /Bore 4
| V1204=1 | Ausbohren 5 /Bore 5

Verweilzeit auf Bohrtiefe in Sekunden /Dwell at depth in seconds
| V1183=2 | Bohren u.Plansenken / Drilling and spotface  T2
| V1186=3 | Tieflochbohren / Deep hole drilling  T2
| V1190=3 | Gewindebohren /Tapping  T2
| V1195=2 | Ausbohren 1 /Bore 1  T1
| V1197=2 | Ausbohren 2 /Bore 2  T1
| V1203=4 | Ausbohren 4 /Bore 4  T1
| V1205=4 | Ausbohren 5 /Bore 5  T1
**Tieflochbohren** /Deep hole drilling:

- **Erste Bohrtiefe** /First drilling depth \( D_1 \)  
  \[ V1184=5 \] [mm]

- **Verweilzeit am Anfangspunkt in Sekunden** /Dwell at the starting point in seconds \( T_1 \)  
  \[ V1185=2 \]

- **Degression** /Degression \( DR \)  
  \[ V1187=2 \]

- **Bohrstrategie** /Drilling strategy
  - \( 1 = \text{Spänebrechen} /\text{Chip breaking} \)
  - \( 2 = \text{Entspanen} /\text{Stock removal} \)  
  \[ V1188=1 \]

**Gewindebohren** /Tapping:

- **Zuschlag Sicherheitsebene** /Allowance Safety plane \( L_1 \)  
  \[ V1189=4 \] [mm]

- **Drehrichtung Rückzug** /Direction of rotation in retraction
  - \( 1 = \text{automatisch} /\text{automatic} \)
  - \( 2 = \text{rechts} /\text{right} \)
  - \( 3 = \text{links} /\text{left} \)  
  \[ V1191=1 \]

- **Drehrichtung nach Zyklus** /Direction of rotation after cycle
  - \( 1 = \text{automatisch} /\text{automatic} \)
  - \( 2 = \text{rechts} /\text{right} \)
  - \( 3 = \text{links} /\text{left} \)  
  \[ V1192=1 \]

- **Gewindebohren zentrisch** /Tapping concentric
  - \( 1 = \text{ohne Geber} /\text{without encoder} \)
  - \( 2 = \text{mit Geber} /\text{with encoder} \)
  - \( 3 = \text{ohne Ausgleichsfutter} /\text{rigid tapping} \)  
  \[ V1193=2 \]
**Werkzeugwechsel / Tool change:**

WKZ–Wechselposition Fräsen / Tool change position – milling
(Reihenfolge max.4, 0=keine Ausgabe / up to 4, 0=no output)

Position Satz / sequence
- \( V860=110: V850=2 \) 1. Achse / 1st axis
- \( V861=120: V851=2 \) 2. Achse / 2nd axis
- \( V862=130: V852=1 \) 3. Achse / 3rd axis
- \( V863=0 : V853=0 \) 4. Achse / 4th axis

Freier DIN–Code oder Text/DIN–Code bei Werkzeugwechsel
/Free DIN code or Text/DIN code used for tool change
\( V1951=\)"

Additionskonstante für Zeichenmakro bei Referenz– und WZW–Punkt
/Addition constant for character macro with reference and tool change point
\( V928=100 \) [mm]

**Grundanfahrstrategie Bearbeitungen / Basic travelling strategy:**

Bewegung an Startpunkt / Movement from the start point
1 = Ebene–Zustellung / Plane–Location
2 = Zustellung–Ebene / Location/Plane
3 = 3–Achsig / 3–axes
4 = Ebene / Plane
\( V1115=1 \)

Interpretation des angegebenen Abstandes
/ Interpretation of the defined distance
1 = Rückzugsabstand / Retracting distance CR
2 = Sicherheitsabstand / Safety clearance CT
3 = Anfahrabstand / Starting distance DS
4 = Sicherheitsebene / Safety plane SP
\( V1116=3 \)

Anfahrabstand / Starting distance
\( V1119=10 \) [mm]
Zuschlag auf Rohteiloberfläche für Sicherheitsebene
/Addition to the outer blank surface for safety plane
\( V1120=8 \) [mm]
## Contour Machining

### Anfahrstrategie / Travelling strategy

<table>
<thead>
<tr>
<th>Number</th>
<th>Description</th>
<th>Symbol</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>direkt /direct</td>
<td>SM1</td>
</tr>
<tr>
<td>2</td>
<td>konturnormal /vertical to machining direction</td>
<td>SM2</td>
</tr>
<tr>
<td>3</td>
<td>konturtangential /tangential to contour</td>
<td>SM3</td>
</tr>
<tr>
<td>4</td>
<td>tangential über Viertelkreis /tangential to a quarter cycle</td>
<td>SM4</td>
</tr>
<tr>
<td>5</td>
<td>tangential aus Kreismittelpunkt /tangential from a circle center</td>
<td>SM5</td>
</tr>
<tr>
<td>6</td>
<td>tangential über Halbkreis /tangential over a half circle</td>
<td>SM6</td>
</tr>
</tbody>
</table>

\[ V_{1125}=4 \]

### Faktor auf Werkzeugradius für Anfahrabstand / Factor from tool nose radius for the starting clearance

\[ V_{1126}=1.2 \]

\[ AR = V_{1126} \times V_{401} \]

### Faktor auf Werkzeugradius für Anfahrradius / Factor from tool nose radius for starting radius

\[ V_{1127}=1.2 \]

\[ AD = V_{1127} \times V_{401} \]

### Absenkzeitpunkt / Tool lowering

<table>
<thead>
<tr>
<th>Number</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>vor Anfahren /before positioning LB</td>
</tr>
<tr>
<td>2</td>
<td>nach Anfahren /after positioning LA</td>
</tr>
</tbody>
</table>

\[ V_{1128}=1 \]

### Absenkgeschwindigkeit / Tool lowering speed

<table>
<thead>
<tr>
<th>Number</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Eilgang /Rapid traverse</td>
</tr>
<tr>
<td>2</td>
<td>Vorschub /Feed</td>
</tr>
</tbody>
</table>

\[ V_{1129}=2 \]

### Zeitpunkt Fräserradiuskorrektur einschalten / Time instance for considering the milling radial offset

<table>
<thead>
<tr>
<th>Number</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>während Anfahrr position /when at start position</td>
</tr>
<tr>
<td>3</td>
<td>während Anfahren /during approach</td>
</tr>
</tbody>
</table>

\[ V_{1130}=1 \]

### Abfahren von Kontur / Retraction from the contour:

### Afahrstrategie / Retraction strategy

<table>
<thead>
<tr>
<th>Number</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>keine Bewegung /no movement EM1</td>
</tr>
<tr>
<td>2</td>
<td>über Konturnormale /vertical to contour EM2</td>
</tr>
<tr>
<td>3</td>
<td>über Konturtangente /tangential to contour EM3</td>
</tr>
<tr>
<td>4</td>
<td>tangential über Viertelkreis /tangential to a quarter circle EM4</td>
</tr>
<tr>
<td>5</td>
<td>tangential auf Kreismittelpunkt /tangential from the center of circle EM5</td>
</tr>
</tbody>
</table>

\[ V_{1131}=4 \]
Fortsetzung **Konturbearbeitung**

Factor from tool diameter for retraction clearance

\[ |V1132|=1.5 \quad DD=V1132^*V401 \]

Factor from Tool diameter for retraction radius

\[ |V1133|=1.3 \quad DR=V1133^*V401 \]

Abhebezeitpunkt /Lifting time instance

1 = vor Abfahren /before retraction 
2 = nach Abfahren /after retraction

\[ |V1134|=2 \]

Abhebegeschwindigkeit /Lifting feed

1 = Eilgang /Rapid traverse 
2 = Vorschub /Feed

\[ |V1135|=1 \]

Zeitpunkt Abschalten Fräserradiuskorrektur /Time instance for switching off the milling radial offset

1 = während Abfahren /during retraction 
3 = nach Abfahrposition /after retraction

\[ |V1136|=3 \]

Strategie Konturbearbeitung /The contour machining strategy:

Bearbeitungsrichtung /Machining direction

1 = in Konturdefinitionsrichtung 
2 = gegen Konturdefinitionsrichtung

\[ |V1160|=1 \]

Zustellaufteilung /Depth of cut

1 = maximal (angegebene Zustellung) 
2 = gleichmässig

(Berechnete Zustellung <<== angegebene Zustellung)

\[ |V1161|=1 \]

Bearbeitung /Machining

1 = umlaufend /rotary 
2 = pendelnd /oscillating

\[ |V1162|=2 \]

Steuerung der Bahnkorrektur durch das Programmiersystem bei Engstellen, Flaschenhalsproblemen und wegfallenden Radien /Path correction due to problems with short corners and

1 = nein /no 2 = ja /yes 

\[ |V1163|=1 \]

OV 

© Siemens AG 1997  All Rights Reserved  6FC5198--AB00

SINUMERIK 840C (BN)
Fortsetzung **Konturbearbeitung**  
Bearbeitungssseite Konturbearbeitung /Machining of contours  
1 = mitte /middle TP  
2 = links /left  
3 = rechts /right  
4 = innen /inner  
5 = aussen /outer  
\[|V1164|=1|\]

<table>
<thead>
<tr>
<th>Volumenfräsen / Solid milling:</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Absenkstrategie /Lowering strategy</strong></td>
</tr>
<tr>
<td>1 = direkt /direct LM1</td>
</tr>
<tr>
<td>2 = über Schräge /as per slope LM2</td>
</tr>
<tr>
<td>3 = über Spirale /spiral LM3</td>
</tr>
<tr>
<td>[</td>
</tr>
</tbody>
</table>

| Faktor auf Werkzeugdurchmesser für Schrägenlänge /Factor from tool diameter for length of slope |
| \[|V1141|=2|\] |
| BL = V1140 * V401 |
| Schrägenwinkel in Grad /Slope angle in Degrees BA |
| \[|V1142|=30|\] |

| Faktor auf Werkzeugdurchmesser für Spiralradius /Factor from tool diameter for spiral radius |
| \[|V1143|=0.75|\] |
| HR = V1143 * V401 |
| Spiralsteigung in mm/U /Spiral pitch in mm/rev HP |
| \[|V1144|=1|\] |

| Drehrichtung der Spirale /Direction of rotation of the spiral |
| 1 = rechts /right HD |
| 2 = links /left |
| \[|V1145|=1|\] |

| Absenkgeschwindigkeit /Lowering speed |
| 1 = Eilgang /Rapid traverse |
| 2 = Vorschub /Feed |
| \[|V1146|=2|\] |

| Abheben beim Zerspanen /Lifting during machining: |
| **Abhebestategie /Lifting strategy** |
| 1 = direkt /direct RM1 |
| 2 = über Schräge /over slope RM2 |
| \[|V1147|=1|\] |

| Faktor auf Werkzeugdurchmesser für Schrägenlänge /Factor from tool diameter for length of slope |
| \[|V1148|=2|\] |
| BL = V1148 * V401 |
| Schrägenwinkel in Grad /Angle of slope in degrees BA |
| \[|V1149|=30|\] |
Fortsetzung **Volumenfräsen**

**Abhebegeschwindigkeit /Lifting speed**
1 = Eilgang /Rapid traverse
2 = Vorschub /Feed

\[V1150=2\]

**Strategie Zerspanen /Strategies for metal cutting:**
überdeckungsgrad der Fräsbahnen in % /Overlapping angle of the milling cut in %
\[V1167=30\]

Anzahl der Bahnen (0 => vollständig ausräumen) /Number of cuts (0 => removal complete)
\[V1168=0\]

**Gesenkschräge in Grad /Lowering angle in Degrees**
\[V1169=0\]

Maximaler Positionierweg im Vorschub in mm /Maximum axial feed in mm
\[V1170=50\]

**Aufteilung der Zustellung /Depth of cuts**
1 = maximal /maximum
2 = gleichmässig /equal
\[V1171=1\]

**Bearbeitungsrichtung (Umlaufsinn) /Direction of machining (direction of motion)**
1 = Uhrzeigersinn /clockwise
2 = Gegenuhrzeigersinn /counterclockwise
\[V1172=1\]

**Zerspanungsrichtung /Machining direction**
1 = aussen –> innen /outer –> inner
2 = innen –> aussen /inner –> outer
\[V1173=1\]

**Behandlung der Konturstartebenen bei Zerspanung (Inselhöhe) /Evaluation of plane orientation dur. machining (Island height)**
1 = ignorieren /ignore
2 = beachten /consider
\[V1174=1\]
### Überfräsen /End milling:

<table>
<thead>
<tr>
<th>Factor from tool diameter for approach clearance</th>
<th>AD</th>
</tr>
</thead>
<tbody>
<tr>
<td>$V1152=1.2$</td>
<td>$AD=V1152\times V401$</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Factor from tool diameter for approach radius</th>
<th>AR</th>
</tr>
</thead>
<tbody>
<tr>
<td>$V1153=1.2$</td>
<td>$AR=V1153\times V401$</td>
</tr>
</tbody>
</table>

**Abhebestrategie /Lifting strategy**

1. direkt /direct  
2. über Schräge /over slope  

<table>
<thead>
<tr>
<th>RM1</th>
</tr>
</thead>
<tbody>
<tr>
<td>$V1154=1$</td>
</tr>
</tbody>
</table>

**Faktor auf Werkzeugdurchmesser für Schrägenlänge /Factor from tool diameter for length of slope**

<table>
<thead>
<tr>
<th>BL</th>
</tr>
</thead>
<tbody>
<tr>
<td>$V1155=2$</td>
</tr>
</tbody>
</table>

**Schrägenwinkel in Grad /Angle of slope in degrees**

<table>
<thead>
<tr>
<th>BA</th>
</tr>
</thead>
<tbody>
<tr>
<td>$V1156=30$</td>
</tr>
</tbody>
</table>

**Abhebegeschwindigkeit /Lifting speed**

1. Eilgang /Rapid traverse  
2. Vorschub /Feed  

<table>
<thead>
<tr>
<th>RM2</th>
</tr>
</thead>
<tbody>
<tr>
<td>$V1157=2$</td>
</tr>
</tbody>
</table>

**Strategie überfräsen /Strategy for surface milling:**

überdeckungsgrad der Fräsbahnen in %  

<table>
<thead>
<tr>
<th>DC</th>
</tr>
</thead>
<tbody>
<tr>
<td>$V1176=15$</td>
</tr>
</tbody>
</table>

**Umlaufsinn /Direction of rotation**

1. Uhrzeigersinn /Clockwise  
2. Gegenuhrzeigersinn /Counterclockwise  

<table>
<thead>
<tr>
<th>CD</th>
</tr>
</thead>
<tbody>
<tr>
<td>$V1177=2$</td>
</tr>
</tbody>
</table>
Schaltfunktion /Switch functions:

Simulationsstartmodus /Simulation startup mode
  1 = satzweise /block by block
  0 = kontinuierlich /continuous
  \[V1371=0\]

Maskentyp für die Eingabe der Konturparameter
/Screen form type for contour parameter input
  1 = einfacher Parametersatz /Simple parameter set
  2 = voller Parametersatz /Full parameter set
  \[V1488=1\]

Automatisches Auflenden der Hilfemasken
/Automatic display of help forms
  1 = ja /yes  2 = nein /no
  \[V1489=1\]

Einheit für die Rundachse auf einer Mantelfläche
/Unit for the rotary axis on a peripheral surface
  1 = Millimeter /Millimeter
  3 = Grad /Degree
  \[V1501=3\]

Geometrie–Eingabe über Steuerbild mit Richtungspfeilen
/Geometry input via control diagram and arrow directions
  0 = nein /no  1 = ja /yes
  \[V1496=0\]

================================ENDE================================

================================THE END================================
4.3 Graphic programming

The following Figure 4.1 shows the basic level of the Graphic Programming System with a milling part.

The view shown in this figure is a DIN isometric perspective.

Bild 4.1  WOP basic level (example)

**Purpose**

The Graphic Programming System permits multi-side machining 1) of:

- all surfaces on any blank or
- the lateral surface of a cylinder or
- an n corner
- on any surface in the space

using:
- drilling/boring and
- milling

technologies.

You can program the tool and workpiece geometry graphically as well as the technological machining by:

- Defining tools, tool magazines and material lists (*Tools Material*),
- Creating tool geometry (*Geometry*),
- Programming the machining sequence (*Machining*),
- Defining the surfaces (*Select Surface*),
- Graphical simulation of the programmed traversing movements (*Machining Simulation*) and
- Creating a part program (*Create Part Program*).

---

1) Only if multi-side machining option is available
The milling level is always active when programm.WOP is first selected. WOP displays the name of the active surface (plane) in the top right of the graphics area (see Fig. 4.1).

The active surface is displayed in the top right of the graphics area as soon as the geometry has been defined.

If you are to read or recalculate an existing workpiece (for instance, when selecting the Machining function), the Graphic Programming System displays all the defined surfaces in the graphics area.

If you follow the procedure shown below, the Graphic Programming System displays the active surface again.

The coordinate zero point "\(\mathbf{0}\)" in the graphics area is the design point of origin for the workpiece.

This zero point does not represent a defined position within the work area of the NC machine.

The functions Create Surface, Select Surface etc. for multi-side machining are planned by the manufacturer.

Conditions:
The surface selected from the blank or finished part must be
• straight (not curved) and
• perpendicular to the design plane.

Axis designation
The following axis designations apply to surfaces that are perpendicular to the design plane:

Bild 4.2 Finished part showing surfaces
4.3.1 Create surface (define)\(^1\)

**Create surface**

**Purpose**

The function **Create Surface** is used to define surfaces for holes, islands, slots, etc. on an existing blank or finished part contour.

Perpendicular to the design plane (top view of blank/finished part) you define the individual elements as surfaces from a

- cornered contour using the function **From Geometry** (see Fig. 4.3) or
- circular contour (cylinder) using the function **Peripheral Surface** (see Fig. 4.3).

You then program the **Geometry** contours and the appropriate **Machining** technology on the individual surfaces.

**Programming notes**

**Create Surface** involves:

Selecting an element from a defined blank/finished part contour and giving it a name.

Applying **Geometry** and **Machining** functions from the Graphic Programming System to the defined surface.

New surfaces can be created perpendicular to the design plane (top view of blank/finished part) on existing elements of the blank/finished part contours (see Fig. 4.3 below).

---

1) Only if multi-side machining option is available
**Procedure to create a surface on a contour**

**Requirement:**
You have created a blank/finished part contour on the design plane.

1. Press the softkey **Create Surface**.

2. If the surface you want to create lies on a straight element, select the function **Surface from Geometry**.

3. Select the corresponding blank or finished part contour and confirm your selection by pressing the **INPUT key**.

   **Note:**
   This step is omitted if there is only one contour.

4. Now select the element for the new surface.
   The contour of the selected surface is shown in **red**.

5. Confirm your selection by pressing the **INPUT key**.

6. Enter the name in the input fields of the interactive screenform **Plane Definition**.

7. After entering the name, close the interactive screenform by pressing the **INPUT key**.

   Depending on which element was selected, the Graphic Programming System generates the surfaces with the specified values.

   The Graphic Programming System displays a top view of the new surface with the corresponding axis designation (see Fig. 4.2).

   As the contour elements of this surface have a new design plane (not a top view of the blank/finished part), they cannot be defined as new surfaces.
If the surface to be created lies on a circular element, then

To create a cylindrical lateral surface, select the **Peripheral Surface** function and enter the following parameters in the **Peripheral Surface** interactive screenform (example):

![Interactive screenform for a cylindrical peripheral surface](image)

The parameters are described in Fig. 4.6 below.

![Blank or finished part geometry as circle](image)

**Other notes**

Pockets are the only additional surfaces that can be created at present. During machining, the system does not recognize any adjacent surfaces that are islands and will ignore them (see Fig. 4.7).

The following illustration shows an example of a surface created on blank geometry. The surfaces have been created on the front (surfaces...
can only be created perpendicular to the design plane).

**Procedure**

1. Press the softkey **Define Surface**.

2. To define a cylindrical surface, select the **Peripheral Surface** function and enter the appropriate parameters in the **Peripheral Surface** interactive screenform.

Close the interactive screenform by pressing the **INPUT key**.

The Graphic Programming System creates a contour plane, which when viewed from the top corresponds to a cylindrical peripheral surface, from these parameters.
### 4.3.2 Edit a defined surface

**Purpose**

Surfaces that have already been defined can be changed or deleted using the **Edit Surface** function.

**Programming notes**

A surface is changed by screenform inputs.

- The depth and the position in the space of any surface can be edited.
- Changes that you make to a supporting element (finished part contour) have no effect on the surfaces constructed from it.

Surfaces that have already been created are deleted using the **Delete Surface** function.

- A surface can only be deleted when all the **Geometry** and **Machining** created on it has been erased.

### 4.3.3 Select Surface

**Purpose**

**Milling plane** is always the default setting during graphic programming.

Use the **Select Surface** function to select from the menu a surface or plane that was defined using the **Create Surface** function (see section 4.3.1).

This surface can be a side (as a geometry element)
- of the blank or finished part
- or a peripheral surface of a cylinder (see Fig. 4.8).

![Select surface menu](image)

1) Only if multi-side machining option is available
4.4 Graphically creating tools

Purpose

If you want to

- generate new tools,
- add to or change the tool master data,
- create or add tool magazines for your workpiece
- add to or modify your material name list for your workpieces

then use the softkey Tools Material.

Programming notes

The Tools Material function initializes the graphics area.
If you have created a workpiece geometry, the programming system will save it for you first.
If you end the function, the programming system loads the saved geometry for you and you continue with workpiece programming.
This loading operation takes a little while. You can tell from the information line that the system is working.
Tools are organized in the tool master data catalog and in tool user magazines.

Tool master data catalog

This catalog should contain all tools and workholders for your machine environment.
You should create in this all the tools and workholders that you use in your machine in the master data catalog.
For every tool, you define:

- general tool data
- a tool geometry,
- the cutting values for the different materials,

The general tool data contains, for example, an identification number.
Each tool has its own type designation.
You put together the user magazine from this master data catalog.
You determine the values for the magazine location (T No.) and the tool offset (D) in the tool user magazine.

Tool user magazine

Copy the tools and workholders from the master data catalog into a user magazine.
The user magazine must be an image of your machine magazine and contains the tools and the toolholders required for machining.
The values for the magazine location (T No.) and for the tool offset (D No.) must have valid numbers.

If a tool is not suitable for the machining task (e.g.: “Tool too short for cut-off”), you can change or add to the tools in your user magazine.

The following table describes the difference between master data and user magazine.

<table>
<thead>
<tr>
<th>Terms</th>
<th>Master data</th>
<th>User magazine</th>
</tr>
</thead>
<tbody>
<tr>
<td>Standard</td>
<td>SIEMENS AG supplies with the SINUMERIK 840C a catalog with predefined work holders and tools that is intended solely for training and initiation. This catalog is called “WZK”. Tool geometry and cutting values have been selected at random. No executable workpiece programs can be generated with these.</td>
<td>You create a user magazine “Exercise” in the programming example.</td>
</tr>
<tr>
<td>Meaning of T and D Nos.</td>
<td>When creating the tool master data, no T and D Nos. should be assigned. In principle this is possible but only advisable when using tool management via T no. The tool master data can contain any number of tools.</td>
<td>When generating (New magazine) or extending (Copy tool) tool magazines, you must enter the T No. for the magazine location and the D No. or D Nos. for the offset memory into the interactive screenform General tool data that appears on the screen.</td>
</tr>
</tbody>
</table>
### Procedure

**Master data catalog** You wish to extend the master data catalog:

1. You are at the programming level *Machining*.
2. Go into programming level *Tools Material / Master data magazines*.
3. Load the master data catalog with the function *Load master data*.
4. Alter or extend the catalog with the functions *Tool* or *Master data*.
5. As soon as you have saved the catalog, return to the function *Machining*.

**Current user magazine** If a tool is not suitable for a machining task, you can always change or add to the tools in your user magazine.

1. You are at the *Machining* programming level.
2. Go into programming level *Tools Material / Masterdata magazine*.
3. Alter or extend the tool magazine with the functions *Tool* or *Master data*.
4. When you have saved the magazine, return to the *Machining* function.

**Notes** When you return to the menu *Machining*, the programming system asks you whether the altered data are to be saved. You change the tool magazine globally when you do this.

- do not just change the data of the magazine loaded for the corresponding machining operation,
- also change the data of the globally defined tool magazine.

If you load a different or create a new tool magazine during the procedure described above, the user magazine defined for the machining task does not change.

### Table: Terms, Master data, User magazine

<table>
<thead>
<tr>
<th>Terms</th>
<th>Master data</th>
<th>User magazine</th>
</tr>
</thead>
</table>
| ID No.      | The type designation for the tool exists once only. They encode for example:  
- manufacturer identity or  
- order number of the tool or  
- the type of tool, a subgroup, the tool holder type or the diameter grouped together within a type designation.  
See user magazine | See tool master data |
|             |             |               |

![Diagram](image-url)
4.4.1 Extend tool master data catalog

Purpose

The tool master data catalog contains predefined workholders and tools. The master data catalog can be extended at any time to include new tools of your own.

Programming notes

The SINUMERIK 840C is supplied with the master data catalog ("WKZ") in the main directory.

As soon as you extend this catalog using the following description, the Graphic Programming System copies the altered catalog as "WKZ" into the user directory.

From now on, the Graphic Programming System always uses the "WKZ" from the user directory.

Procedure

1. Load master data
2. Select the Tool function
3. Generate new tool then select, one after the other
   - Machining type and
   - Tool class and then define the geometry of the tool or if you want to
- Copy tools
- Change tools or
- Delete tools,
you can then call functions
- All
- T number
- D number or
- ID number
to search for the tools in the current tool magazine and if you are searching for several tools you can
select the tools offered block by block with the toggle key,
confirm with the INPUT key and enter the values one after the other or if you are deleting tools, confirm after the prompt.

4. Once you have generated the tools, press RECALL and then

5. Save master data using the Save master data function.
4.4.2 Generate (new) magazine with (new) tools

Purpose

In principle, you should always generate a new tool in the tool master data and then copy it into the tool (user) magazine.

Programming notes

The function Copy tool only copies tools from the tool master data catalog into the current user magazine.

It is not possible to copy from one user magazine to another.

With the functions, Copy, Create new or Change tool, the Graphic Programming System stores the tool in the magazine.

Procedure

As it is recommended that you always create and edit tools in the tool masterdata and then copy them into the tool (user) magazine, please proceed as follows:

1. You are on programming level Machining.
2. Go to programming level Tools Material / Masterdata magazines.
3. Load the masterdata directory with the function Load masterdata.
4. Edit or extend the directory with the functions Tool.
5. Save the master data
6. Load or create a user magazine.
7. Copy the tools from the masterdata into the user magazine.
8. Change the ID No., D No., T No. or geometry data, if necessary.
9. Save the current user magazine.
When you change tools in the magazine, you can use the functions

- **All**
- **T and D number**
- **Geometry data** or
- **Technology**

...to access the appropriate parameters directly (see Fig. 4.9).

---

**Bild 4.9 Change tool**

The **All** function displays all the interactive screenforms for the tool that is to be changed.

The **T and D number** function enables the name, the T and D number, the identifying text and the feed interpretation to be changed.

The **Geometry data** function enables the geometry data and the insertion location to be changed.

The **Technology** function enables the tool cutting values to be changed.
4.4.3 Tool classes

Purpose

The programming system offers various tool classes and workholder classes with standardized geometries.

Programming notes

The available tool types are described in Section 5.

Procedure

When programming a new tool (New tool) the programming system asks you in the interactive mode:

1. for the tool class
2. the dimensions and
3. the technological parameters.

When you have entered the “geometry data” of the tool, the programming system displays the tool in its machining position.

Bild 4.10 Tool
4.5 Materials and cutting values

When you create tools you assign cutting values for different materials to each tool.

You define the different materials with the function Materials.

With the function Tools, you can, for example, create or alter the tool cutting values.

The cutting values are only suggestions for machining definitions and can be altered whenever necessary.

4.5.1 Materials

The materials are organized in a list of materials. These material designations (see Fig. 4.11) are also shown in a list of materials when you are programming the blank.

The material designation is a user-defined text. At this point, you can decide whether you wish to enter

- individual material names, or
- material groups.
4.5.2 Cutting values

The programming system manages cutting values for 64 materials. The 64 cutting value data are assigned to each tool. The number of different cutting values is limited to the number of materials. The list of materials itself assigns a number to the particular material (Fig. 4.11). You will find this number in the interactive screenform for Cutting values in the line Material under No. (see Fig. 4.12 “Cutting values”).

Programming notes

The No. shows you at which field you are located in the list of materials.

The field Accept enters the parameters and causes the next interactive screenform Cutting values to appear on the screen.

You terminate the entire function by means of the field OK. The function OK does not accept any parameters.

With the fields Forward and Back you page through in the list of materials.
The cutting values are entered in the **Cutting values** interactive screenform (see Fig. 4.12) when you generate or change a tool (**New tool** or **Change tool** functions).

You enter the cutting values under **New tool** or **Change tool** in the interactive screenform **Cutting values**.

Graphic help is available to explain the feed parameters.

The abbreviations for the feed parameters are:

- Plunge feed (FTI = F1),
- Machining feed (FR = F2),
- Withdrawal feed (FTR = F3),
- At the present time the parameter F is not evaluated by the system.

Feeds that you specify with value “0” are not evaluated by the system.
Each drilling and milling machining function gives you the opportunity to modify the cutting values for each programmed machining operation.

As does the function **Edit program step**.

The interactive screenform for the cutting values has additional input fields for:

- Feed in percent (%)
- Data from the magazine

Values entered using the program step editor are retained for the machining stage.

These modifications do not affect the original cutting values; they only apply to the programmed machining.

Each material has its own cutting values.

For each machining operation, the Graphic Programming System compares the cutting values entered for the feed against the original cutting values from the tool magazine and shows them as a percentage (%).

If you are using the values from the tool magazine, the Graphic Programming System enters **100%**.

Use this key to select the cutting values from the original magazine for the **Drilling** or **Milling** functions.
4.6 Geometry

Purpose

You create your workpiece geometry with the help of the function Geometry.

Programming notes

The term “Contour” will be used below in place of “Blank, contour and groove”.

In designing geometry, the Graphic Programming System distinguishes between

- Oriented geometry,
- Standard geometry and
- Construction geometry input.

Oriented geometry

Here, the design elements are entered one after the other by means of a menu guided contour input, from the start point to the end point.

Standard geometry

The graphic programming system offers frequently used geometries as standard geometries (e.g. rectangle, circle, ...).

Construction geometry

If contour elements in the drawing are not directly dimensioned completely and can be determined only by means of relative relationships to other contour elements, then construction elements are needed.

The auxiliary geometry function is provided for such cases.

You can enter any construction elements in any sequence and then produce a contour.

The various softkey functions and menu trees under the softkey Geometry are given in the Appendix (Section 5).
4.6.1 Oriented geometry with contour elements

Purpose
You define a workpiece contour with menu-guided element input. The function **Oriented geometry** creates contours.

Programming notes
Fig. 4.14 below shows an example of the programming function **Oriented Geometry**:

*Bild 4.14  Oriented geometry function*
When creating geometries for oriented and construction geometries, you can define global contour transitions. The system therefore automatically displays a screenform.

As these transitions are not always but rather seldom required, the display of this screenform is controlled from software version 6.3 on. A field enabling this control is therefore available in the screenforms for blank and finished part.

The field global contour transitions is initialized with "no", which corresponds to the majority of the cases.

Contours are edited in the same way.
When you have called up the function **Oriented geometry**, enter the parameters for the global contour transitions.

![Interactive screenform Global contour transitions](image)

**Bild 4.17** Interactive screenform *Global contour transitions*

Fig. 4.18 below illustrates the concept of left/right bends:

![Left and right bends](image)

**Bild 4.18** Left and right bends

Every corner of the contour is assigned the entered global transitions (chamfer or rounding).

If, at a later stage, you wish to change a particular transition in an interactive screenform, the programming system overwrites the global transition at this position.

Globally defined transitions
- are not activated until the **OK** softkey has been pressed.

Locally defined chamfers/roundings are not affected by **Edit/Global transitions**.
You define the parameters for the initial point of the geometry you are designing in the interactive screenform **Initial point**. The dimensions always refer to the current coordinate zero.

If you wish to enter the initial point as polar coordinates, you must delete the values entered for X and Y before you call up the screenform "Initial point polar".

In the empirical value file you can define via the variable “V1196” whether you wish to program 

“V1496 = 0”

or program the geometric elements via the horizontal softkey bar with the functions

- **Line right**, (with angle 0°)
- **Line left**, (with angle 180°)
- **Line up**, (with angle 90°)
- **Line down**, (with angle 270°)
- **Line any**, (with any angle)
- **Arc left (ccw)**, 
- **Arc right (cw)**.

“V1496 = 1”

with the cursor keypad

The following Fig. 4.20 appears in the graphic field.
You can create the geometry elements one after the other by pressing the relevant cursor keys.

**Note**
To move in the log line you must close the displayed keypad display with the **ESC key** and then position the cursor on a new element. You can activate the keypad again with the function **Geometry elements**.

**Contour elements**
A contour consists of contour elements and contour transition elements. Starting with the initial point of the contour, you join the elements together or insert elements.

The elements are:

- initial point
- straight lines
- arcs

At the beginning you can add the following transition elements to the above elements

- chamfer
- rounding.
There are two ways of defining a contour consisting of the elements **Straight line** or **Arc**:

- Minimum parameter input with a **simple parameter set** or
- Input of all parameters using the **full parameter set**

The following table provides an overview of the possible individual parameters:

<table>
<thead>
<tr>
<th>Element</th>
<th>Simple Parameter Set</th>
<th>Full Parameter Set</th>
<th>Graphical Parameter Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Straight line</td>
<td><img src="image1" alt="Diagram of Straight line" /></td>
<td><img src="image2" alt="Diagram of Full Parameter Set" /></td>
<td><img src="image3" alt="Diagram of Straight line Contour" /></td>
</tr>
<tr>
<td>Arc</td>
<td><img src="image4" alt="Diagram of Circular arc" /></td>
<td><img src="image5" alt="Diagram of Full Parameter Set" /></td>
<td><img src="image6" alt="Diagram of Arc of circle Contour" /></td>
</tr>
</tbody>
</table>

The **absolute** dimension refers to the current coordinate zero.

The **incremental** dimension refers to the end point of the last element to be defined.

If you have created the contour and programmed **Machining**, the Graphic Programming System ignores the machining instruction if the contour is altered or erased.

If these parameters do not fully describe the contour element, the system calculates the remaining unknown parameters from adjacent contour elements (where possible).
**Parameter input**

The following table provides some information about the parameters:

<table>
<thead>
<tr>
<th>Item</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Cartesian/polar parameters</strong></td>
<td>Elements can be defined as Cartesian or polar elements. <strong>Note:</strong> If you change the method of input while defining the elements, you must delete all the user inputs, i.e. all values without a “+”, “−” or “=” suffix plus the transition angle AT, together with chamfering and rounding. The input fields of the coordinate system you did not select are information fields and cannot be overwritten.</td>
</tr>
<tr>
<td><strong>Information fields</strong></td>
<td>The interactive screenform of a contour element contains information fields (gray) with default parameters. These values have been calculated by the programming system and cannot be edited.</td>
</tr>
</tbody>
</table>
| **Input fields**            | You define open (unwritten) input fields with the usual parameters. Parameters calculated by the system are indicated as such in the input fields. These are:  
  • “=” exact value  
  • “+” value rounded up  
  • “−” value rounded down |
<p>| <strong>Calculated parameters</strong>   | Values from the programming system that are shown in the display fields with “=”, “+” or “−” cannot be overwritten. Only when you delete the parameter that the system uses to calculate these values and the input fields are empty can something be entered in the input fields again. |
| <strong>Values for diameter</strong>     | The value for the X axis is always represented as a diameter. This applies to both $E_X$ and $dE_X$.                                                                                                         |
| <strong>$E_X$ (end point X), $E_Z$ (end point Y)</strong> | The dimension for the parameters $E_X$ and $E_Y$ are always expressed as an absolute value and refers to the current coordinate zero (not the initial point $IP$).                                            |
| <strong>$dE_X, dE_Y$</strong>            | The dimensions for the parameters $dE_X$ and $dE_Y$ are incremental and refer to the end point of the last element to be defined.                                                                           |</p>
<table>
<thead>
<tr>
<th>Item</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Rounding, Chamfering</td>
<td>Rounding or chamfering is entered directly into the interactive screenform. Rounding or chamfering refers to the contour transition to the previous element. The predefined global transition (see next page) will be overwritten.</td>
</tr>
<tr>
<td>Angle data</td>
<td>Angle data always refers to the direction of the element. If the <strong>Transition AT</strong> field is blank, the Graphic Programming System calculates the transition. The following diagrams explain the <strong>Transition AT</strong> parameter:</td>
</tr>
<tr>
<td>Contour transition element</td>
<td>Transition elements are chamfering and rounding and refer to the previous element.</td>
</tr>
</tbody>
</table>
**Information line**

The individual elements that you have programmed are displayed in the information line below the graphics area.

This gives you an overview of the elements that have already been input. Note the following with respect to the information line:

- The abbreviated designation of the elements in the information line are:
  - **IP** = Initial point of the oriented geometry or contour
  - **LI** = Line
  - **AW** = Arc right
  - **AL** = Arc left

- **Colors**
  - **Yellow** Start point
  - **Blue** Defined element
  - **White** Incompletely defined element (contour element cannot be represented)
  - **Red** Contour element selected by the cursor
  - **Gray** Elements behind the current contour element

- **Insert**

The programming system inserts the next contour element to the right of the cursor that points to a contour element.

If you position the cursor on one of the gray colored contour elements, the programming system joins the inserted elements to the following elements.
• Operation:
The following table describes the function of the keys in the information line:

<table>
<thead>
<tr>
<th>Keys</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>These keys control the position of the cursor within the information line. Insofar as a contour element is already displayable, you will see this element highlighted in the graphics area. If you want to insert a contour element, use these keys to move the cursor to the appropriate position in the information line.</td>
</tr>
<tr>
<td></td>
<td>Use this key to position the cursor on the initial point.</td>
</tr>
<tr>
<td></td>
<td>Use this key to position the cursor on the last element.</td>
</tr>
<tr>
<td></td>
<td>If the cursor is positioned on a geometry element, you will be able to: use the <strong>INPUT key</strong> to call up the appropriate interactive screenform for the element and modify the parameters.</td>
</tr>
<tr>
<td></td>
<td>If the cursor is positioned on a contour element, you will be able to: erase the element using the <strong>Erase key</strong>.</td>
</tr>
<tr>
<td></td>
<td>If a contour element is marked with a &quot;*&quot; (e.g.: &quot;GE&quot;) and the cursor is currently positioned on this element, you will be able to: press this key to display the variants in the graphic area.</td>
</tr>
</tbody>
</table>
In addition to the key combinations described on the previous page for editing the elements, you can modify the parameters accordingly using the procedure described below.

You can use the **Edit elements** softkey function to
- Change or erase elements

**Procedure:**

<table>
<thead>
<tr>
<th>Keys</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Edit elements</td>
<td>1. You can modify elements and the contour with the function <strong>Edit elements</strong>.</td>
</tr>
<tr>
<td></td>
<td>2. You select the element to be edited with the cursor keys and operate either the function <strong>Change element</strong> or <strong>Erase element</strong>.</td>
</tr>
<tr>
<td>Change element</td>
<td>3. You edit the parameters of the selected element in the relevant interactive screenform with the function <strong>Change element</strong>.</td>
</tr>
<tr>
<td>Erase element</td>
<td>You erase the selected element with the function <strong>Erase element</strong>.</td>
</tr>
</tbody>
</table>

- **Select variants**

When defining a contour, it can occur that two variants are offered as alternatives from the given parameters for the shape of the contour.

The Graphic Programming System displays these elements with a "*" (e.g.: "*BR") in the information line.

**Procedure:**

<table>
<thead>
<tr>
<th>Keys</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Edit elements</td>
<td>1. Select the function <strong>Edit elements</strong>.</td>
</tr>
<tr>
<td></td>
<td>2. Use the cursor keys to place the cursor on the element whose variants you wish to change.</td>
</tr>
<tr>
<td></td>
<td>This element is indicated by a &quot;*&quot; in the information line.</td>
</tr>
</tbody>
</table>
3. You can change your selection with the softkey **Select variant**. Operate this function.

<table>
<thead>
<tr>
<th>Key</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Select variant</td>
<td>You can change your selection with the softkey <strong>Select variant</strong>. Operate this function.</td>
</tr>
<tr>
<td>Green</td>
<td>The Graphic Programming System displays the construction variants in different colors (&quot;green&quot; and &quot;yellow&quot;).</td>
</tr>
<tr>
<td>Yellow</td>
<td>4. You make your selection by pressing either the Green or the Yellow softkey.</td>
</tr>
</tbody>
</table>

**Error messages**

The following table contains information regarding the error messages:

<table>
<thead>
<tr>
<th>Error message</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>“Parameter contradiction”</td>
<td>It is possible that the geometry element definition does not coincide with the elements that precede or follow this element (e.g.: when <strong>editing</strong> a contour definition).</td>
</tr>
<tr>
<td></td>
<td>Perhaps you have selected different coordinate systems and the parameters do not match or If, for example, you want to edit the initial point AP, then:</td>
</tr>
<tr>
<td></td>
<td>1. Reset the input fields of the parameters for the following element (de-lete with the space bar).</td>
</tr>
<tr>
<td></td>
<td>An undefined element in the information line is displayed in “white” by the Graphic Programming System.</td>
</tr>
<tr>
<td></td>
<td>2. Edit the new parameters for the IP.</td>
</tr>
<tr>
<td></td>
<td>3. Edit the known parameters for the following element.</td>
</tr>
</tbody>
</table>
4.6.2 Construction geometry

**Purpose**

If you do not make use of standard geometry (rectangle, circle, n corner) for generating a workpiece contour, or if the dimensioning of the individual contour elements is not complete, then the Graphic Programming System offers the possibility of constructing any workpiece contour made up of construction geometry elements. The **Construction geometry** softkey is provided for this purpose.

In this section, all construction possibilities of point, line and circle are listed in tabular form.

**Programming notes**

**Construction geometry elements**

For a contour construction of your own design, you first of all build up construction geometry elements, that is points, lines and circles.

You place the contour on these construction geometry elements.

More than forty possible definitions are available for this purpose.

Lines have no initial and end points, in other words no boundary is given, and circles are always full circles.

The angles can be entered as required.

The sequence of programming the construction elements does not have to be the same as the later sequence in the contour.

**Contour**

The direction of the contour is also specified later when the contour definition is made. You require an initial point and an end point which you also have to define beforehand. The initial point can be identical to the end point.

You cannot save the designed geometry with the construction elements until contour definition has been completed. Automatic saving also takes place only after contour definition has been completed.

If you require intermediate storage of construction elements generated, you create a minimum contour (point, line, point) on the elements and press the **OK** softkey to end the design. You can then save the geometry together with its construction elements and continue the definition with the **Geometry/edit** function.
Select zero point  
Value inputs always refer to a reference point. For example, the workpiece reference point or another previously defined reference point.
When the Select zero point key is pressed, the workpiece zero point is selected.

The function Numeric input refers to the workpiece zero point.

Angle  
The following origin applies for angle definition:

90° or –270°
180° or –180°
0° or 360°
270° or –90°

Bild 4.21  Angle definition

The size of the angle increases counterclockwise.

Interactive screenform Global contour transitions  
When you have called up the function Oriented geometry, enter the parameters for the global contour transitions.
See page 4-38 for a description of other parameters.

Changing the infeed  
To machine arc–in sections you must be able to change the infeed coordinates within a path contour.

To do this, you can call the function Change infeed under the function Create contour and then enter the value in a screenform.
### Tables

The tables below list all possible design types.

The table below describes the designing possibilities for **points**.

<table>
<thead>
<tr>
<th>Help graphics</th>
<th>Definition</th>
<th>Explanation</th>
</tr>
</thead>
<tbody>
<tr>
<td><img src="image1.png" alt="Image" /></td>
<td><strong>Point</strong> = Reference point, horizontal value, vertical value</td>
<td>You design a point starting from a reference point by entering the horizontal and vertical values: 1. Select the reference point 2. Enter <strong>numeric</strong> values for horizontal (X value) and vertical (Y value)</td>
</tr>
<tr>
<td><img src="image2.png" alt="Image" /></td>
<td><strong>Point</strong> = Reference point, distance, angle</td>
<td>You design a point starting from a reference point by entering the distance and the angle: 1. Select the reference point 2. Enter the <strong>numeric</strong> values for distance and angle</td>
</tr>
<tr>
<td><img src="image3.png" alt="Image" /></td>
<td><strong>Point</strong> = 2 reference lines</td>
<td>You design a point of intersection of two reference lines: 1. Select first reference line 2. Select second reference line</td>
</tr>
<tr>
<td><img src="image4.png" alt="Image" /></td>
<td><strong>Point</strong> = 1 reference line, 1 reference circle, location</td>
<td>You design a point of intersection of a reference line and a reference circle: 1. Select reference line 2. Select reference circle 3. Select the appropriate one from the solutions offered for the point (location)</td>
</tr>
</tbody>
</table>

---

4 Programming Functions
4.6.2 Construction geometry
<table>
<thead>
<tr>
<th>Help graphics</th>
<th>Definition</th>
<th>Explanation</th>
</tr>
</thead>
</table>
| ![Diagram](Bild 4.24) | Point = 2 reference circles, location | You design a point of intersection of two reference circles:  
1. Select first reference circle  
2. Select second reference circle  
3. Select the right point from the solutions offered (location). |
| ![Diagram](Bild 4.24) | Point = 2 reference points | You design a point in the middle between two reference points:  
1. Select first reference point  
2. Select second reference point |
| ![Diagram](Bild 4.25) | Point = 1 reference point, 1 reference line | You design a mirrored point at a reference line:  
1. Select the reference point to be mirrored  
2. Select the reference line to the used for mirroring  
3. Press OK to confirm the interactive form opened. |
| ![Diagram](Bild 4.25) | Point = 1 reference circle | You design the center of a reference circle:  
1. Select reference circle  
2. Select and confirm **numeric** values |

_Bild 4.24_  
_Bild 4.25_
The table below describes the designing possibilities for **lines**.

<table>
<thead>
<tr>
<th>Help graphics</th>
<th>Definition</th>
<th>Explanation</th>
</tr>
</thead>
<tbody>
<tr>
<td><img src="image" alt="Line = 2 reference points" /></td>
<td>Line = 2 reference points</td>
<td>You draw a line through two reference points: 1. Select first reference point 2. Select second reference point</td>
</tr>
<tr>
<td><img src="image" alt="Line = 1 reference point, angle" /></td>
<td>Line = 1 reference point, angle</td>
<td>You design a line with a specific angle through one reference point: 1. Select reference point 2. Enter <strong>numeric</strong> value for angle</td>
</tr>
<tr>
<td><img src="image" alt="Line = 1 reference line, distance, location" /></td>
<td>Line = 1 reference line, distance, location</td>
<td>You design a line parallel to the reference line: 1. Select reference line 2. Enter <strong>numeric</strong> value for distance 3. Select the correct one from the offered solutions for the line (location)</td>
</tr>
<tr>
<td><img src="image" alt="Line = 2 reference lines" /></td>
<td>Line = 2 reference lines</td>
<td>You design a mirrored line: 1. Select first reference line about which mirroring is required 2. Select second reference line which is to be mirrored</td>
</tr>
<tr>
<td>Help graphics</td>
<td>Definition</td>
<td>Explanation</td>
</tr>
<tr>
<td>---------------</td>
<td>------------</td>
<td>-------------</td>
</tr>
</tbody>
</table>
| ![Line = 1 reference point, 1 reference line, angle](diagram1) | Line = 1 reference point, 1 reference line, angle | You design a line through a reference point with an angle referred to a reference line:  
1. Select reference point  
2. Select reference line  
3. Enter **numeric** value for angle. |
| ![Line = 1 reference point, 1 reference circle, location](diagram2) | Line = 1 reference point, 1 reference circle, location | You design a line through a reference point with this line touching a reference circle:  
1. Select reference point  
2. Select reference circle  
3. Select the correct one from the solutions offered for the line (location) |
| ![Line = 1 reference circle, angle location](diagram3) | Line = 1 reference circle, angle location | You design a line which touches a reference circle at a certain angle:  
1. Select reference circle  
2. Enter **numeric** value for angle  
3. Select the correct one from the solutions offered for the line (location) |
| ![Line = 2 reference circles, location](diagram4) | Line = 2 reference circles, location | You design a line which is a tangent to two reference circles:  
1. Select first reference circle  
2. Select second reference circle  
3. Select the correct one from the solutions offered for the line (location) |
<table>
<thead>
<tr>
<th>Help graphics</th>
<th>Definition</th>
<th>Explanation</th>
</tr>
</thead>
<tbody>
<tr>
<td><img src="image1.png" alt="Image" /></td>
<td>Line = 1 reference line, 1 reference circle, location</td>
<td>You design a line parallel to the reference line and tangential to the reference circle: 1. Select reference line 2. Select reference circle 3. Select the correct one from the solutions offered for the line (location).</td>
</tr>
<tr>
<td><img src="image2.png" alt="Image" /></td>
<td>Line = 1 reference point, horizontal value, vertical value, angle</td>
<td>You design a line, displaced horizontally and vertically starting from a reference point, and an angle: 1. Select reference point 2. Enter horizontal (X) and vertical (Y) numeric values for the displaced point and the angle at which the line runs through the point.</td>
</tr>
<tr>
<td><img src="image3.png" alt="Image" /></td>
<td>Line = 1 reference point, 2 x horizontal values, 2 x vertical values</td>
<td>You design a line starting from a specific reference point through two points by: 1. Selecting the reference point 2. Entering the numeric values for the two points through which the line runs. X value = horizontal Y value = vertical</td>
</tr>
</tbody>
</table>
The table below describes the designing possibilities for circles.

<table>
<thead>
<tr>
<th>Help graphics</th>
<th>Definition</th>
<th>Explanation</th>
</tr>
</thead>
</table>
| ![Circle](image1) | Circle = 1 reference point, radius | You design a circle around a reference point with a certain radius:  
1. Select reference point  
2. Enter numeric value for radius. In this case, the X and Y values remain 0. |
| ![Circle](image2) | Circle = 1 reference point, 1 reference line, radius location | You design a circle through a reference point, the circle being tangential to a reference line and enter a radius:  
1. Select reference point  
2. Select reference line  
3. Enter numeric value for radius  
4. Select the correct one from the solutions offered for the circle (location) |
### 4.6.2 Construction geometry

<table>
<thead>
<tr>
<th>Help graphics</th>
<th>Definition</th>
<th>Explanation</th>
</tr>
</thead>
</table>
| ![Image](circle_2_reference_lines.png) | Circle = 2 reference lines, radius, location | You design a circle tangential to two reference lines:
1. Select first reference line
2. Select second reference line
3. Enter **numeric** value for radius |

<table>
<thead>
<tr>
<th>Radius</th>
<th>0.0000</th>
</tr>
</thead>
</table>

*Bild 4.33*

4. Select the correct one from the solutions offered for the circle (location)

| ![Image](circle_1_reference_line.png) | Circle = 1 reference line, 1 reference circle, radius, location | You design a circle tangential to a reference line and a reference circle:
1. Select reference line
2. Select reference circle
3. Enter **numeric** value for radius |

<table>
<thead>
<tr>
<th>UK</th>
<th>Radius</th>
<th>0.0000</th>
</tr>
</thead>
</table>

*Bild 4.34*

4. Select the correct one from the solutions offered for the circle (location)
<table>
<thead>
<tr>
<th>Help graphics</th>
<th>Definition</th>
<th>Explanation</th>
</tr>
</thead>
</table>
|               | Circle = 2 reference circles, radius, location | You design a circle tangential to two reference circles:  
1. Select first reference circle  
2. Select second reference circle  
3. Enter **numeric** value for radius  

![Bild 4.35](image1)  
4. Select the correct one from the solutions offered for the circle (location)  

|               | Circle = 1 reference circle, radius difference | You design a circle which is concentric to the reference circle:  
1. Select reference circle  
2. Enter **numeric** value for the radius difference (+/−).  

![Bild 4.36](image2) |
### Circle = 1 reference point, horizontal value, vertical value, radius

Starting from a reference point, you design a new center and the corresponding circle:

1. Select reference point
2. Enter the **numeric** values for the radius of the circle as well as for the horizontal (X) and vertical (Y) displacement of the center referred to the reference point

<table>
<thead>
<tr>
<th>Radius</th>
<th>0.0000</th>
</tr>
</thead>
<tbody>
<tr>
<td>X value</td>
<td>0.0000</td>
</tr>
<tr>
<td>Y value</td>
<td>0.0000</td>
</tr>
</tbody>
</table>

*Bild 4.37*

### Circle = 2 reference points

You design a circle with a reference point as its center and the second reference point as its radius:

1. Select first reference point as center
2. Select second reference point as radius
3. Press **OK** to confirm the interactive form displayed or enter other radius as **numeric** value

<table>
<thead>
<tr>
<th>Radius</th>
<th>0.0000</th>
</tr>
</thead>
</table>

*Bild 4.38*
<table>
<thead>
<tr>
<th>Help graphics</th>
<th>Definition</th>
<th>Explanation</th>
</tr>
</thead>
</table>
| Circle = 1 reference circle, 1 reference line | You mirror a circle at a reference line:  
1. Select reference circle to be mirrored  
2. Select reference line to be used for mirroring  
3. Confirm by pressing OK the interactive screenform offered under numeric. In this case the value for the radius remains 0. |
| ![Circle Diagram](image1) | ![OK](image2) | ![Radius](image3) |
| Circle = 1 reference circle, 1 reference line | You design a circle tangential to a line and a circle:  
1. Select reference circle  
2. Select reference line  
3. Enter numeric radius  
4. Select the correct one from the solutions offered for the circle (location) |
| ![Circle Diagram](image4) | ![OK](image5) | ![Radius](image6) |
| Circle = 2 reference points, radius, location | You design a circle which runs through two reference points:  
1. Select first reference point  
2. Select second reference point  
3. Enter numeric value for radius  
4. Select the correct one from the solutions offered for the circle (location) |
<p>| <img src="image7" alt="Circle Diagram" /> | <img src="image8" alt="OK" /> | <img src="image9" alt="Radius" /> |</p>
<table>
<thead>
<tr>
<th>Help graphics</th>
<th>Definition</th>
<th>Explanation</th>
</tr>
</thead>
<tbody>
<tr>
<td><img src="image1.png" alt="Circle" /></td>
<td>Circle = 1 reference point, 1 reference circle, radius, location</td>
<td>You design a circle running through a reference point and tangential to the reference circle: 1. Select reference point 2. Select reference circle 3. Enter <code>numeric</code> value for radius</td>
</tr>
<tr>
<td><img src="image2.png" alt="Circle" /></td>
<td>Circle = 1 reference point, 1 reference line</td>
<td>You design a circle with a reference point as its center and tangential to a reference line: 1. Select reference point 2. Select reference line 3. Press <code>OK</code> to confirm the interactive form offered or enter <code>numeric</code> value for another radius of the tangential circle</td>
</tr>
</tbody>
</table>

*Bild 4.41*

4. Select the correct solution from those offered for the circle (location).

*Bild 4.42*
<table>
<thead>
<tr>
<th>Help graphics</th>
<th>Definition</th>
<th>Explanation</th>
</tr>
</thead>
</table>
| Circle = 1 reference point, 1 reference circle | You design a circle with a reference point as its center and tangential to a reference circle:  
1. Select reference point  
2. Select reference circle  
3. Press **OK** to confirm the interactive form offered or enter numeric value for other radius of the tangential circle. |

![Diagram of circle with one reference point and one reference circle](image1.png)

| Circle = 3 reference points | You design a circle which runs through three reference points:  
1. Select first reference point  
2. Select second reference point  
3. Select third reference point |

![Diagram of circle with three reference points](image2.png)

| Circle = 2 reference points, 1 reference line, location | You design a circle running through two reference points and tangential to a reference line:  
1. Select first reference point  
2. Select second reference point  
3. Select reference line  
4. Select the correct one from the solutions offered for the circle (location) |

![Diagram of circle with two reference points and one reference line](image3.png)
<table>
<thead>
<tr>
<th>Help graphics</th>
<th>Definition</th>
<th>Explanation</th>
</tr>
</thead>
</table>
| Circle = 2 reference points, 1 reference circle, location | You design a circle which runs through two reference points and is tangential to a reference circle:  
1. Select first reference point  
2. Select second reference point  
3. Select reference circle  
4. Select the correct one from the solutions offered for the circle (location) | |
| Circle = 1 reference point, 2 reference lines, location | You design a circle which runs through a reference point and is tangential to two reference lines:  
1. Select reference point  
2. Select first reference line  
3. Select second reference line  
4. Select the correct one from the solutions offered for the circle (location) | |
| Circle = 3 reference lines, location | You design a circle which is tangential to three reference lines:  
1. Select first reference line  
2. Select second reference line  
3. Select third reference line  
4. Select the correct one from the solutions offered for the circle (location) | |
| Circle = 1 reference point, 1 reference line, 1 reference circle, location | You design a circle which runs through a reference point and which is a tangent to a circle and a line:  
1. Select reference point  
2. Select reference line  
3. Select reference circle  
4. Select the correct one from the solutions offered for the circle (location) | |
<table>
<thead>
<tr>
<th>Help graphics</th>
<th>Definition</th>
<th>Explanation</th>
</tr>
</thead>
<tbody>
<tr>
<td><img src="circle.png" alt="Circle" /></td>
<td>Circle = 1 reference point, 2 reference circles, location</td>
<td>You design a circle which runs through a reference point and touches two reference circles as a tangent: 1. Select reference point 2. Select first reference circle 3. Select second reference circle 4. Select the correct one from the solutions offered for the circle (location)</td>
</tr>
<tr>
<td><img src="circle2.png" alt="Circle" /></td>
<td>Circle = 2 reference lines, 1 reference circle, location</td>
<td>You design a circle which touches two reference lines and a reference circle as a tangent: 1. Select first reference line 2. Select second reference line 3. Select reference circle 4. Select the correct one from the solutions offered for the circle (location)</td>
</tr>
<tr>
<td><img src="circle3.png" alt="Circle" /></td>
<td>Circle = 1 reference line, 2 reference circles, location</td>
<td>You design a circle which touches a reference line and two reference circles as a tangent: 1. Select reference line 2. Select first reference circle 3. Select second reference circle 4. Select the correct one from the solutions offered for the circle (location)</td>
</tr>
<tr>
<td><img src="circle4.png" alt="Circle" /></td>
<td>Circle = 3 reference circles, location</td>
<td>You design a circle which touches three reference circles as a tangent: 1. Select first reference circle 2. Select second reference circle 3. Select third reference circle 4. Select the correct one from the solutions offered for the circle (location)</td>
</tr>
</tbody>
</table>
4.6.3 Standard geometry

**Purpose**
You can call frequently used blank and contour elements such as

- Rectangles
- Circles
- n corners

in the graphic programming system with the function **Standard geometry**.

If you want to create a contour for a groove, the system offers you

- Elongated hole
- Rectangular groove
- Annular groove
- n–corner groove
- Radial groove
- Circular groove

as a groove **standard geometry**.

**Programming notes**

**n–corners**

The outside circle of an n–cornered groove always runs through the corner points, not as a tangent to the rounded corners (see Fig. 4.44).
4.6.4 Holes

**Purpose**
When creating a group of hole points, you have to know a few special things:

- creating a user-defined group of hole points
- skipping of hole points in a group

**Programming notes**

**Creating a user-defined group of hole points**

A user-defined hole pattern is made up of

- various individual points
- rows of points
- circles of points
- boxes of points.

**Define point elements**

Before creating the user-defined hole pattern, generate a construction point from the elements line or circle as described under **Construction geometry**. You generate a point group directly with the functions row of points, circle of points or point box.

**Define hole pattern**

With the function **Define hole pattern** you create a hole pattern on the previously constructed point elements as described under **Define contour** (construction geometry).
Procedure

Define point elements

1. Go into the programming level Geometry/Create new/Holes/Any hole pattern.
2. Create point elements under the softkey function Const. geo elements.
3. You obtain the point elements with the functions
   - Point
   - Line
   - Circle
   - Row of points
   - Circle of points
   - Point box
4. Under the softkey function Define pattern you produce a hole pattern on the point elements.

Define hole pattern

1. Select Define pattern.
2. Select point elements.
3. The softkey function Accept confirms the entire pattern of hole points.
4. The softkey function OK ends definition of the hole pattern.

Skipping hole points

1. Go into the programming level Geometry/Create new/Holes/Any hole pattern.
2. Press the softkey function Define pattern.
3. Select a point structure.
4. The selected group of hole points are marked in red by the programming system.
5. The information line requests you to “Skip single points from the point pattern”.
6. You select the single point by positioning the crosshair on the point which is to the skipped.
7. The point is skipped by pressing the INPUT key. The red highlighting is switched off.
Inverse

8. If you activate the function **Inverse** the programming system reverses the active/passive holes.

**Note**

This function helps you if there are a large number of holes but only a few of these are required.

9. The function **Accept** ends input of the point pattern.

When the starting plane has been defined you create the geometry as described for the function **Holes**.

4.6.5 Groove

**Purpose**

The functions for the groove are described in section 5.

**Programming notes**

When you use the **Oriented geometry** function to create a groove, you create the center line of the groove.

The system generates two contour elements, the

- the center line of the groove and
- the groove itself.

When you process the groove, select the corresponding geometrical element in this case.
4.6.6 Copy contour or elements

**Purpose**
Select the function **Copy** if you want to
- move,
- mirror,
- reverse mirror or
- scale contour elements or complete contours
or if you want to
- program a parallel contour.

The figure below shows the programming level Geometry/Copy.

![Geometry/Copy diagram](image)

*Bild 4.45 Copy*

**Programming notes**
The softkey functions and the information line guide you through the operating sequence.
The function **Copy** either relates to
- complete contours or
- individual contour elements.

*Delete copied contour* If you make multiple copies of a contour (e.g. **mirror**) and then decide to delete one of them (**Delete geometry** function), you will initially only be able to delete the most recently copied contour, etc.
Procedure

Copying complete contour definitions

1. Select the contour with the cursor keys.
2. Confirm your selection with the INPUT key.
3. End your selection with INPUT if you do not wish to enter any more contour definitions or elements.
4. Enter the parameters in the interactive screenform Copy and confirm with INPUT.

Copy contour elements

1. Select the function Contour definition start and select an element with the Crosshair + INPUT.
   This element is the beginning of the contour definition that is to be copied.
2. Select the function Contour definition end and select a final element with the crosshair + INPUT.
3. Confirm your selection with the INPUT key.
4. Enter the parameters in the interactive screenform Copy that appears.
5. Confirm the parameters with the INPUT key.
4.7 Machining

Purpose

You use the Machining softkey function when you

- machine the programmed workpiece geometry with the technologies drilling/milling.

Programming hints

You machine standard contours or user-defined contours with 2 1/2 D milling or drilling.

Interactive screenforms

Some of the parameters in the interactive screenforms have default values. The programming system obtains these values from an empirical values file (see Section 4.2, Page 4–2).

Select the contours or holes to be machined with the selection control.

This section provides you with programming information for:

- Milling technology
- Drilling technology
- Programming the insert blocks
- Simulation
4.7.1 Select surface

Before you program the milling or drilling operation with the softkey function Machining, select the relevant surface.

Select a defined surface or plane from a selection menu under the function Create surface (see Section 4.3.1).

The graphic programming system displays a screenform Position plane change (see Fig. 4.46).

Note

Descriptions are given in the help display to help you to understand the parameter Output.

The parameters "+" or "-" in Side machined according to process A – B – C define which rotary axes are used to machine on the selected plane.

The machine manufacturer determines whether rotary axis definition refers to the workpiece or the tool.

1) Only if multi-side machining option is available
4.7.2 Magazine for tools

**Purpose**

For the various machining tasks to be performed on a workpiece you use a magazine which contains tools which you then use for the machining tasks.

A magazine can either be a user magazine or the master data magazine.

**Programming notes**

In the interactive screenform **Magazine** displayed on the screen you define whether you want to load tool master data or a user magazine.

The user magazines already contain the correct T and D values.

If you are using **master data** then add the relevant values for magazine location T and tool offsets D in the screenform **General tool data**.

If tools are missing in the magazine displayed, you can alter or add to the contents at any time on programming level **Tools Material/Master-data Magazines**.

You can replace the loaded magazine with another with the function **Edit program step** (see Section 4.9.2, Page 4–118).
4.7.3 Beginning of program

![Interactive screen form Beginning of program](image)

**Purpose**

Enter

- Program type
- Program number
- Comments about the part program
- Zero offset for the machine zero\(^1\)

in the interactive screen form **Beginning of program**.

**Programming notes**

The **program type** and **program no.** information together make up the program name.

<table>
<thead>
<tr>
<th>Main program</th>
<th>MPF3</th>
</tr>
</thead>
<tbody>
<tr>
<td>Program no. 3</td>
<td>SPF100</td>
</tr>
<tr>
<td>Subroutine</td>
<td></td>
</tr>
<tr>
<td>Program no. 100</td>
<td></td>
</tr>
</tbody>
</table>

Information about the **drawing no.** etc. appear as comments in the part program.

The Graphic Programming System creates the part program with the function **Create part prog.**

With the function **Create part program** the Graphic Programming System converts the geometry and machining information to the positions.

---

1) Only required for multi-side machining with turned part.
### 4.7.4 Tool selection

Select the function **Tool selection**.

When you select tools, you can use the following functions:

- **All**
- **T number**
- **D number**
- **Id. number** or
- **Geometry data**

To call up the corresponding tool directly (see Fig. 4.50).

The **All** function selects all the tools in the magazine.

The functions **T, D** or **Id. number** enable a tool to be selected directly on the basis of its **T, D** or **id. number**.

The **Geometry data** function enables a particular tool to be selected for the machining of special geometry on the basis of a search criterion (see Fig. 4.51).
Evaluation of tool insertion length

In the current system, it is possible to use tools (mills, drills) for machining with feed values exceeding the insertion length of the tool.

This can cause collisions at the machine. The insertion length of the tools is therefore monitored as from software version 6.3.

If a fault occurs, the message

"Insertion length of the tool < machining depth"

is output.

Machining is then not permitted.
4.7.5 Milling

Purpose

Various types of machining are available to you here:

- Contour milling
- Solid milling
- Solid milling parallel to the axis
- Manual milling
- Surface milling
- Surface milling parallel to the axis

Programming notes

You can specify the approach and retract strategy of the milling tool for each function.

This User’s Guide looks in detail at the input parameters and the available command sequences in the following sections.

Contour milling

You machine selected contours with cutter radius compensation (CRC) G41/G42 if this is required.

Solid milling

You machine selected pockets, islands or both.

The programming system computes the paths without cutter radius compensation (CRC) G40.

If the selected pocket contains islands that must be considered, these must also be selected. Pockets with different end planes cannot be machined in one working step. Islands with different start planes can be machined together. In this case it is possible to choose whether or not to include the start plane in the machining process.

Holes or cycle reference points that have already been programmed can be selected as insertion points.

Solid milling parallel to the axis

The Graphic Programming System computes the cutting paths for the pockets, islands (or both) parallel to the axis.

Manual milling

The cutting operations for a pocket follow a specific strategy. You guide the tool center path either by means of the crosshair or by entering coordinates.

Surface milling

You machine the surface of a selected contour. The pockets can also be machined. The programming system computes the paths parallel to the contour.

Surface milling parallel to the axis

The Graphic Programming System computes the cutting paths for the surface to be machined parallel to the axis.
Start point for contour milling

You can define a start point for machining closed contours or sections of contours.

Machine contour section for contour milling

You define an initial and/or end element for a contour section.

Individual sections cannot be machined from geometry that you have created using the function Standard geometry.

Procedure

1. Go into the programming level Machining/milling.
2. Select a tool.
3. Select a type of machining.

4.7.6 Contour milling

Purpose

You have created the various contours under the function Geometry in the following programming modes:

- Standard geometry
- Oriented geometry
- Construction geometry

If you used the Standard geometry function to create the contour, the Graphic Programming System considers the contour to be closed.

If you used the Construction geometry or Oriented geometry function to create the contour, the Graphic Programming System considers the contour to be an individual geometry element.

The following section describes the Contour milling interactive screenform. As far as the Contour milling function is concerned, the parameters in this interactive screenform enable you to:

- Modify the cutting parameters.
- Insert DIN blocks before and after machining.
- Determine the approach and retract strategy of the tool in respect of the workpiece contour.
- Define the technological parameters for the machining process.
Programming notes

Standard geometry

The programming system computes the paths for a closed contour. You can define the start point by means of the crosshair.

Oriented geometry and construction geometry

The user can decide whether the programming system computes

- closed contours,
- open contours or the
- contour sections (elements).

Start point standard

As a standard function, the programming system stores start points for the beginning of the contour. Unless you have specified differently, the system begins to process the contour at this point.

The Graphic Programming System recognizes whether you created the contour using the Standard geometry function or from individual elements (Oriented geometry).

The system determines where the start point of the machining is from the relevant geometry.

<table>
<thead>
<tr>
<th>Geometry</th>
<th>Start points standard</th>
</tr>
</thead>
<tbody>
<tr>
<td>Circle</td>
<td>0°</td>
</tr>
<tr>
<td>Rectangle</td>
<td>Left bottom</td>
</tr>
<tr>
<td>n corner (without angle)</td>
<td>Left bottom</td>
</tr>
<tr>
<td>Elongated hole</td>
<td>Left</td>
</tr>
<tr>
<td>Rectangular groove</td>
<td>Left bottom</td>
</tr>
<tr>
<td>Annular groove</td>
<td>0°</td>
</tr>
<tr>
<td>n corner groove</td>
<td>Left bottom</td>
</tr>
<tr>
<td>Radial groove</td>
<td>0°</td>
</tr>
<tr>
<td>Circular groove</td>
<td>0°</td>
</tr>
<tr>
<td>Oriented geometry</td>
<td>Initial point (defined manually)</td>
</tr>
<tr>
<td>Construction geometry</td>
<td>Start point (defined manually)</td>
</tr>
</tbody>
</table>

Contour direction

In Oriented geometry and Construction geometry the contour follows the direction in which you have constructed it.

In the Standard geometry the contour traces a path to the left (counterclockwise).

Simulation

When simulating contour milling, the Graphic Programming System ignores the activation and deactivation of cutter radius path compensation.
Contour milling

interactive screenform

Cutting values

The individual parameters are described below.

You filled in the cutting values interactive screenform (see page 4-33) when creating the tool.

The material of the workpiece has been taken into account.

The material used for the blank is the default material.

If you wish to alter these values again, use this key to display the Cutting values interactive screenform and modify the parameters (refer as well to page 4-34 in section 4.5.2).

These changes to the cutting values only apply to the most recent machining operation.

If you want to retain these values for future use, you must modify and resave the data in the magazine using the Tools Material function.

If necessary, you can add a DIN block to the programmed milling operation (see page 4-107).

This block could, for example, be a comment.

You determine the approach strategy of your tool in respect of the 1st start point following a tool change or the last machining operation. The options are

- XY/Z (Traversing path – – – in Fig. 4.55)
- Z/XY (Traversing path —— in Fig. 4.55)
- XYZ (Traversing path – · · · – in Fig. 4.55)
- XY (Positioning in the plane).

The default value XY/Z is stored in the empirical values file.
The following illustration shows the approach options for the 1st start point.

![Diagram showing approach options for 1st start point]

The default value **XYZ/Z** is stored in the empirical values file.

**Safety distance CT**
The **safety distance CT** is the distance between the finished part contour and the tool.
Distances during approach/withdrawal

The following diagram in Fig. 4.56 explains in detail the possible ways of approaching the first start point up to the start plane PS. The system takes into account either

- Withdrawal distance CR (from the retract strategy) or
- Safety distance CT (from the approach strategy) or
- Approach distance DS or
- Safety plane SP.

Notes regarding the parameters CR, CT, DS, and SP

The approach distance DS and the safety distance SP can be edited in the contour milling interactive screenform.

The approach distance DS must be greater than or equal to the safety distance CT.

The safety distance CT can be edited in the interactive screenform Approach contour (see Fig. 4.57, page 4-82).

The retract distance CR can be edited in the interactive screenform Withdraw from contour (see Fig. 4.60, page 4-84).

If the string “####.###” is displayed, this input field cannot be edited.

After the first start point, the cutter always travels to the safety distance CT so as not to exceed the number of NC blocks in the part program.

In the case of “centered” milling, the system always takes the approach distance DS and the safety plane SP into account.
The following diagram in Fig. 4.57 describes the parameters of the **Approach contour** interactive screenform and the ways of approaching the contour.

The approach distance AD is the approach line on, for example, a quarter circle.

You edit the safety distance CT in this Approach contour interactive screenform (refer as well to Fig. 4.56, page 4-81).
You have two options
- Lower before approach and
- Lower after approach.

The following figures illustrate these strategies:

Bild 4.58  Lower before approach

In the case of contour milling “inside, outside or left/right”, the cutter radius correction (CRC) is active. You must make sure that the cutter does not collide with the workpiece during its approach or as it is lowered.

CRC on

You can influence the approach strategy with
- CRC on while approaching position (indicated by (1) in Fig. 4.58 and 4.59)
- CRC on during lowering (indicated by (2) in Fig. 4.58 and 4.59)
- CRC on during approach (indicated by (3) in Fig. 4.58 and 4.59)
Withdraw from contour

The following diagram in Fig. 4.57 describes the parameters in the Withdraw from contour interactive screenform and the withdrawal options.

You edit the retract distance CR in this Withdraw from contour interactive screenform (refer as well to Fig. 4.56, page 4-86).
Technological parameters

Fig. 4.61 describes the technological parameters of the interactive screenform Contour milling.

<table>
<thead>
<tr>
<th>Start plane PS</th>
<th>0.0000</th>
</tr>
</thead>
<tbody>
<tr>
<td>End plane PE</td>
<td>-20.0000</td>
</tr>
<tr>
<td>Allowance cont. AC</td>
<td>8.0000</td>
</tr>
<tr>
<td>Allowance base AB</td>
<td>8.0000</td>
</tr>
<tr>
<td>Infeed depth IF</td>
<td>16.0000</td>
</tr>
<tr>
<td>Tool position TP centre (tool)</td>
<td></td>
</tr>
</tbody>
</table>

$\text{n} = \text{Traversing}$

$\text{CT} = \text{Safety distance}$

$\text{SP} = \text{Safety plane}$

Bild 4.61 Technological parameters for contour milling

Start plane PS, End plane PE

The start and end planes of a machining operation always refer to the selected solid.

You determined the parameters for the start and end planes when you created the geometry. The Graphic Programming System enters these parameters in the input fields.

In the case of different start and end planes, the default values for the input fields are taken from the highest start plane and highest end plane.

Bild 4.62 Start and end planes

Infeed depth IF

The default infeed value is derived from the maximum cutting depth of the tool.
**Tool position**

The **Tool position** selection field offers you a choice of positions for the milling tool:

- **center (on)**
- **left (le)**
- **right (re)**
- **inside (in)**
- **outside (ou)**

Fig. 4.63 below illustrates the use of these parameters.

In the case of individual elements, the choice of tool position is between **left (le)** and **right (re)**.

In the case of closed contours, there are additional options of **inside (in)** and **outside (ou)**.
**Path milling strategy** *(contour milling)*

The following illustration explains in detail the parameters in the **Path milling strategy** interactive screenform.

---

**Note regarding Narrow point offset OV parameter**

In the case of **Narrow point offset Yes**, the cutter will only go as far into the narrow point without damaging the contour as is possible on account of its radius.

If the milling tool is larger than the narrow point and you selected **Narrow point offset No**, the Graphic Programming System will generate an error message.
Procedure

1. Go into the programming level Machining/Milling.
2. Select a tool.
3. Go into the programming level Contour milling.
4. Select the contour.
5. You can define a start point if you do not want to use the default point as the start point.
6. If the Graphic Programming System detects that the selected contour was generated using the Construction geometry or Oriented geometry functions, you can define individual contour elements as start or end elements.
7. Enter parameters in the interactive screenform Contour milling.

Programming notes on Contour milling

You will now learn how to graphically program the machining of the closed contour or of individual contour sections with the Contour milling type of machining.

To make it easier to trace the direction of the contour, you can switch on the direction arrows. To do this, press the vertical softkey Representation/Direction arrows on/off.

A tool is required for machining. You select a tool and activate the function Contour milling.

Bild 4.65 Information line, contour milling
Information line

The information line below the graphics area guides when programming.
You select a contour.

Start point

If you do not use the start point given by the programming system then define the Start point of the tool with this key.

Select the start point for the X/Y coordinates manually with the crosshair.
The system takes the Z coordinate from the programmed contour parameters.
The INPUT key sets the start point selected with the crosshair and ends contour selection.

Note

The programming system identifies the contour element that is closest to the setpoint.

Bild 4.66 Contour milling

You are offered graphic help for every parameter.
In the interactive screenform Contour milling the parameters have defaults taken from the empirical values file.

You can overwrite all parameters.

Finally, confirm the parameters that have been entered. The Graphic Programming System computes and displays the travel paths.

Contour milling of part contours

Contours whose geometry you have created under the functions Oriented geometry or Construction geometry can be broken down into
individual contour elements. You then process these contour elements separately from the complete contour. Various hints on programming are given below.

You have selected a tool and activated the function Contour milling.

You are requested to select a contour. You have generated the geometry of the contour under Construction geometry or Oriented geometry and you have defined the start point yourself.

**Note**

The programming system considers the trace of the contour starting from the start point of the geometry.

**Red elements**

Elements shown in red are elements that are machined in contour milling.

**Blue elements**

Elements of the geometry shown in blue are elements which are not machined with contour milling.

With the functions **Start element** or **End element** you define the contour elements that you are machining.

You select contour elements with the crosshair.

Confirm the selection by pressing the **INPUT key**

and press the **INPUT key** again to end element selection.

In the interactive screenform Contour milling you parameterize the machining operation and

confirm the parameters.

The programming system computes and displays the travel paths for the Contour milling type of machining.

**Note**

If the programming system displays the milling traversing movements on the wrong side of the contour, change the sign in the **Radius** input field.

Sequence of operation:

1. Open interactive screenform Contour milling.
2. Open continuation screenform Approach strategy.
3. Make changes in input field Radius.
4.7.7 Solid milling

**Purpose**

The **Solid milling** function machines selected pockets and islands. The Graphic Programming System computes the tool center path (without tool radius offset).

**Programming notes**

Select a tool and confirm by pressing the softkey **Solid milling**.

Program the machining operation in the **Solid interactive screenform**.

---

**Bild 4.67 Solid milling**

Some parameters have already been explained in the previous pages. This User’s Guide does not therefore go into any greater detail at this point.

The "see Fig. ...Page..." lines in Figure 4.67 refer to the pages where these parameters are described.
The Graphic Programming System prompts you to select the contours that are to be used during solid milling.

If, for instance, the selected solid contains an island, it will have to be selected.

Pockets with different end planes are only machined up to the highest end plane (refer as well to the Island height strategy parameter, Fig. 4.75).

Islands of different heights (start plane) can be machined together. This enables start planes to be included in the machining operation.

The Graphic Programming System always detects the highest start and end planes from all the selected contours.

Figures 4.68 and 4.69 below show the stock removal operation in the case of pockets or islands with differently defined start and end planes or geometry.

You have now

- Selected all pockets and islands and
- Set the **Island height strategy** parameter.
If the external contour is a blank contour, the blank contour overshoots as far as the tool radius.

If the distance between an island and the blank contour is less than the tool radius, the tool will not travel around the island and the island will have to be reworked.

If the external contour is a finished part contour, the tool travels up to the contour.

An external pocket must be present so that automatic pocket and island detection can be performed.

The Lower parameter field enables you to select the

- Lower directly
- Lower over slope
- Lower over spiral

functions and assign the relevant parameters.

Figure 4.70 below shows the Lower during solid milling interactive screenform.

In the case of solid milling where the external contour is not a blank, you can select already programmed holes or cycle reference points as insert holes for the lowering operation. The insert holes must be selected together with the relevant pockets and/or islands.

The chosen lowering strategy, the tool and the contour must all be taken into account when selecting the position of the points to ensure that no collisions occur during the lowering operation.
In the case of a **direct** lowering strategy, you should only use tools that are designed for insert operations and also use an insert hole if appropriate.

Select a relief hole (or reference point) to define the start point of the machining on the contour in such a way that a contour violation does not occur during “Lower over spiral”.

The **Raise** parameter field enables you to select the

- Raise directly
- Raise over slope

functions and assign the relevant parameters.

Figure 4.71 shows the **Raise during solid milling** interactive screenform.

![Bild 4.71 Raise directly or over slope](image)

**Strategy**

You open the **Stock removal strategy** interactive screenform shown below from the **Strategy** parameter field.

![Bild 4.72 Stock removal strategy](image)
Max. positioning distance with feed

The maximum positioning distance FD applies to the feed (G01) only when removing stock.

It has no effect during approach or withdrawal (see Fig. 4.73).

![Bild 4.73 Max. positioning distance with feed](image)

Machining direction

The Stock removal strategy interactive screenform allows you to specify the machining direction as

- Clockwise
- Counter-clockwise
- Machining direction in → out
- Machining direction out → in

Figure 4.74 below shows the Stock removal strategy interactive screenform and the Help screen for the machining direction.

![Bild 4.74 Machining direction strategy](image)

If the external contour is a blank contour, you should select the Machining direction out → in strategy.

The system then lowers the tool outside the blank contour and approaches the contour.

The Machining direction in → out strategy should be used in the case of pockets that have a finished part as an external contour.
### Island height

The highest start and end planes of all the selected contours (see Figs. 4.68 and 4.69) are used in the machining proposition.

Figure 4.75 below describes the **Island height** parameter.

![Bild 4.75 Island height](image)

You must select all the affected islands.

### 4.7.8 Solid milling parallel to the axis

#### Purpose

The **Solid milling parallel to the axis** function removes stock from selected pockets and islands parallel to the selected axis.

The contours are selected in the same way as in Solid milling parallel to the contour.

#### Programming notes

See section 4.7.7 Solid milling for a description of the parameters.

![Bild 4.76 Solid milling parallel to the axis](image)
4.7.9 Manual milling

Purpose
You remove stock (machine) according to your own strategy.

You define:

1. The approach and exit strategy in the interactive screenform Manual milling (see Fig. 4.77) and

2. The movements of the milling tool.

Using the preassigned parameters in the interactive screenform Manual milling and the first entered point, the Graphic Programming System creates the blocks for the approach and exit movements as a DIN part program.

If you want to define your own approach and exit strategy, you must reset the predefined parameters in the interactive screenform Manual milling to ZERO ("0").

You can define the machining with the functions

- Feed rate,
- Rapid traverse,
- Numeric input and
- with the crosshair.
You are in the menu tree **Machining/Milling**.

Select a tool, and press the softkey **Manual milling**.

**Bild 4.77 Manual milling**

You parameterize the machining operation in the interactive screenform **Manual milling**.

**Comments on the input fields**

**Predefined parameters**

The predefined parameters affect the machining strategy and must be set to ZERO ("0") if necessary.

**Cutting values**

You make entries in the interactive screenform for cutting values by opening a continuation screenform in the input field "Cutting values".

Use the **INPUT key** to confirm the parameters for the cutting values.

**Total depth = 0, infeed = 0**

The programming system positions the tool on the manually defined points.

You can move the tool in three dimensions.

You machine the stock down to the end plane in accordance with your own strategy.

**Total depth > 0, infeed > 0**

You specify a **Total depth** and the value for the various **Infeeds**.

The traversing movements that you have entered manually for the first plane are shifted by the programming system for each infeed until the end plane is reached.

**Note**

Numerically entered positional values can be traversed either at rapid traverse rate or at feedrate.
4 Programming Functions

4.7.9 Manual milling

Comments

You describe the machining operations. This text appears as a comment in the work schedule.

Having accepted the parameters, enter manually the traversing movements of the cutter.

The function **Abort** cancels the entire operation.

The function **OK** accepts the graphically programmed operations.

The traversing conditions of the tool are set using the functions **Feed** and **Rapid traverse**.

This key erases the last manual traversing movement.
You specify any positions that you require as well as the reference plane for the depth by means of the Numeric input softkey.

**Start and machining coordinates X, Y, Z**

There are two possibilities for inputting the start coordinates. These two possibilities are:

1st possibility

You specify the start and machining coordinates (X, Y, Z) directly.

2nd possibility

You control the X/Y start position with the crosshair.

The Graphic Programming System takes the Z value entered in Numerical input and the predefined values from the interactive screenform Manual milling (see Fig. 4.77).

You confirm with the INPUT key.

**Procedure**

The information line guides you in graphic programming.

The programming system displays a crosshair on the screen in the graphics area.

1. Go into the programming level Machining/Milling/Manual milling.
2. Select the functions Rapid traverse or Feed.
3. Using the Crosshair or numeric inputs, approach the start and target points. The programming system draws a tool track with tool diameter.
4. Confirm these points with the INPUT key.
5. With the softkey function OK, end machining.
4.7.10 Surface milling

You machine the surface of a selected contour, selected islands or selected blank.

The tool travels over the selected contours by a cutter path overlap amount.

The programming system computes the paths parallel to the contour. Machining always takes place from the outside to the inside.

The values for the approach distance are taken from the programming system from the empirical values file. The empirical value ensures correct infeed of the cutter.

The surface area shown in the diagram above will be surface milled as the next example.

Before beginning with the Surface milling function, you must include a face miller in the tool magazine Exercise.

1. Go into the programming plane Machining/Milling/Surface milling.
2. Select the contour.
3. Enter parameters in the interactive screenform Surface milling.

Select the required tool, the face miller, for surface milling.

Having chosen the cutter, activate the milling function Surface milling and select the blank contour.
The interactive screenform **Surface milling** appears with the default parameters from the empirical values file.

![Surface milling: Interactive screenform](image)

You are offered graphic help for each parameter.

You enter the parameters as well as the approach and retraction strategy and accept by pressing the **INPUT key**.

**Note**

You can follow the machining operation (approach and retraction strategy) on the screen by means of the **Machining simulation** function.

The system computes the cutter paths parallel to the contour. You can see that the blank contour has been overlapped at the edge with the cutter path overlap of 30%. As a result, the corners of very acute angled contours may not be completely machined.

Remachining is possible with the **Manual milling** function.
4.7.11 Surface milling parallel to the axis

**Purpose**

The **Surface milling parallel to the axis** function is used to machine the surface parallel to the selected axis.

The contours are selected in the same way as in Surface milling parallel to the contour.

**Programming notes**

For a description of the parameters, see section 4.7.10 Surface milling.

---

Bild 4.81 Surface milling parallel to the axis
4.7.12 Drilling

Purpose

You drill with various drilling cycles. These are the cycles L81 to L89.

The programming of the standard cycles is described separately in the “SINUMERIK 840/840C Cycles Programming Instructions”.

Programming notes

You are in the programming level Machining/Drilling.

Select the required drilling technology, for example Deep hole.

Select the tool required for machining.

Having selected the drill, activate the drilling function with the softkey Value input and select the hole.

Bild 4.82 Drilling technology

You see an interactive screenform Deep hole drilling.

You are offered graphic help for each parameter.

You enter the parameters as well as the approach and retraction strategy and accept with the INPUT key.
Note

You can follow the machining operation on the screen with the approach and retraction strategy under the function Machining simulation.

Procedure

1. Load a magazine with the required drilling tool using the function Load magazine.
2. Go into the programming level Machining/Drilling.
3. Select a drilling technology.
4. Select a tool if necessary.
5. Activate the function with Value input.
6. Select the hole.
7. Enter the parameters in an interactive screenform.
8. Supply the interactive screenform for the cycle with the necessary parameters.
4.7.13 Machine and auxiliary functions

**Purpose**
The function **Machine aux. fct.** enables you to use your own applications.

**Skip NC Blocks**
The Graphic Programming System offers an application (macro), which can be used to skip blocks.
The macro is called **106 skippable blocks**.

In the part program, this macro sets a code "/" or resets it.

**Programming notes**
In the part program, the Graphic Programming System marks the NC blocks concerned with "/".

**Programming example**
The following NC blocks are an example for skipping of measuring cycles:

```
%MPF1
.
.
N10 G00 D12 X104 F0.8 ;
/N15 R11=0 R22=1 ... L972 ;"/" skip measuring cycle
/N20...
/N25...
N55 G01 X85 F0.6 ;skip terminated
```

**Procedure**
1. Change to the programming level **Machining**.
2. Select the function **Machine aux. fct.**.
3. If several macros are available, select the appropriate one with the **cursor keys** and the **INPUT key**.
4. Enter the necessary parameters in the interactive screenform.
5. Carry out the next program steps for workpiece programming.
4.7.14 Insert blocks

With the function **Insert block (DIN)** you can include instructions in graphic programming.

You make entries in the input fields in the interactive screenform **Insert block** corresponding to the DIN code that you expect in the part program.

In the interactive screenform the system distinguishes between:

- insert blocks (labeled with “1” in Fig. 4.84) whose position the Graphic Programming System takes account of and simulates as well as
- unassigned DIN code (labeled with “2” in Fig. 4.84) which the Graphic Programming System ignores (does not simulate).

The following functions can appear as a DIN block in the part program:

- Zero offset
- Position at feed rate or rapid traverse
- Spindle direction of rotation and speed
- Oriented spindle stop
- Coolant on/off
- DIN code

The system adds a current block number to each block.

With the selection field on the right next to the various input fields, you decide whether you insert the DIN command.

The meaning of this character is:

- **minus “–”** = no output
- **plus “+” or “1,2”** = output in the block
You enter a sequence number (e.g.: “1” or “2”) for the positions, e.g. X / Z.

This number specifies the order in which you wish to move the various axes X and Z (coordinates).

For example, enter

X = “1” and Z = “2” in field “No.”.

and the programming system will generate two blocks:

“N05 X... L_F”
“N10 Z... L_F”

Enter X = “1” and Z = “1” and the output is in one block

“N05 X... Z... L_F”

In these input fields you can define rotary axis movement (absolute or relative, via G90/G91). The rotary axes can only be traversed in a single block each if they are defined with relative positioning.

The movements programmed in the insert blocks form are not simulated.

All linear axis movements refer to the current plane.

Up to the input field Unassigned DIN code, the programming system interprets the inputs.

Two lines are provided for the input field Unassigned DIN code. If both these lines are filled, the programming system converts these into two NC blocks.

If you have marked further input fields as being active, the first line of the input field Unassigned DIN code is attached at this block.

"N... M04 Unassigned DIN code L_F”

The second line is always the second

The following applies for the inserted blocks in the Graphic Programming System:

- The Graphic Programming System transfers the inputs in the part program under the function Create part program.
- The entries can be edited subsequently.
- The programming system does not simulate the written machining steps.
- The programming system does not check and evaluate the inputs (no syntax check).
**Procedure**

1. Go into the programming level **Machining**.
2. Activate the function **Insert block (DIN)**.
3. Enter parameters in the interactive screenform **Insert blocks**.
4. Mark the input fields with the selection screenform on the right ("+", 
"–", “1...4”).

**Note**

Unassigned DIN code is not marked as such.

5. Confirm the inputs.
4.7.15 Machining Simulation

The programmed tool travel motion can be demonstrated in the graphics field on the screen with the function **Machining simulation**.

In the case of **Machining simulation**, the Graphic Programming System distinguishes between:

- Drilling and milling without multi-side machining and drilling and milling with multi-side machining but without defined surfaces and
- Drilling and milling with multi-side machining with defined surfaces.

The movements of the workpiece or tool actually existing on the machine are simulated.

**Programming notes**

Simulation is portrayed on the screen using a simplified image of the tool and a solid circular disc with the tool diameter.

**Drilling and milling without multi-side machining** and drilling and milling with multi-side machining but without defined surfaces

**Purpose**

Before you can display simulation of the programmed machining in the graphics field, you can set this to suit your own requirements.

**Bild 4.85 Simulation mode**

Having pressed the **Machining simulation** softkey, you will see on the screen the interactive screenform **Simulation mode** as shown in the figure above.
Select the representation in the selection fields.
The selection fields are:

**Display mode**
- Continuous
- Blockwise
- Delete with screen
- Delete without screen

**Simulation with**
- Tool
- Crosshair
- Simple tool

**Display**
- Do not leave
- Leave at end points
- Always leave
- No traversing paths

Bild 4.86 Example of simulation mode

The following figure shows an example for a machining simulation with the above settings for the simulation mode.

Bild 4.87 Machining simulation
You can follow the movement of the tool on screen by the special-effects graphics.

The following **Simulation mode** interactive screenform determines the course of the simulation across the defined surfaces in a Graphic Programming System with multi-side machining:

![Simulation mode](image)

**Display mode**
- Continuous
- Blockwise
- Delete with screen
- Delete without screen

True position representation
- No
- Yes

**Simulation with**
- Tool
- Crosshair
- Simple tool

**Display**
- Do not leave
- Leave at end points
- Always leave
- No traversing paths

**Select surface**
Select the surface on which the simulation is to be performed from the **Select surface** selection field.

You can control the simulation process using the functions in the horizontal softkey menu.
Procedure

1. You have created a workpiece geometry and programmed the various machining steps.
2. Go into the programming level **Machining**.
3. Activate the function **Machining simulation**.
4. Set the modes you require in the interactive screenform **Simulation mode**.
5. Control the simulation process using the functions in the horizontal softkey menu.
4.8 Create part program

Purpose
You can create an executable part program in DIN code with the function Create part program.

Procedure
You have technologically machined the workpiece geometry.
Press the softkey Create part prog. to create an executable DIN program.

You can alter the parameters in the screenform Main program end now displayed on the screen and then confirm your entries with the INPUT key.

The programming system creates the part program for your example.

When you press the softkey Create part program, the programming system calculates the part program from the information that you have entered for geometry and machining.
4.9  Program steps (work schedule)

**Purpose**

During graphic programming of the machining process, the programming system generates a work schedule containing the program steps (operations) that you have programmed.

In the work schedule, you

- are provided with an overview of all graphically generated program steps involved in machining,
- add, move, delete or edit operations for machining (not for the geometry).

**Programming notes**

You can modify the work schedule, for example, by **Editing** the program steps.

**Editable program steps**

Editable program steps are:

- Load magazine,
- Load master data,
- Beginning of program,
- Tool change,
- Milling/drilling operations,
- End of program,
- Machine/help functions,
- Insert block.

**Blank and finished part geometry**

Geometries can only be altered under **Geometry/Edit**.

**Load magazine**

You load a tool magazine when you press softkey **Machining** for the first time.

You can replace this magazine with a different user magazine master data WKZ with the function **Edit program step** (for procedure see Section 4.9.2, Page 4–118).

**Beginning of program**

You create the “Beginning of program” when pressing the softkey **Machining** for the first time.

**End of program**

You generate the “End of program” on pressing the softkey **Create part prog**. This program step must always be the last in the work schedule.
**Procedure**

After calling up the vertical softkey function **Program steps**, the programming system builds up a selection screenform. This displays the list of program steps.

![Diagram of program steps](image)

*Bild 4.89  Program steps: program step names*

Only when you have selected one of the vertical softkey functions **Edit**, **Insert**, **Delete**, **Move** or **Skip** can you select one or several program steps.

Select one of the program step functions **Edit**, **Insert**, **Delete**, **Move** or **Skip** from the vertical softkey menu.

Then place the cursor on the required program step.

Execute the function by pressing the **INPUT** key

or abort using the **RECALL** key.

Press the **CLEAR** key to cancel the last operator action in the editing mode.
4.9.1 Combining program steps to blocks

Purpose
By following the procedure below you can combine several successive program steps into one block for the functions Delete, Move or Skip.

Procedure
You have activated the function Program steps, the program steps are listed on the screen (see Fig. 4.89).

1. Select one of the program step functions Delete, Move or Skip from the vertical softkey menu bar.

2. Then position the cursor onto the first program step you wish to include in a block.

3. Mark the first program step with this key. The marked program step is highlighted with a color.

4. Then place the cursor on the last program step to be included in a program block.

5. Now mark the block with all the program steps it is to include.

6. Fig. 4.90 is an example of the function Move.

7. Conclude the block selection function with the INPUT key.

8. Now execute the function Delete, Move or Skip using the INPUT key for the selected block

or abort the function with the RECALL key.
4.9.2  Edit program step

Purpose
Using the Edit function you can edit
- the parameters for a program step, or
- the program step name.

The procedure for editing program steps is the same as for generating them originally.

Programming notes
If you replace a tool under Edit program steps, you must ensure that the units for the new tool correspond to the units for the feedrate of the old tool (e.g.: mm/min.).

You select the units in the “Cutting values” screenform (see Fig. 4.12, Page 4–33).

Procedure (example)
You are in the function Program steps and the program step overview is displayed.

1. Press the softkey Edit.

2. Position the cursor, for example, on Tool change (or Load magazine etc.)

3. and activate the function Edit.
   You can now correct the program steps individually.

4. Close the interactive screenform (in the example, Tool change point) with the INPUT key.
   This opens the selection screenform with the tools (or e.g.: Tool magazines (WKZ = Master data catalog)) of the current tool magazine.

5. You can select a different tool (or e.g. tool magazine) with the cursor keys.
6. and then replace with the **INPUT** key.

At the same time you leave the function **Edit**, or abort the function with the **RECALL** key.

Once you have exited the function **Edit**, the function is completed and the vertical softkey menu is again in the basic level for the function **Program steps**.

### 4.9.3 Edit program step name

**Purpose**
You can edit the text of a program step in an operation. The program steps concerned here are those for tool change and machining.

**Programming notes**
You cannot edit the text for Beginning of program, Load magazine, Load masterdata and End of program.

**Procedure (example)**

1. Press the softkey **Edit**,

2. put the cursor on **Tool**,

3. and with this key open the interactive screenform for the comments text, the program step name.

4. Edit the text through the alphanumeric keyboard.

5. Terminate editing of the program step name by pressing the **INPUT** key.
4.9.4 Insert program steps

The function **Insert** adds in the work schedule:

- individual program steps, or
- program step sequences

**Purpose**

You insert the new program steps **in front of** the selected program step.

Only program steps for the technology can be inserted.

**Programming notes**

1. Press the softkey **Insert**.

2. An instruction in the command line tells you to place the cursor on the input position.

   Place the cursor on the program step in front of which you wish to insert.

3. On pressing the **INPUT** key you activate the function **Insert**.

As long as you are in the program step function **Insert**, the line above the graphics area displays **Insert** (see Fig. 4.92).
**Bild 4.92 Insert program steps**

**Terminating the Insert function**

If you want to end the function **Insert**, press the softkey **Program steps** and then the softkey **Insert off**.

The programming system performs the computations for the added program steps for the machining operations.
4.9.5 Delete program steps

Purpose
You delete the complete program step with the softkey Delete.
You can group several program steps together in a block.

Programming notes
If a used tool is no longer in the tool magazine, you must then
1. delete the corresponding tool change from the work schedule and
2. then reinsert it (program).

Procedure (example)

1. Press the softkey Delete.

The operation marked in colour can be deleted.

2. Position the cursor on the program step to be deleted.

3. You delete the entire operation by pressing the INPUT key.
The programming system then recomputes all operations.

If you accidentally delete the wrong program step you can reverse the
process with the CLEAR key.
4.9.6 Move program steps

The function **Move** transfers the program steps of a selected operation to another point in the work schedule.

**Programming notes**

You always move **in front of** the selected program step.

You can combine several program steps to one block.

**Procedure (example)**

1. Press the softkey **Move**.

2. Position the cursor to the program step (operation) that you wish to move

3. and activate the function.

4. Now select the program step in front of which you wish to insert

5. and conclude the function.

The program step overview appears with the moved operation.

Press the function **Recalculate** and the programming system will recalculate the machining operations.
4.9.7 Program step incorrect

**Purpose**

In the work schedule, "non-current" machining operations (i.e. all machining operations after an error in calculation, e.g. after a non-existing tool), are marked in a different color.

**Procedure**

These machining operations cannot be selected in the work schedule. They cannot be
- edited
- moved.

Only the first incorrect machining operation can be selected. If you wish to delete or skip all incorrect machining operations, this must be done step by step.

As from software version 6.4, all incorrect machining operations can be selected for deletion and skipping without a safety inquiry.
4.9.8 Recalculate program steps

Recalculate

Purpose
If you have changed the work schedule and wish to remain in the function Program steps and to see the result of the change, then press the softkey Recalculate.

Programming notes
On exiting the function Program steps, the programming system also recalculates the modified program steps. The function Recalculate cannot be aborted.

Procedure
Recalculate
After activating the softkey function Recalculate, the programming system calculates the defined program steps and then reconstructs the updated geometry with its tool travel paths.
4.9.9 Skip program steps

Purpose

Program steps that you do not require at the moment for a specific machining sequence can be skipped without being deleted by activating the function Skip. You can deactivate this function at any time.

Programming notes

In the selection screenform of the program steps, skipped operations are marked with a "*".

You can combine several program steps to one block.

Procedure (example)

1. You are at the program step overview level and press the softkey Skip.

2. You move the cursor to the program step that you wish to skip.

3. Activate the function Program step/Skip by pressing the INPUT key.

The contour or machining sequence is no longer displayed and this indicates that the program step is no longer a part of the work schedule. However, the program step is still contained in the work schedule and can be re-activated by pressing the softkey Skip once again.
4.10  **Vertical softkey functions**

If, when programming your workpiece, you wish to save the data, wish to change the views, or require information on the geometry, then you make use of the functions in the **Vertical** softkey menu. These functions in the vertical softkey menu are available to you at all levels of programming.

The functions:
- **Save**
- **Read**
- **Views**
- **Zoom**
- **Representation**
- **Info**

and the menu tree are described in the Appendix.

---

**Notes on automatic file save function**

The graphic programming system saves the contour and machining programmed by you at regular intervals (approx. 5 minutes).

If you have not saved the last alterations after a system failure (cold restart or power failure), you can call up the last version automatically saved with the **Read/Read auto save** function.

- **Auto save does not take place when you**
  - create oriented geometry
  - create construction geometry

The function **Save** deletes the last file automatically saved. All changes and additions can then be read in with the function **Read/Read original**.
4.11 Pocket calculator function

**Purpose**

The Graphic Programming System has a pocket calculator function. You can call up the pocket calculator without leaving the selected interactive screenform.

**Programming hints**

If the cursor is in a numeric input field in the interactive screenform on the screen, you can call up a pocket calculator function by pressing this key.

If the cursor is positioned on text selection fields, this key steps through the individual options.

The pocket calculator functions in accordance with the basic rules of arithmetic which are described below.

**Procedure**

After calling up the pocket calculator, an input field appears with the cursor at the beginning of it. You can perform calculations in this field.

Entries are possible for calculations that exceed the length of the input field.

When the beginning or the end of the input field is reached, the text is rolled beyond the input field (window function). The current inputs are displayed and the numbers and letters entered initially disappear from the display although they are included in the calculation.

No more than 80 characters may be entered.

The following keys are available for editing the input field:

You move the cursor by one position to the right but not beyond the first free position behind the entry.
You move the cursor by one position to the left but no further than the beginning of the input field.

You place the cursor at the beginning of the line.

You put the cursor on the first free position behind the entered text.

You put the cursor on the first position in the interactive screenform.

You put the cursor on the last character in the interactive screenform. If the entry is shorter than the input field, the cursor is placed at the first free position behind the text.

This key deletes the whole entry (without confirmation!). The text from the previous calculator call is displayed.

With this key you terminate input and start calculation of the expression. The result is displayed.

If you wish to exit the pocket calculator and transfer the result to the input field, then press this key once again.

Abort the pocket calculator function and return to the program, or to the original contents of the input field from which you called the pocket calculator.

*Input error*  
In the case of an input error, the pocket calculator is not exited. Instead, the cursor moves automatically to the location of the input error.

You can correct the input

or abort the pocket calculator function.
4.11.1 Basic arithmetic operations

The symbols for the four basic arithmetic operations are:

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Operation</th>
</tr>
</thead>
<tbody>
<tr>
<td>+</td>
<td>Addition</td>
</tr>
<tr>
<td>−</td>
<td>Subtraction</td>
</tr>
<tr>
<td>*</td>
<td>Multiplication</td>
</tr>
<tr>
<td>/</td>
<td>Division</td>
</tr>
</tbody>
</table>

Addition and subtraction have lower priority than division and multiplication.

Example Input: 5+5*2
Result: 15; because 5*2 is calculated first. Then the first 5 is added.

4.11.2 Operators

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Operation</th>
</tr>
</thead>
<tbody>
<tr>
<td>^</td>
<td>Exponentiation</td>
</tr>
<tr>
<td>%</td>
<td>Percentage increase/decrease</td>
</tr>
</tbody>
</table>

Exponentiation has a higher priority than division or multiplication.

Example Input: 2*2^2
Result: 8; because the multiplication is performed last.

The percentage increase or decrease has the highest priority of all operators which means that this operation is always the first to be performed in an expression.

Example Input: 2*200%10
Result: 440, increase of 10 percent.

Note A negative number must be used for a decrease, i.e. a 20-percent decrease would be entered as follows: 200%–20.
4.11.3 Signs

"+" or "−" are permissible as signs before any number or variable and with all operations. Signs are not permissible before functions and brackets.

4.11.4 Brackets

Execution of an expression can be controlled by using brackets. Parentheses (i.e. round brackets) are used as bracket symbols at all levels.

A maximum of 10 nesting levels is available. This maximum is limited if a great number of operations with different priority follow each other.

*Example*

Input: 2.5*(3+100)

Result: 257.5

The addition is performed first in this expression and it is followed by the multiplication.

4.11.5 Functions

The range of functions includes the trigonometric functions and their inverse functions as well as logarithmic functions and conversion functions of technical importance.

The argument must be specified in parentheses for all functions. The argument can also be a mathematical expression consisting of operations and further functions.

*Example*

- Input: SIN(90)
  Result: 1
  The sine of 90 degrees is calculated.

- Input: COS(SQR(4*4))
  Result: 0.9976
  The product 4*4 is calculated, its square root is determined and finally the cosine of the result is formed.

*Note*

In general, the angular unit for the trigonometric functions is degrees. Appropriate functions are available for converting from and into radians.
List of available functions:

<table>
<thead>
<tr>
<th>Name</th>
<th>Function</th>
</tr>
</thead>
<tbody>
<tr>
<td>ABS</td>
<td>Absolute value</td>
</tr>
<tr>
<td>ACS</td>
<td>Arc cosine</td>
</tr>
<tr>
<td>ASN</td>
<td>Arc sine</td>
</tr>
<tr>
<td>ATN</td>
<td>Arc tangent</td>
</tr>
<tr>
<td>CME</td>
<td>Conversion from inches into cm</td>
</tr>
<tr>
<td>COS</td>
<td>Cosine</td>
</tr>
<tr>
<td>DEC</td>
<td>Conversion from sexagesimal angular measurement into decimal angular measurement. The argument has the same format as the result of DMS.</td>
</tr>
<tr>
<td>DEG</td>
<td>Conversion from radians into degrees.</td>
</tr>
<tr>
<td>DMS</td>
<td>Conversion from decimal angular measurement to sexagesimal angular measure. The result has the following format: AA.MMSS AA Angle MM Minutes SS Seconds</td>
</tr>
<tr>
<td>EXP</td>
<td>Inverse function for natural logarithm</td>
</tr>
<tr>
<td>FAC</td>
<td>Factorial</td>
</tr>
<tr>
<td>FMT</td>
<td>Expanded format specification: A value is only used if it is not “NL”. Example: FMT(“+07.30&gt;”) + With “+” sign (“-” is always output) 0 With leading zeros 7 Total number of places in front of the decimal point (including sign) , Separator between integer and fractional part (e.g. “,” “;” “;”) 3 Total number of places behind the decimal point 0 With trailing zeros &gt; Output the number in right-justified (left-justified) format</td>
</tr>
<tr>
<td>FOR</td>
<td>Format specification for representation of result and conversion of numeric variables into string variables. Example: FOR(10.4) sets the standard format with an overall length of 10 places and 4 places behind the point. By specifying a negative format, leading zeros are output. The greatest settable format is 20.9.</td>
</tr>
<tr>
<td>INC</td>
<td>Conversion from cm into inches.</td>
</tr>
<tr>
<td>INT</td>
<td>Integer component</td>
</tr>
<tr>
<td>LNA</td>
<td>Natural logarithm to the base of e</td>
</tr>
<tr>
<td>LOG</td>
<td>Common logarithm to the base of 10</td>
</tr>
<tr>
<td>Name</td>
<td>Function</td>
</tr>
<tr>
<td>------</td>
<td>----------</td>
</tr>
<tr>
<td>MAC</td>
<td>Contents of the specified variables are used as formula in the expression.</td>
</tr>
<tr>
<td>RAD</td>
<td>Conversion from degrees into radians</td>
</tr>
<tr>
<td>SGN</td>
<td>Sign &lt;br&gt;SGN = –1 if result &lt; 0 &lt;br&gt;SGN = 1 if result &gt; 0 &lt;br&gt;SGN = 0 if result = 0 &lt;br&gt;e.g.: Input: SGN (12–20) &lt;br&gt;Result: –1</td>
</tr>
<tr>
<td>SIN</td>
<td>Sine</td>
</tr>
<tr>
<td>SQR</td>
<td>Calculates square root</td>
</tr>
<tr>
<td>TAN</td>
<td>Tangent</td>
</tr>
<tr>
<td>TEN</td>
<td>Inverse function for common logarithm</td>
</tr>
<tr>
<td>TRU</td>
<td>Removes all trailing blanks in the specified numerical places.</td>
</tr>
</tbody>
</table>
4.12 Function Exit Program.WOP

Please read the information in this section before exiting the Graphic Programming System.

**Purpose**
This section tells you how to save the files that you have created with the Graphic Programming System.

The individual operations necessary to exit graphic programming are listed in the “Programming Examples” section of the corresponding Environment Description (e.g.: “Graphic Programming System on the PC”).

**Programming notes**

- **Workpiece geometry and machining**
  - If you change to a new workpiece, the programming system asks you whether you wish to save the workpiece geometry and machining that you have created.
  - You can also save your workpiece geometry and machining instructions during graphic programming by operating the vertical softkey **Save**.
  - The programming system stores the workpiece geometry file in a workpiece directory **not** visible to you.

- **Note**
  - You always work under a workpiece name (e.g. “PART 1”) in the Graphic Programming System.

- **Part program**
  - When you leave the function **Create part program**, the Graphic Programming System automatically stores the part program in your workpiece directory (e.g.: “PART 1”). You do not have to save it yourself.
5 Appendix

Section 5 describes
- the menu trees of the graphic programming functions,
- the individual horizontal and vertical softkey functions,
- the tool types and their parameters
- and contains a workpiece sketch and work schedule for the exercise.

**Horizontal basic level**

Function descriptions of the softkeys are provided in the tables below.
The numbers above each softkey in the diagram correspond to the numbers of the function descriptions in the tables.

*Bild 5.1 Horizontal softkey functions in the basic level*
<table>
<thead>
<tr>
<th>Menu tree number</th>
<th>Functions</th>
<th>Function description</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Tools Material</td>
<td>For generating and managing tools, tool master data, tool user magazines and the materials list with the functions</td>
</tr>
<tr>
<td></td>
<td></td>
<td>- Masterdata magazine</td>
</tr>
<tr>
<td></td>
<td></td>
<td>- Material data.</td>
</tr>
<tr>
<td>2</td>
<td>Geometry</td>
<td>You create/edit the workpiece contours with standard geometry elements or user-defined geometry elements or you move the construction zero point with the geometry functions</td>
</tr>
<tr>
<td></td>
<td></td>
<td>- Create new</td>
</tr>
<tr>
<td></td>
<td></td>
<td>- Edit</td>
</tr>
<tr>
<td></td>
<td></td>
<td>- Delete</td>
</tr>
<tr>
<td></td>
<td></td>
<td>- Copy</td>
</tr>
<tr>
<td></td>
<td></td>
<td>- Create surface (define) (^1),</td>
</tr>
<tr>
<td></td>
<td></td>
<td>- Select surface (^1) and</td>
</tr>
<tr>
<td></td>
<td></td>
<td>- Edit surface (^1).</td>
</tr>
<tr>
<td>3</td>
<td>Machining</td>
<td>You define the drilling/milling technology for the workpiece, simulate machining and generate a part program with the functions:</td>
</tr>
<tr>
<td></td>
<td></td>
<td>- Milling</td>
</tr>
<tr>
<td></td>
<td></td>
<td>- Drilling</td>
</tr>
<tr>
<td></td>
<td></td>
<td>- Machine/help functions</td>
</tr>
<tr>
<td></td>
<td></td>
<td>- Insert block (DIN),</td>
</tr>
<tr>
<td>4</td>
<td>Select surface</td>
<td>You select the plane for the subsequent machining and enter the position of the tool for changing plane. You also define the rotary axis.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>The machine manufacturer defines whether the rotary axis refers to the tool or workpiece.</td>
</tr>
</tbody>
</table>

\(^1\) Only if multi-side machining option is available
<table>
<thead>
<tr>
<th>Menu tree number</th>
<th>Functions</th>
<th>Function description</th>
</tr>
</thead>
</table>
| 5                | Machining simulation   | The drilling/milling machining process that you have programmed and the traversing paths of the tools used are simulated on the screen. While this function is active, you can use the following functions to modify the display:  
  - Jump to the next machining operation  
  - Jump to the next NC block  
  - Change over block-by-block/continuous  
  - Speed + (faster)  
  - Speed – (slower) |
| 6                | Create part prog.      | You create the part program in accordance with the machining steps.                    |
| 7                | End                    | You quit WOP programming without saving the programmed geometry or machining. In order to avoid loss of graphically programmed data note the following:  
  **Note:** Save the programmed workpiece before you quit the graphic programming system with **End**. |
5.1 Tools Material

Bild 5.2 Tools Material functions
<table>
<thead>
<tr>
<th>Menu tree number</th>
<th>Functions</th>
<th>Function description</th>
</tr>
</thead>
</table>
| 1                | Tools Material          | For generating and managing tools, tool master data, tool user magazines and the materials list with the functions  
|                  |                         | - Masterdata magazine  
|                  |                         | - Material data.      |
| 1.1              | Masterdata magazine     | You can change the parameters for the tool master data or the tools stored in a specific magazine or add new parameters.  
|                  |                         | The functions are:  
|                  |                         | - Load magazine  
|                  |                         | - New magazine  
|                  |                         | - Save magazine  
|                  |                         | - Delete magazine  
|                  |                         | - Load masterdata  
|                  |                         | - Save masterdata  
|                  |                         | - Create tool      |
| 1.1.1            | Load magazine           | The available tool magazines are displayed and you can select the one you require. |
| 1.1.2            | New magazine            | You generate a new magazine using the tools from the tool master data.  
|                  |                         | You copy the selected tool from the tool master data into the magazine currently loaded. Make sure that the tool ID number is unambiguous when copying within the master data. |
| 1.1.3            | Save magazine           | You save the modified or added tools in the magazine currently loaded or in a new magazine. |
| 1.1.4            | Load master data        | You load the tool master data and are then able to edit them. |
| 1.1.5            | Save master data        | You save the modified or added tools in the master data. |
| 1.1.6            | Tool                    | You generate or change tools (preferably in the master data) using the following functions  
|                  |                         | - New tool  
|                  |                         | - Copy tool  
|                  |                         | - Change tool and  
<p>|                  |                         | - Delete tool. |</p>
<table>
<thead>
<tr>
<th>Menu tree number</th>
<th>Functions</th>
<th>Function description</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.1.6.1</td>
<td>New tool</td>
<td>You generate all geometrical and technological parameters of a tool you have selected from a type list. The system displays the geometry of the tool. Tool input is in five steps: 1. Select type of machining (milling or drilling). 2. Select tool type (e.g.: end mill). 3. Enter the general tool characteristic data. Note: The tool ID number must be unambiguous. Each tool has its own number. 4. Enter the tool geometry data. After input, the geometry is displayed graphically by the system. 5. Enter the cutting value file assigned to the material group.</td>
</tr>
<tr>
<td>1.1.6.2</td>
<td>Copy tool</td>
<td>There are two types of copying. • You copy within the tool master data after having loaded the master data. • You copy a tool from the master data into the tool user magazine that you have loaded. When copying within the master data, always make sure that the tool ID number is unambiguous.</td>
</tr>
<tr>
<td>1.1.6.3</td>
<td>Change tool</td>
<td>You change the tool parameters in the currently loaded tool magazine using the functions: • All • T &amp; D number • Geometry data • Technology</td>
</tr>
<tr>
<td>1.1.6.3.1</td>
<td>All</td>
<td>Changing the tool you have selected requires 3 steps: 1. Enter general tool parameters. The tool ID number must be unique. 2. Enter the tool geometry data. 3. Enter the material-specific cutting data. Note: You can combine several tools to a block with the toggle key.</td>
</tr>
<tr>
<td>Menu tree number</td>
<td>Functions</td>
<td>Function description</td>
</tr>
<tr>
<td>------------------</td>
<td>----------------------</td>
<td>-------------------------------------------------------------------------------------</td>
</tr>
<tr>
<td>1.1.6.3.2</td>
<td>T &amp; D number</td>
<td>You change the name, the T or D number, the id. text and the feed parameters.</td>
</tr>
<tr>
<td>1.1.6.3.3</td>
<td>Geometry data</td>
<td>You change the geometry data and the insert position.</td>
</tr>
<tr>
<td>1.1.6.3.4</td>
<td>Technology</td>
<td>You change the tool cutting values.</td>
</tr>
<tr>
<td>1.1.6.M.1</td>
<td>Accept</td>
<td>The displayed tool geometry is confirmed and you are interactively prompted for the parameters of the cutting values for the individual materials.</td>
</tr>
<tr>
<td>1.1.6.M.2</td>
<td>Edit</td>
<td>This function enables you to change the parameters of the preceding interactive forms.</td>
</tr>
<tr>
<td>1.1.6.M.3</td>
<td>Abort</td>
<td>The generated geometry is ignored.</td>
</tr>
<tr>
<td>1.1.6.4</td>
<td>Delete tool</td>
<td>You delete the selected tool in the tool magazine currently loaded.</td>
</tr>
<tr>
<td>1.4</td>
<td>Material data</td>
<td>You generate a material list and modify and save it with the following functions</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• Modify material list</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• Save material list</td>
</tr>
<tr>
<td>1.4.1</td>
<td>Change mat. list</td>
<td>You can generate a material list for the workpiece.</td>
</tr>
<tr>
<td>1.4.2</td>
<td>Save mat. list</td>
<td>The changes and additions are stored in the material list.</td>
</tr>
</tbody>
</table>
5.2 Geometry

1. Use selection control to graphically select contours.
2. Enter parameters in an interactive form.

Bild 5.3 Geometry functions
2.1 Create new

2.1.1 Blank 2.1.2 Finished part 2.1.3 Groove 2.1.4 Holes 2.1.5

Interactive screenform

2.1.1.1 Oriented geometry 2.1.1.2 Construct. geometry 2.1.1.3 Standard geometry 2.1.1.4 Abort 2.1.1.5 OK

2.1.1.1a 2.1.1.2a

Oriented geometry Edit elements

2.1.1.2.1a 2.1.1.2.2a 2.1.1.2.3a

Change element Erase element Select variants

2.1.1.1.1 2.1.1.1.2 2.1.1.1.3 2.1.1.1.4 2.1.1.1.5 2.1.1.1.6 2.1.1.1.7

Line right Line left Line up Line down Line any Arc ccw Arc cw

The following applies for the geometry element to be constructed (line/arc):

1. Parameter inputs in an interactive screenform.
2. a: Displays of the solution (where there are two possible solutions, selection via softkey menu).
   b: The construction (display of the solution) of the element is solved by the programming system with the parameters entered for the next element or the one after that.

Bild 5.4 Geometry/Create_new functions
Create contours graphically with the selection control.

Chamfer Rounding Delete transition

Edit or create transitions graphically with the selection control.

Graphic construction of a user-defined geometry.

Edit or create the contours graphically with the selection control.

Bild 5.5  Geometry/Create_new/Construction geometry functions
2.1.1.5 Standard geometry (blank/contour)

Interactive screenform

2.1.1.5.1 Rectangle
2.1.1.5.2 Circle
2.1.1.5.3 n-corner

Interactive screenform

2.1.3.5 Standard geometry (groove)

Interactive screenform

2.1.3.5.1 Elong. hole
2.1.3.5.2 Rectang. groove
2.1.3.5.3 Annular groove
2.1.3.5.4 n-corner groove
2.1.3.5.5 Radial groove
2.1.3.5.6 Circular groove

Interactive screenform

Bild 5.6  Geometry/Create_new/Standard geometry functions
Edit or create the holes graphically with the selection control.

Interactive screenform

Construct.

elements

Define

pattern

Abort

OK

Point

Line

Circle

Row of points

Circle of points

Point box

Edit const. geo

Graphic construction of a user-defined geometry.

Interactive screenform

Edit parameter

New const. element

Del const. element

Edit or create the contours graphically with the selection control.

Bild 5.7  Geometry/Create_new/Holes functions
<table>
<thead>
<tr>
<th>Menu tree number</th>
<th>Functions</th>
<th>Function description</th>
</tr>
</thead>
</table>
| 2                | Geometry  | You create/edit the workpiece contours with standard geometry elements or user-defined geometry elements or you move the construction zero point with the geometry functions:  
|                  |           | - Create new  
|                  |           | - Edit  
|                  |           | - Delete  
|                  |           | - Copy  
|                  |           | - Create surface (define)\(^1\),  
|                  |           | - Select surface \(^1\) and  
|                  |           | - Edit surface \(^1\). |
| 2.1              | Create new | You create the following types of geometry elements:  
|                  |           | - Blank  
|                  |           | - Contour  
|                  |           | - Groove  
|                  |           | - Holes  
|                  |           | - Cycle ref. point |
| 2.1.1            | Blank     | You create the blank contour of the workpiece with specified material, workpiece reference plane and material thickness using the functions:  
|                  |           | - Oriented geometry  
|                  |           | - Construction geometry  
<p>|                  |           | - Standard geometry |
| 2.1.1.1          | Oriented geometry | Join an oriented element (straight line, arc) to an oriented contour definition. |
| 2.1.1.1.1        | Line right | Generate an oriented line with angle 0°. |
| 2.1.1.1.2        | Line left | Generate an oriented line with angle 180°. |
| 2.1.1.1.3        | Line up | Generate an oriented line with angle 90°. |
| 2.1.1.1.4        | Line down | Generate an oriented line with angle 270°. |
| 2.1.1.1.5        | Line any | Generate an oriented line with a user-defined start angle. |
| 2.1.1.1.6        | Arc left ccw | Generate an oriented arc counterclockwise (ccw). |
| 2.1.1.1.7        | Arc right cw | Generate an oriented arc clockwise (cw). |</p>
<table>
<thead>
<tr>
<th>Menu tree number</th>
<th>Functions</th>
<th>Function description</th>
</tr>
</thead>
<tbody>
<tr>
<td>2.1.1.1.1a</td>
<td>Geometry elements</td>
<td>For description see P. 4–45.</td>
</tr>
<tr>
<td>2.1.1.1.2a</td>
<td>Edit elements</td>
<td>Edit the parameters of the selected element.</td>
</tr>
<tr>
<td>2.1.1.1.2.1a</td>
<td>Change element</td>
<td>Change the numeric parameters of the selected oriented contour element.</td>
</tr>
<tr>
<td>2.1.1.1.2.2a</td>
<td>Erase element</td>
<td>Erase the selected oriented contour element.</td>
</tr>
<tr>
<td>2.1.1.1.2.3a</td>
<td>Select variant</td>
<td>Select another construction variant if there are several possible solutions.</td>
</tr>
<tr>
<td>2.1.1.4</td>
<td>Construction geometry</td>
<td>You construct a contour of your own choice by constructing elements (point, line, circle) and create a contour on this. You make use of the functions:</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• Const. geo elements</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• Contour definition</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• Element transition</td>
</tr>
<tr>
<td>2.1.1.4.1</td>
<td>Const. geo elements</td>
<td>You always require reference elements for designing the construction geometry. The first reference elements are the X or Y axis as reference lines and the zero point as construction reference point. Using the cursor keys, you guide the crosshair and with the INPUT key you confirm the selection of the reference elements. <strong>You do not have to position the crosshair exactly.</strong> The system finds the reference element that is closest to the crosshair. The system supplies you with interactive screens into which you enter the parameters for the elements. When the construction has been completed, the system computes several solutions. Select the solution you require with the crosshair and accept this by pressing the INPUT key. You can make use of circles, lines and points as construction elements. You generate these elements with the functions:</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• Point</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• Line</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• Circle</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• Row of points</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• Circle of points</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• Point box</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• Edit construction geometry.</td>
</tr>
<tr>
<td>Menu tree number</td>
<td>Functions</td>
<td>Function description</td>
</tr>
<tr>
<td>------------------</td>
<td>-----------</td>
<td>----------------------</td>
</tr>
<tr>
<td>2.1.1.4.1.1</td>
<td>Point</td>
<td>You design a construction geometry point.</td>
</tr>
<tr>
<td>2.1.1.4.1.2</td>
<td>Line</td>
<td>You design a construction line.</td>
</tr>
<tr>
<td>2.1.1.4.1.3</td>
<td>Circle</td>
<td>You design a construction circle.</td>
</tr>
<tr>
<td>2.1.1.4.1.4</td>
<td>Row of points</td>
<td>You design construction points on a line.</td>
</tr>
<tr>
<td>2.1.1.4.1.5</td>
<td>Circle of points</td>
<td>You design construction points on a circle or circular arc.</td>
</tr>
<tr>
<td>2.1.1.4.1.6</td>
<td>Point box</td>
<td>You design construction points on a filled or empty parallelogram.</td>
</tr>
</tbody>
</table>
| 2.1.1.4.1.7      | Edit const. geo | You edit construction geometry elements with the functions:  
|                  |           | - Change parameter  
|                  |           | - New construction element  
|                  |           | - Delete construction element |
| 2.1.1.4.1.7.1    | Change parameter | You select an element (crosshair, keyboard) and change the parameters in the interactive screenform. |
| 2.1.1.4.1.7.2    | New const. element | You select an element (crosshair, keyboard) and redefine it. |
| 2.1.1.4.1.7.3    | Delete const. element | You select an element (crosshair, keyboard) and delete it. |
| 2.1.1.4.2        | Contour definition | You generate a user-defined workpiece contour by selecting the construction elements (crosshair, keyboard). The contour begins with an initial point and ends with an end point. The initial point can also be identical with the end point. |
| 2.1.1.4.4        | Transition elements | You modify the corners of the workpiece contour using the functions:  
|                  |           | - Chamfer  
|                  |           | - Radius  
<p>|                  |           | - Delete |
| 2.1.1.4.4.1      | Chamfer   | You select a corner of the geometry and enter the value for the chamfer. |
| 2.1.1.4.4.1.7    | OK        | You end this function. |
| 2.1.1.4.4.2      | Radius    | You select a corner of the geometry and enter the radius. |
| 2.1.1.4.4.7      | Delete transition | You select a corner of the geometry and delete the transition element. |</p>
<table>
<thead>
<tr>
<th>Menu tree number</th>
<th>Functions</th>
<th>Function description</th>
</tr>
</thead>
<tbody>
<tr>
<td>2.1.1.4.6</td>
<td>Abort</td>
<td>You end the construction, the programmed elements or modifications are lost.</td>
</tr>
<tr>
<td>2.1.1.4.7</td>
<td>OK</td>
<td>The design of the defined contour is complete. The construction geometries are not displayed. You see the contour.</td>
</tr>
<tr>
<td>2.1.1.5</td>
<td>Standard geometry (blank/contour)</td>
<td>You create the workpiece contour with specification of the start and end planes using the functions:</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• Rectangle</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• Circle</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• n-corner</td>
</tr>
<tr>
<td>2.1.1.5.1</td>
<td>Rectangle</td>
<td>You define a rectangular contour.</td>
</tr>
<tr>
<td>2.1.1.5.2</td>
<td>Circle</td>
<td>You define a circular contour.</td>
</tr>
<tr>
<td>2.1.1.5.3</td>
<td>n-corner</td>
<td>You define a contour with n-corners.</td>
</tr>
<tr>
<td>2.1.2</td>
<td>Contour</td>
<td>You create the workpiece contour with specification of the start and end planes using the functions:</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• Oriented geometry</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• Edit elements</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• Construction geometry</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• Standard geometry</td>
</tr>
<tr>
<td>2.1.3</td>
<td>Groove</td>
<td>You create the workpiece groove contour with specification of the start plane, end plane and groove width by means of the functions:</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• Oriented geometry</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• Edit elements</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• Construction geometry</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• Standard geometry</td>
</tr>
<tr>
<td>2.1.3.5</td>
<td>Standard Geometry (Groove)</td>
<td>You create the groove contour of the workpiece with specification of the start plane, end plane and width by using the following functions:</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• Elongated hole</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• Rectangular groove</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• Annular groove</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• n-corner groove</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• Radial grooves</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• Circular groove</td>
</tr>
<tr>
<td>Menu tree number</td>
<td>Functions</td>
<td>Function description</td>
</tr>
<tr>
<td>------------------</td>
<td>-----------------</td>
<td>--------------------------------------------------------------------------------------</td>
</tr>
<tr>
<td>2.1.3.5.1</td>
<td>Elongated hole</td>
<td>You define an elongated hole.</td>
</tr>
<tr>
<td>2.1.3.5.2</td>
<td>Rectangular groove</td>
<td>You define a rectangular groove (slot).</td>
</tr>
<tr>
<td>2.1.3.5.3</td>
<td>Annular groove</td>
<td>You define an annular groove (slot).</td>
</tr>
<tr>
<td>2.1.3.5.4</td>
<td>n-corner groove</td>
<td>You define a groove (slot) with n-corners.</td>
</tr>
<tr>
<td>2.1.3.5.5</td>
<td>Radial grooves</td>
<td>You define radial grooves (slots) with slot length (SL), radius (R1), initial angle (A1), incremental angle (A2) and number of slots (NS).</td>
</tr>
<tr>
<td></td>
<td>Circular groove</td>
<td>You define a circular groove (slot) with the radius (RM), opening angle (A2) and initial angle (A1).</td>
</tr>
<tr>
<td>2.1.4</td>
<td>Holes</td>
<td>You generate holes with specification of diameter and depth by using the following functions:</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• Single hole</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• Row of holes</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• Circle of holes</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• Hole box</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• Any hole pattern</td>
</tr>
<tr>
<td>Menu tree number</td>
<td>Functions</td>
<td>Function description</td>
</tr>
<tr>
<td>-----------------</td>
<td>--------------------</td>
<td>--------------------------------------------------------------------------------------</td>
</tr>
<tr>
<td>2.1.4.1</td>
<td>Single hole</td>
<td>You define the single hole.</td>
</tr>
<tr>
<td>2.1.4.2</td>
<td>Row of holes</td>
<td>You define the row of holes.</td>
</tr>
<tr>
<td>2.1.4.3</td>
<td>Circle of holes</td>
<td>You define the circle of holes.</td>
</tr>
<tr>
<td>2.1.4.4</td>
<td>Hole box</td>
<td>You define the hole box.</td>
</tr>
</tbody>
</table>
| 2.1.4.5         | Any hole pattern   | You program a group of hole points. You design construction elements (point, line) and define on this a group of hole points as a contour. You make use of the functions:  
  * Construction geometry elements  
  * Define pattern  
  * Edit pattern |
| 2.2             | Edit               | You select a contour graphically and edit the parameters directly in an interactive screenform. |
| 2.3             | Delete             | You select a contour graphically and delete it after the request for confirmation.    |
| 2.4             | Copy               | You select contours graphically and generate modified copies. The functions are:  
  * Translate and rotate  
  * Mirror  
  * Reverse mirror  
  * Scaling  
  * Parallel contour |
| 2.4.1           | Translate          | You copy selected contours by translation in the X, Y and Z axes or you rotate the copied contours around an X/Y point by a specified angle around the Z axis. |
|                 | Rotate             |                                                                                      |
| 2.4.2           | Mirror             | You copy selected contours by mirroring about a straight line which you define by means of a reference point and an angle. |
| 2.4.3           | Reverse mirror     | You copy selected contours as described under Mirror. In addition, the direction of machining of the mirrored contour is inverted. This is important for generating closed contours. |
| 2.4.4           | Scaling            | You copy selected contours. The copied contour is magnified or reduced by the factor specified. |
| 2.4.5           | Parallel contour   | You define the parallel contour right or left by directly inputting a distance.  
  **Note:** The parallel contour can be applied to only one contour. |
<table>
<thead>
<tr>
<th>Section</th>
<th>Description</th>
</tr>
</thead>
</table>
| 2.5     | Create surface (define)\(^1\) You create a new surface. These surfaces are a part of  
|         |             | - the blank or finished part geometry or  
|         |             | - the peripheral surface of the geometry as a free definition. |
| 2.5.1   | Surface from geometry\(^1\) You select a geometry element that the system defines as a surface directly from the created blank or finished part. |
| 2.5.2   | Peripheral surface\(^1\) A cylinder peripheral surface can be created. This peripheral surface is a development of the blank or finished part geometry that is a circle. |
| 2.5.3   | Free definition\(^1\) You can define a surface freely in space. |
| 2.6     | Select surface\(^1\) You select the current plane for the following graphic programming. |
| 2.7     | Edit surface\(^1\) The dimensions of created surfaces can be edited or deleted. |
| 2.7.1   | Modify surface\(^1\) You modify the parameters of a created surface. |
| 2.7.2   | Delete surface\(^1\) You delete a created surface. |

1) Only if multi-side machining option is available
5.3 Machining

**Bild 5.8** Machining and Drilling functions
3.1

3.1.1 Tool selection
3.1.2 Contour milling
3.1.3 Solid milling
3.1.4 Manual milling
3.1.5 Surface milling
3.1.6 Surface milling, par. to axis
3.1.7 Manual milling

A tool is selected interactively with the programming system.

1. Select contours graphically with the selection control
2. Parameter input into an interactive screenform

Select contour(s) graphically

Contour of a Standard geometry
Contour of an Oriented geometry or Construction geometry

Graphically define start point

(1.) Graphically select initial element
(2.) Graphically select end element

Interactive screenform

Graphic manual milling and inserting of the functions stored on the softkey bar.

1. Select contours
2. Parameter input in an interactive screenform.

Interactive screenform

3.1.7.1 3.1.7.2 3.1.7.3 2.1.1.4.6 2.1.1.4.6

Feed Rapid traverse Undo Numeric input Abort OK

Bild 5.9 Milling functions
### Menu tree number 3

<table>
<thead>
<tr>
<th>Functions</th>
<th>Function description</th>
</tr>
</thead>
</table>
| Machining | You define the drilling/milling technology for the work-piece, simulate machining and generate a part program with the functions:  
  - Milling  
  - Drilling  
  - Machine/help functions  
  - Insert block (DIN), |

#### 3.1 Milling

You choose a tool for the machining process and select between the various milling functions:

- Tool selection
- Contour milling
- Solid milling
- Solid milling parallel to the axis
- Surface milling
- Surface milling parallel to the axis and
- Manual milling

#### 3.3.1.1 Tool selection

You determine the tool change point, strategy and select the tool.

The following functions

- All
- T number
- D number
- Id. number
- Geometry data

are provided for locating and selecting the tool.

#### 3.3.1.1.1 All

You select a tool from all the tools in the magazine.

#### 3.3.1.1.2 T number

Direct selection of a tool by its T number.

#### 3.3.1.1.3 D number

Direct selection of a tool by its D number.

#### 3.3.1.1.4 Id. number

Direct selection of a tool by its id. number.

#### 3.3.1.1.5 Geometry data

Selection of a tool by its geometry search pattern.
### Menu tree number | Functions | Function description
--- | --- | ---
3.1.2 | Contour milling | You machine selected contours with optional tool radius compensation.  
For machining a contour, you either use the standard start point or define a manual start point.  
Machining takes place  
- along the contour, or  
- with CRC adjacent to the contour.  
For a contour that has been created under **Oriented geometry** or **Construction geometry**, the following parts can be machined:  
- Closed contour  
- Open contour  
- Contour sections  
The approach and retraction strategy can be selected as required.  
Depending on the parameterization, various strategies can be used for depth infeed and machining direction. At points where the tool cannot follow the contour, an alternative contour can be made up where no contour violation can occur with the selected tool (path compensation).

3.1.3 | Solid milling | You machine selected pockets with islands.

3.1.4 | Solid milling parallel to the axis | You machine selected pockets with islands parallel to the axis.

3.1.5 | Surface milling | You machine the surface of the selected island parallel to the contour and mill selected pockets at the same time.  
You machine the surface of a selected contour. You select a suitable approach and retract strategy. Machining always proceeds from the outside to the inside. The paths are generated parallel to the contour. Specified pockets are milled. All selected contours are overtravelled by the degree of overlap.  
**Note:** It is not possible to have an island on the surface to be machined, nor is it possible to machine the bottom of a pocket.

3.1.6 | Surface milling parallel to the axis | You machine the surface of the selected island parallel to the axis and mill selected pockets at the same time.
### Menu tree number | Functions | Function description
--- | --- | ---
3.1.7 | Manual milling | You can move the tool at rapid traverse rate or at feed-rate either with the crosshair or by inputting coordinates. Incorrectly entered movements can be cancelled. The tool path is represented in the graphics area with the tool width. The entire machining definition can be rejected by the abort function. When entering coordinates (tool centre path) the current tool position provides the default values. For these functions, you work with the crosshair and the softkeys:
- Feed
- Rapid traverse
- Undo
- Numeric inputs

3.1.7.1 | Feed | You traverse the paths programmed after this at feed-rate.

3.1.7.2 | Rapid traverse | You position to the points programmed after this at the rapid traverse rate.

3.1.7.3 | Undo | You ignore the machining definition that has just been executed and revert to the previous one.

3.1.7.4 | Numeric input | You enter the next tool position (cutter centre path) directly as coordinates \((X/Y/Z)\).
## Functions

<table>
<thead>
<tr>
<th>Menu tree number</th>
<th>Functions</th>
<th>Function description</th>
</tr>
</thead>
</table>
| 3.2              | Drilling  | You select the drilling technology. The system offers you the following drilling operations:  
|                  |           | • Centre  
|                  |           | • Countersink  
|                  |           | • Deep hole  
|                  |           | • Thread  
|                  |           | • Bore to DIN  
|                  |           | Having selected a tool if necessary, you select the hole and parameterize an interactive screenform with the technological parameters.  
|                  |           | The drilling cycle operates with the value of the **Depth** of the selected drilling geometry element and the empirical values file (unless you have specified another depth).  
| 3.2.1            | Center    | You select a tool for the hole and select graphically the hole to be machined with cycle L81.  
|                  | Drill     |  
| 3.2.1.2          | Value     | You select the hole points and then enter the technological parameters in an interactive screenform.  
|                  | input     |  
| 3.2.2            | Countersink | You select a tool for the hole and select graphically the hole to be machined with cycle L82.  
| 3.2.3            | Deep hole | You select a tool for the hole and select graphically the hole to be machined with cycle L83.  
| 3.2.4            | Thread    | You select a tool for the thread and select graphically the hole to be machined with cycle L84.  
| 3.2.5            | Bore to   | You can choose between several boring cycles (L85...L89).  
<p>|                  | DIN       |<br />
| 3.2.5.1, 3.2.5.2, 3.2.5.3, 3.2.5.4, 3.2.5.5 | Bore 1...5 | You select a tool for the hole and select graphically the hole to be machined with cycle (L85...L89).  |</p>
<table>
<thead>
<tr>
<th>Menu tree number</th>
<th>Functions</th>
<th>Function description</th>
</tr>
</thead>
<tbody>
<tr>
<td>3.4</td>
<td>Machine / aux. fct.</td>
<td>You call up machine–specific auxiliary and switching functions. These functions can support user cycles or be defined as programming system macro. These functions do not have a reference to the geometry elements.</td>
</tr>
<tr>
<td>3.5</td>
<td>Insert block (DIN)</td>
<td>You initiate control-specific functions and enter freely programmable DIN blocks. This freely defined DIN code is not checked by the control.</td>
</tr>
<tr>
<td>4</td>
<td>Select surface</td>
<td>You select the current plane for the subsequent machining operation.</td>
</tr>
</tbody>
</table>
| 5                | Machining simulation | The drilling/milling machining process that you have programmed and the traversing paths of the tools used are simulated on the screen. While this function is active, you can use the following functions to modify the display:  
|                  |                    | - Jump to the next machining operation                                               |
|                  |                    | - Jump to the next NC block                                                          |
|                  |                    | - Change over block-by-block/continuous                                              |
|                  |                    | - Speed + (faster)                                                                  |
|                  |                    | - Speed – (slower)                                                                  |
| 6                | Create part prog.  | You create the part program in accordance with the machining steps.                  |
5.4 Vertical softkey menu

Bild 5.10 Vertical softkey functions

1) Function only active with Construction geometry
2) Function only active with Machining
<table>
<thead>
<tr>
<th>Menu tree number</th>
<th>Functions</th>
<th>Function description</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Save</td>
<td>You save the current geometry file. You entered the name in the programming system before calling the function <em>Program. WOP</em> under <em>Workpiece/New</em>.</td>
</tr>
</tbody>
</table>
| 2                | Read        | You use the following functions for reading or initializing the workpiece file:  
<p>| 2.1              | Read original | You read the geometry file you have saved last. The geometrical data generated in the graphics display are lost. |
| 2.2              | Read autosave | The graphic programming system automatically saves your programmed geometry and machining operation after a certain interval which can be looked at at any time with this softkey. |
| 2.3              | New         | All geometry and machining data are deleted. You completely reprogram the workpiece under the same name. |</p>
<table>
<thead>
<tr>
<th>Menu tree number</th>
<th>Functions</th>
<th>Function description</th>
</tr>
</thead>
<tbody>
<tr>
<td>3</td>
<td>Views</td>
<td>You can use the following views for displaying the workpiece geometry:</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• Plan view</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• Side view</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• Front view</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• DIN isometry</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• Rotate isometry</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• Set 4 views</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• Activate 4 views</td>
</tr>
<tr>
<td>3.1</td>
<td>Plan view</td>
<td>You change your workpiece view to plan view.</td>
</tr>
<tr>
<td>3.2</td>
<td>Side view</td>
<td>You change your workpiece view to side view from the right.</td>
</tr>
<tr>
<td>3.3</td>
<td>Front view</td>
<td>You change your workpiece view to front view.</td>
</tr>
<tr>
<td>3.4</td>
<td>DIN isometry</td>
<td>You change your workpiece view to top view from diagonally above.</td>
</tr>
<tr>
<td>3.5</td>
<td>Rotate isometry</td>
<td>You select the view “from diagonally above”.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>1. With the cursor keys, you rotate a displayed model (at the bottom left in the graphics area)</td>
</tr>
<tr>
<td></td>
<td></td>
<td>2. You confirm the view with the INPUT key.</td>
</tr>
<tr>
<td>3.6</td>
<td>Set 4 views</td>
<td>You change the sizes of the 4 windows relative to each other.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>1. With the cursor keys, you position the crosshair (the crosshair also shows how the screen is divided up)</td>
</tr>
<tr>
<td></td>
<td></td>
<td>2. You confirm the division with the INPUT key.</td>
</tr>
<tr>
<td>3.7</td>
<td>Activate 4 views</td>
<td>The graphics area is divided into four windows. It consists of the plan view, side view, front view and DIN isometry.</td>
</tr>
<tr>
<td>Menu tree number</td>
<td>Functions</td>
<td>Function description</td>
</tr>
<tr>
<td>------------------</td>
<td>---------------</td>
<td>-----------------------------------------------------------------------------------------------------------------------------------------------------</td>
</tr>
<tr>
<td>4</td>
<td>Zoom</td>
<td>Under this function, you apply the following functions to the screen image in your graphics area</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• Refresh</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• Select a viewport</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• Set this viewport</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• Move the viewport</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• Enter a viewport directly</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• Display workpiece in general view (autozoom).</td>
</tr>
<tr>
<td>4.1</td>
<td>Refresh</td>
<td>You refresh the screen image with its geometrical data in the graphics area.</td>
</tr>
<tr>
<td>4.2</td>
<td>Select viewport</td>
<td>You zoom or reduce the workpiece geometry displayed.</td>
</tr>
<tr>
<td>4.3</td>
<td>Set viewport</td>
<td>The viewport created under 4.2 is set for the further representation (even when reconstructing the screen).</td>
</tr>
<tr>
<td>4.4</td>
<td>Move viewport</td>
<td>You redefine the centre of the display.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>1. With the cursor keys, position the crosshair to the point in the graphic field which is to be the centre of screen division.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>2. With the INPUT key, confirm the new division.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Select <strong>Set viewport</strong> if this display centre point is to be used as the initial representation.</td>
</tr>
<tr>
<td>4.5</td>
<td>Enter viewport</td>
<td>You enter the viewport directly. The following are the parameters:</td>
</tr>
<tr>
<td></td>
<td></td>
<td>$X_{\min}, X_{\max}, Y_{\min}, Y_{\max}, Z_{\min}, Z_{\max}$</td>
</tr>
<tr>
<td>4.7</td>
<td>Autozoom</td>
<td>Workpiece is displayed in general view in the graphics field.</td>
</tr>
</tbody>
</table>
## Menu tree number | Functions | Function description
--- | --- | ---
5 | Representation | You can display the workpiece geometry in various ways. You can select
- Dir. arrows on/off
- Dimensioning on/off
- All planes on/off
- Display selection for planes
- Color text
- Color graphics
- Save setup

### 5.1 Dir. arrows on/off
You conditionally display or don’t display the direction arrows for your workpiece contour.

### 5.2 Dimensioning on/off
You display or don’t display the scale of the graphics field.

### 5.3 All planes on/off
All planes defined are displayed in the corresponding view in the graphics field.
A “*” in front of the plane display stands for the representation “All planes on”.

### 5.4 Display plane select.
You obtain information on all planes (surfaces) defined (created). The surfaces are displayed by the system in a selection screenform.

### 5.5 Colour text
Below the graphics area, the corresponding number codings for the individual colours are shown.

### 5.6 Colour graphics
The number codings for the individual colours are shown below the graphics area.

### 5.7 Save setup
You save the new setup.
|   | Program Info | This function provides you with information on the following data or parameters:  
|   |   | • Construct. element  
|   |   | • Angle / spacing  
|   |   | • System status  
| 6.1 | Construct. element | The absolute parameters and the geometry syntax of the geometry element selected are displayed.  
|     |     | This function is active only with the Construct. element function.  
| 6.2 | Angle spacing | You select two construction elements and the system displays the spacing or the angle as absolute parameters. This function is active only with the Construct. element function.  
| 6.3 | Contour element | The Graphic Programming System displays the parameters of the geometry elements, which you have created with the function Oriented geometry.  
|     |     | You select the individual element.  
| 6.4 | Contour dist./angl | The Graphic Programming System displays the distance or angle of two geometry elements belonging together. You have created the elements with the function Oriented geometry.  
|     |     | You select the objects individually.  
| 6.5 | System status | You obtain information on the software version and on the workpiece.  

<table>
<thead>
<tr>
<th>Menu tree number</th>
<th>Functions</th>
<th>Function description</th>
</tr>
</thead>
</table>
| 7                | Program steps | This function provides an overview of your program steps. The program step headers are displayed. The machining operations can be influenced under the functions:  
- Edit  
- Insert  
- Delete  
- Move  
- Recalculate  
- Skip  
  **Note:** This function cannot be applied to geometries. |
| 7.1              | Edit      | The program step names and the individual program steps for machining with their parameter assignment forms (numeric parameters) can be modified. |
| 7.2              | Insert    | Program steps for machining are inserted. |
| 7.2.1            | Insert off| You terminate the **Insert** function. |
| 7.2.2            | Insert    | The **Insert** function is continued. |
| 7.3              | Delete    | You delete the entire program step. |
| 7.4              | Move      | You move individual program steps in the work schedule. |
| 7.6              | Recalculate| The parameters which may have been changed and the additions made for machining are recalculated.  
The display of the updated geometry with its machining is refreshed in the graphics field. |
| 7.7              | Skip      | You skip program steps from the work schedule or conditionally insert them.  
The program steps are not deleted. |
### 5.5 Tool types

#### 5.5.1 Milling cutters

<table>
<thead>
<tr>
<th>Tool Type</th>
<th>Diameter</th>
<th>Length</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>End mill</td>
<td>DT1</td>
<td>0.000m</td>
<td>1.55</td>
</tr>
<tr>
<td></td>
<td>LC1</td>
<td>0.000m</td>
<td></td>
</tr>
<tr>
<td></td>
<td>CR</td>
<td>0.000m</td>
<td></td>
</tr>
<tr>
<td></td>
<td>NUT</td>
<td>0</td>
<td></td>
</tr>
<tr>
<td></td>
<td>DS</td>
<td>0.000m</td>
<td></td>
</tr>
<tr>
<td>Diameter of shank</td>
<td>DT5</td>
<td>0.000m</td>
<td>179</td>
</tr>
<tr>
<td>Diameter of holder</td>
<td>DS1</td>
<td>0.000m</td>
<td></td>
</tr>
<tr>
<td>Length of holder</td>
<td>LS1</td>
<td>0.000m</td>
<td></td>
</tr>
<tr>
<td>Insert length</td>
<td>LT1</td>
<td>0.000m</td>
<td></td>
</tr>
<tr>
<td>Length dimension</td>
<td>LS</td>
<td>0.000m</td>
<td></td>
</tr>
</tbody>
</table>

![End mill diagram](image1)

**Fig. 5.11 End mill**

<table>
<thead>
<tr>
<th>Tool Type</th>
<th>Diameter</th>
<th>Length</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Face cutter</td>
<td>DT1</td>
<td>0.000m</td>
<td>179</td>
</tr>
<tr>
<td></td>
<td>LC1</td>
<td>0.000m</td>
<td></td>
</tr>
<tr>
<td></td>
<td>CR</td>
<td>0.000m</td>
<td></td>
</tr>
<tr>
<td></td>
<td>NUT</td>
<td>0</td>
<td></td>
</tr>
<tr>
<td></td>
<td>DS</td>
<td>0.000m</td>
<td></td>
</tr>
<tr>
<td>Diameter of shank</td>
<td>DT5</td>
<td>0.000m</td>
<td></td>
</tr>
<tr>
<td>Diameter of holder</td>
<td>DS1</td>
<td>0.000m</td>
<td></td>
</tr>
<tr>
<td>Length of holder</td>
<td>LS1</td>
<td>0.000m</td>
<td></td>
</tr>
<tr>
<td>Insert length</td>
<td>LT1</td>
<td>0.000m</td>
<td></td>
</tr>
<tr>
<td>Length dimension</td>
<td>LS</td>
<td>0.000m</td>
<td></td>
</tr>
</tbody>
</table>

![Face cutter diagram](image2)

**Fig. 5.12 Face cutter**
### Drill/groove mill

<table>
<thead>
<tr>
<th>Diameter 1</th>
<th>DT1</th>
<th>0.000 mm</th>
</tr>
</thead>
<tbody>
<tr>
<td>Cutting edge length</td>
<td>LC1</td>
<td>0.000 mm</td>
</tr>
<tr>
<td>Corner radius</td>
<td>CR</td>
<td>0.000 mm</td>
</tr>
<tr>
<td>Number of teeth</td>
<td>NUT</td>
<td>0</td>
</tr>
<tr>
<td>Degree of overlap</td>
<td>DS</td>
<td>0</td>
</tr>
</tbody>
</table>

| Diameter of shank | DT5 | 0.000 mm |
| Diameter of holder | DS1 | 0.000 mm |
| Length of holder  | LS1 | 0.000 mm |

| Insert length | LT1 | 0.000 mm |
| Length dimension | LS  | 0.000 mm |

---

**Fig. 5.13** Slotting mill

### T groove mill

<table>
<thead>
<tr>
<th>Diameter 1</th>
<th>DT1</th>
<th>0.000 mm</th>
</tr>
</thead>
<tbody>
<tr>
<td>Cutting edge length</td>
<td>LC1</td>
<td>0.000 mm</td>
</tr>
<tr>
<td>Corner radius</td>
<td>CR</td>
<td>0.000 mm</td>
</tr>
<tr>
<td>Number of teeth</td>
<td>NUT</td>
<td>0</td>
</tr>
<tr>
<td>Degree of overlap</td>
<td>DS</td>
<td>0</td>
</tr>
</tbody>
</table>

| Diameter of shank | DT5 | 0.000 mm |
| Diameter of holder | DS1 | 0.000 mm |
| Length of holder  | LS1 | 0.000 mm |

| Insert length | LT1 | 0.000 mm |
| Length dimension | LS  | 0.000 mm |

---

**Fig. 5.14** T-slot cutter

### Face mill

<table>
<thead>
<tr>
<th>Diameter 1</th>
<th>DT1</th>
<th>0.000 mm</th>
</tr>
</thead>
<tbody>
<tr>
<td>Cutting edge length</td>
<td>LC1</td>
<td>0.000 mm</td>
</tr>
<tr>
<td>Cutting edge angle</td>
<td>AT1</td>
<td>0.000 mm</td>
</tr>
<tr>
<td>Head length</td>
<td>LT1</td>
<td>0.000 mm</td>
</tr>
<tr>
<td>Number of teeth</td>
<td>NUT</td>
<td>0</td>
</tr>
<tr>
<td>Degree of overlap</td>
<td>DS</td>
<td>0</td>
</tr>
</tbody>
</table>

| Diameter of shank | DT5 | 0.000 mm |
| Diameter of holder | DS1 | 0.000 mm |
| Length of holder  | LS1 | 0.000 mm |

| Insert length | LT1 | 0.000 mm |
| Length dimension | LS  | 0.000 mm |

---

**Fig. 5.15** Face mill
Fig. 5.16  Corner mill (90 degrees)

Fig. 5.17  Angle mill (A)

Fig. 5.18  Angle mill (B)
### Side mill

<table>
<thead>
<tr>
<th>Diameter 1</th>
<th>DI1</th>
<th>0.000mm</th>
</tr>
</thead>
<tbody>
<tr>
<td>Cutting edge length</td>
<td>LC1</td>
<td>0.000mm</td>
</tr>
<tr>
<td>Corner radius</td>
<td>CR</td>
<td>0.000mm</td>
</tr>
<tr>
<td>Number of teeth</td>
<td>NS</td>
<td>0</td>
</tr>
<tr>
<td>Degree of overlap</td>
<td>DS</td>
<td>0</td>
</tr>
<tr>
<td>Diameter of shank</td>
<td>DT5</td>
<td>0.000mm</td>
</tr>
<tr>
<td>Diameter of holder</td>
<td>DS1</td>
<td>0.000mm</td>
</tr>
<tr>
<td>Length of holder</td>
<td>LS1</td>
<td>0.000mm</td>
</tr>
<tr>
<td>Insert length</td>
<td>LT1</td>
<td>0.000mm</td>
</tr>
<tr>
<td>Length dimension</td>
<td>LS</td>
<td>0.000mm</td>
</tr>
</tbody>
</table>

![Fig. 5.19 Side mill](image)

### Circular mill

<table>
<thead>
<tr>
<th>Diameter 1</th>
<th>DT1</th>
<th>0.000mm</th>
</tr>
</thead>
<tbody>
<tr>
<td>Cutting edge length</td>
<td>LC1</td>
<td>0.000mm</td>
</tr>
<tr>
<td>Corner radius</td>
<td>CR</td>
<td>0.000mm</td>
</tr>
<tr>
<td>Number of teeth</td>
<td>NS</td>
<td>0</td>
</tr>
<tr>
<td>Degree of overlap</td>
<td>DS</td>
<td>0</td>
</tr>
<tr>
<td>Diameter of shank</td>
<td>DT5</td>
<td>0.000mm</td>
</tr>
<tr>
<td>Diameter of holder</td>
<td>DS1</td>
<td>0.000mm</td>
</tr>
<tr>
<td>Length of holder</td>
<td>LS1</td>
<td>0.000mm</td>
</tr>
<tr>
<td>Insert length</td>
<td>LT1</td>
<td>0.000mm</td>
</tr>
<tr>
<td>Length dimension</td>
<td>LS</td>
<td>0.000mm</td>
</tr>
</tbody>
</table>

![Fig. 5.20 Circular mill](image)
5.5.2 Drills

**Fig. 5.21 Centre drill**

| Diameter 1 | DT1 | 0.000mm |
| Diameter 2 | DT2 | 0.000mm |
| Tip length | LT2 | 0.000mm |
| Tip angle | AT1 | 0.000deg |
| Diameter of holder | DS1 | 0.000mm |
| Length of holder | LS1 | 0.000mm |
| Insert length | LT1 | 0.000mm |
| Length dimension | LS | 0.000mm |

**Fig. 5.22 NC spot drill**

| Diameter 1 | DT1 | 0.000mm |
| Tip angle | AT1 | 0.000deg |
| Tip length | LE | 0.000mm |
| Diameter of holder | DS1 | 0.000mm |
| Length of holder | LS1 | 0.000mm |
| Insert length | LT1 | 0.000mm |
| Length dimension | LS | 0.000mm |

**Fig. 5.23 Twist drill**

| Diameter 1 | DT1 | 0.000mm |
| Tip angle | AT1 | 0.000deg |
| Tip length | LE | 0.000mm |
| Diameter of holder | DS1 | 0.000mm |
| Length of holder | LS1 | 0.000mm |
| Insert length | LT1 | 0.000mm |
| Length dimension | LS | 0.000mm |
### Solid Drill

<table>
<thead>
<tr>
<th>Diameter 1</th>
<th>DT1</th>
<th>0.000mm</th>
</tr>
</thead>
<tbody>
<tr>
<td>Diameter of holder</td>
<td>DS1</td>
<td>0.000mm</td>
</tr>
<tr>
<td>Length of holder</td>
<td>LS1</td>
<td>0.000mm</td>
</tr>
<tr>
<td>Insert length</td>
<td>LT1</td>
<td>0.000mm</td>
</tr>
<tr>
<td>Length dimension</td>
<td>LS</td>
<td>0.000mm</td>
</tr>
</tbody>
</table>

**Fig. 5.24** Rev. tip drill

### Metric ISO regular tap

<table>
<thead>
<tr>
<th>Diameter 1</th>
<th>DT1</th>
<th>0.000mm</th>
</tr>
</thead>
<tbody>
<tr>
<td>Core diameter</td>
<td>DT2</td>
<td>0.000mm</td>
</tr>
<tr>
<td>Cutting edge length</td>
<td>LC1</td>
<td>0.000mm</td>
</tr>
<tr>
<td>Tip angle</td>
<td>AT1</td>
<td>0.000deg</td>
</tr>
<tr>
<td>Tip length</td>
<td>LE</td>
<td>0.000mm</td>
</tr>
<tr>
<td>Diameter of shank</td>
<td>DT5</td>
<td>0.000mm</td>
</tr>
<tr>
<td>Diameter of holder</td>
<td>DS1</td>
<td>0.000mm</td>
</tr>
<tr>
<td>Length of holder</td>
<td>LS1</td>
<td>0.000mm</td>
</tr>
<tr>
<td>Insert length</td>
<td>LT1</td>
<td>0.000mm</td>
</tr>
<tr>
<td>Length dimension</td>
<td>LS</td>
<td>0.000mm</td>
</tr>
</tbody>
</table>

**Fig. 5.25** Metric ISO regular tap
Fig. 5.26 Metric ISO fine tap

Fig. 5.27 Whitworth tap

Fig. 5.28 Counterbore without pilot
**Fig. 5.29** Counterbore with pilot

<table>
<thead>
<tr>
<th>Diameter 1 (DT1)</th>
<th>0.000 mm</th>
</tr>
</thead>
<tbody>
<tr>
<td>Head length (LC1)</td>
<td>0.000 mm</td>
</tr>
<tr>
<td>Diameter cone (DT2)</td>
<td>0.000 mm</td>
</tr>
<tr>
<td>Cone length (LE)</td>
<td>0.000 mm</td>
</tr>
</tbody>
</table>

| Diameter of shank (DT5) | 0.000 mm |
| Diameter of holder (DS1) | 0.000 mm |
| Length of holder (LS1) | 0.000 mm |

| Insert length (LT1) | 0.000 mm |
| Length dimension (LS) | 0.000 mm |

**Fig. 5.30** Countersink

| Diameter 1 (DT1) | 0.000 mm |
| Diameter 2 (DT2) | 0.000 mm |
| Cutting edge length (LC1) | 0.000 mm |
| Tip angle (AT1) | 0.000 deg |

| Diameter of shank (DT5) | 0.000 mm |
| Diameter of holder (DS1) | 0.000 mm |
| Length of holder (LS1) | 0.000 mm |

| Insert length (LT1) | 0.000 mm |
| Length dimension (LS) | 0.000 mm |

**Fig. 5.31** Reamer

| Diameter 1 (DT1) | 0.000 mm |
| Core diameter (DT2) | 0.000 mm |
| Cutting edge length (LC1) | 0.000 mm |
| Tip angle (AT1) | 0.000 deg |
| Tip length (LE) | 0.000 mm |

| Diameter of shank (DT5) | 0.000 mm |
| Diameter of holder (DS1) | 0.000 mm |
| Length of holder (LS1) | 0.000 mm |

| Insert length (LT1) | 0.000 mm |
| Length dimension (LS) | 0.000 mm |
### 5.5 Tool types

#### Boring rod 1

<table>
<thead>
<tr>
<th>Diameter 1</th>
<th>DT1</th>
<th>0.090 mm</th>
</tr>
</thead>
<tbody>
<tr>
<td>Cutting edge length</td>
<td>LC1</td>
<td>0.090 mm</td>
</tr>
<tr>
<td>Diameter of shank</td>
<td>DT5</td>
<td>0.090 mm</td>
</tr>
<tr>
<td>Diameter of holder</td>
<td>DS1</td>
<td>0.090 mm</td>
</tr>
<tr>
<td>Length of holder</td>
<td>LS1</td>
<td>0.090 mm</td>
</tr>
<tr>
<td>Insert length</td>
<td>LT1</td>
<td>0.090 mm</td>
</tr>
<tr>
<td>Length dimension</td>
<td>LS</td>
<td>0.090 mm</td>
</tr>
</tbody>
</table>

![Boring rod 1 Diagram](image1)

**Fig. 5.32** Boring rod 1

#### Boring rod 2

<table>
<thead>
<tr>
<th>Diameter 1</th>
<th>DT1</th>
<th>0.090 mm</th>
</tr>
</thead>
<tbody>
<tr>
<td>Cutting edge length</td>
<td>LC1</td>
<td>0.090 mm</td>
</tr>
<tr>
<td>Tip length</td>
<td>LE</td>
<td>0.090 mm</td>
</tr>
<tr>
<td>Diameter of shank</td>
<td>DT5</td>
<td>0.090 mm</td>
</tr>
<tr>
<td>Diameter of holder</td>
<td>DS1</td>
<td>0.090 mm</td>
</tr>
<tr>
<td>Length of holder</td>
<td>LS1</td>
<td>0.090 mm</td>
</tr>
<tr>
<td>Insert length</td>
<td>LT1</td>
<td>0.090 mm</td>
</tr>
<tr>
<td>Length dimension</td>
<td>LS</td>
<td>0.090 mm</td>
</tr>
</tbody>
</table>

![Boring rod 2 Diagram](image2)

**Fig. 5.33** Boring rod 2
Fig. 5.34  Boring rod 3

Fig. 5.35  Boring rod 4
### 5.6 Error messages

The following table describes possible error messages in the Graphic Programming System. This table is valid for the software version SW 6 of the Turning and Drilling/Boring and Milling versions.

<table>
<thead>
<tr>
<th>Number</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Vocabulary word unknown</td>
</tr>
<tr>
<td>2</td>
<td>Label not found</td>
</tr>
<tr>
<td>3</td>
<td>File not found</td>
</tr>
<tr>
<td>4</td>
<td>Read error</td>
</tr>
<tr>
<td>5</td>
<td>Element not P, L, K</td>
</tr>
<tr>
<td>6</td>
<td>Element number wrong</td>
</tr>
<tr>
<td>7</td>
<td>Modifier wrong</td>
</tr>
<tr>
<td>8</td>
<td>Reference element not found</td>
</tr>
<tr>
<td>9</td>
<td>Contour transition not possible</td>
</tr>
<tr>
<td>10</td>
<td>Number of ref. elements wrong</td>
</tr>
<tr>
<td>11</td>
<td>Geometry language wrong or unknown</td>
</tr>
<tr>
<td>12</td>
<td>Element not found</td>
</tr>
<tr>
<td>13</td>
<td>Circle direction wrong</td>
</tr>
<tr>
<td>14</td>
<td>Transition element not possible</td>
</tr>
<tr>
<td>15</td>
<td>Write error</td>
</tr>
<tr>
<td>16</td>
<td>Too many synchronous points</td>
</tr>
<tr>
<td>17</td>
<td>Machine file has error</td>
</tr>
<tr>
<td>18</td>
<td>Macro not found</td>
</tr>
<tr>
<td>19</td>
<td>Parameter value wrong or missing</td>
</tr>
<tr>
<td>20</td>
<td>Memory full</td>
</tr>
<tr>
<td>21</td>
<td>Illegal file name</td>
</tr>
<tr>
<td>22</td>
<td>Too many conversions! Enter again!</td>
</tr>
<tr>
<td>23</td>
<td>Access rejected</td>
</tr>
<tr>
<td>24</td>
<td>Block too long</td>
</tr>
<tr>
<td>25</td>
<td>Partial machining exists</td>
</tr>
<tr>
<td>26</td>
<td>Internal error</td>
</tr>
<tr>
<td>27</td>
<td>Send error</td>
</tr>
<tr>
<td>28</td>
<td>Interface fault</td>
</tr>
<tr>
<td>29</td>
<td>Wire erosion</td>
</tr>
<tr>
<td>30</td>
<td>Overflow</td>
</tr>
<tr>
<td>31</td>
<td>Invalid numerical character</td>
</tr>
<tr>
<td>Number</td>
<td>Description</td>
</tr>
<tr>
<td>--------</td>
<td>--------------------------------------------------</td>
</tr>
<tr>
<td>32</td>
<td>Invalid operation character</td>
</tr>
<tr>
<td>33</td>
<td>Wrong handling of variable</td>
</tr>
<tr>
<td>34</td>
<td>Stack overflow</td>
</tr>
<tr>
<td>35</td>
<td>Wrong data type</td>
</tr>
<tr>
<td>36</td>
<td>Character chain length wrong</td>
</tr>
<tr>
<td>37</td>
<td>Delete not possible as reference exists</td>
</tr>
<tr>
<td>38</td>
<td>Skip not possible as reference exists</td>
</tr>
<tr>
<td>39</td>
<td>Move not possible reference exists</td>
</tr>
<tr>
<td>40</td>
<td>Design not possible</td>
</tr>
<tr>
<td>41</td>
<td>Contour not continuous</td>
</tr>
<tr>
<td>42</td>
<td>Parallel path not possible</td>
</tr>
<tr>
<td>43</td>
<td>In paraxial overall view only</td>
</tr>
<tr>
<td>44</td>
<td>4–window output not possible</td>
</tr>
<tr>
<td>45</td>
<td>Option not available</td>
</tr>
<tr>
<td>46</td>
<td>Impermissible input</td>
</tr>
<tr>
<td>47</td>
<td>Aborted with ESC</td>
</tr>
<tr>
<td>48</td>
<td>Label table full</td>
</tr>
<tr>
<td>49</td>
<td>Wrong view to select element</td>
</tr>
<tr>
<td>50</td>
<td>Surface unknown</td>
</tr>
<tr>
<td>51</td>
<td>Pocket milling</td>
</tr>
<tr>
<td>52</td>
<td>Faulty setup file</td>
</tr>
<tr>
<td>53</td>
<td>Label already defined</td>
</tr>
<tr>
<td>54</td>
<td>Standard block not found</td>
</tr>
<tr>
<td>55</td>
<td>Word not found</td>
</tr>
<tr>
<td>56</td>
<td>NC block nesting too deep</td>
</tr>
<tr>
<td>57</td>
<td>NC block not found</td>
</tr>
<tr>
<td>58</td>
<td>Word wrong</td>
</tr>
<tr>
<td>59</td>
<td>Error in macro</td>
</tr>
<tr>
<td>60</td>
<td>Contour not marked or wrongly</td>
</tr>
<tr>
<td>61</td>
<td>Contour change</td>
</tr>
<tr>
<td>62</td>
<td>No residual contour created</td>
</tr>
<tr>
<td>63</td>
<td>Cut segmentation not possible</td>
</tr>
<tr>
<td>64</td>
<td>Invalid path, check setup!</td>
</tr>
<tr>
<td>65</td>
<td>Too many blocks in machine file!</td>
</tr>
<tr>
<td>Number</td>
<td>Description</td>
</tr>
<tr>
<td>--------</td>
<td>--------------------------------------------------</td>
</tr>
<tr>
<td>66</td>
<td>Too many words in machine file!</td>
</tr>
<tr>
<td>67</td>
<td>Block in machine file too long!</td>
</tr>
<tr>
<td>68</td>
<td>Screenform not found</td>
</tr>
<tr>
<td>69</td>
<td></td>
</tr>
<tr>
<td>70</td>
<td>Error on exiting the system</td>
</tr>
<tr>
<td>71</td>
<td></td>
</tr>
<tr>
<td>72</td>
<td></td>
</tr>
<tr>
<td>73</td>
<td></td>
</tr>
<tr>
<td>74</td>
<td>Error in calculation</td>
</tr>
<tr>
<td>75</td>
<td>Write error (diskette full)</td>
</tr>
<tr>
<td>76</td>
<td>Transfer file not available</td>
</tr>
<tr>
<td>77</td>
<td>Error in transfer file</td>
</tr>
<tr>
<td>78</td>
<td>Error on storing (critical errors)</td>
</tr>
<tr>
<td>79</td>
<td>No geometry available</td>
</tr>
<tr>
<td>80</td>
<td>Error on opening the results file</td>
</tr>
<tr>
<td>81</td>
<td>Error on reading the results file</td>
</tr>
<tr>
<td>82</td>
<td>Contour not ascending in work file</td>
</tr>
<tr>
<td>83</td>
<td>Text 35 has errors (names standard blocks)</td>
</tr>
<tr>
<td>84</td>
<td>Element global (overwrite with 0!)</td>
</tr>
<tr>
<td>85</td>
<td>No NC block in editor</td>
</tr>
<tr>
<td>86</td>
<td>Setup file missing or faulty</td>
</tr>
<tr>
<td>87</td>
<td>NC block cannot be calculated</td>
</tr>
<tr>
<td>88</td>
<td>Error on opening the macro file</td>
</tr>
<tr>
<td>89</td>
<td>Error on reading the macro file</td>
</tr>
<tr>
<td>90</td>
<td>Specified macro not found</td>
</tr>
<tr>
<td>91</td>
<td>Max. 9 nestings in subprograms</td>
</tr>
<tr>
<td>92</td>
<td>Graphics macro file not available</td>
</tr>
<tr>
<td>93</td>
<td>NC file missing or faulty</td>
</tr>
<tr>
<td>94</td>
<td>NC macro file not available</td>
</tr>
<tr>
<td>95</td>
<td>Too many user graphics macros</td>
</tr>
<tr>
<td>96</td>
<td>Too many user NC macros</td>
</tr>
<tr>
<td>97</td>
<td>Too many Siemens graphics macros</td>
</tr>
<tr>
<td>98</td>
<td>Too many Siemens NC macros</td>
</tr>
<tr>
<td>99</td>
<td></td>
</tr>
<tr>
<td>Number</td>
<td>Description</td>
</tr>
<tr>
<td>--------</td>
<td>--------------------------------------------------</td>
</tr>
<tr>
<td>100</td>
<td>Tools, general error</td>
</tr>
<tr>
<td>101</td>
<td>Error tool</td>
</tr>
<tr>
<td>102</td>
<td>Tool not found</td>
</tr>
<tr>
<td>103</td>
<td>Tool catalog not found</td>
</tr>
<tr>
<td>104</td>
<td>No tool in magazine</td>
</tr>
<tr>
<td>105</td>
<td>No tool catalog</td>
</tr>
<tr>
<td>106</td>
<td>ID number twice</td>
</tr>
<tr>
<td>107</td>
<td>Error on assigning variables</td>
</tr>
<tr>
<td>108</td>
<td>No material list available</td>
</tr>
<tr>
<td>109</td>
<td>Error in material</td>
</tr>
<tr>
<td>110</td>
<td>No workholder in magazine</td>
</tr>
<tr>
<td>111</td>
<td>No tool or workholder in magazine</td>
</tr>
<tr>
<td>112</td>
<td>Insert length of tool too small</td>
</tr>
<tr>
<td>113</td>
<td>Layer not yet defined</td>
</tr>
<tr>
<td>114</td>
<td>Insufficient internal memory</td>
</tr>
<tr>
<td>115</td>
<td>Vocabulary word cannot be edited</td>
</tr>
<tr>
<td>116</td>
<td>No tool in catalog</td>
</tr>
<tr>
<td>117</td>
<td>No workholder in catalog</td>
</tr>
<tr>
<td>118</td>
<td>No tool or workholder in catalog</td>
</tr>
<tr>
<td>119</td>
<td>No machine file loaded</td>
</tr>
<tr>
<td>120</td>
<td>General error workholder</td>
</tr>
<tr>
<td>121</td>
<td>No active workholder found</td>
</tr>
<tr>
<td>122</td>
<td>Contour not workholder</td>
</tr>
<tr>
<td>123</td>
<td>Workholder requires designation</td>
</tr>
<tr>
<td>124</td>
<td></td>
</tr>
<tr>
<td>125</td>
<td></td>
</tr>
<tr>
<td>126</td>
<td>Printer not ready!</td>
</tr>
<tr>
<td>127</td>
<td>SW module available</td>
</tr>
<tr>
<td>128</td>
<td>Geometry too long</td>
</tr>
<tr>
<td>129</td>
<td>System cannot be saved</td>
</tr>
<tr>
<td>130</td>
<td>Error in variable geometry</td>
</tr>
<tr>
<td>131</td>
<td>No pre–drill points found</td>
</tr>
<tr>
<td>132</td>
<td>Too many subroutines</td>
</tr>
<tr>
<td>133</td>
<td>Subroutine not found</td>
</tr>
<tr>
<td>134</td>
<td>Limit of subroutine calls reached</td>
</tr>
<tr>
<td>Number</td>
<td>Description</td>
</tr>
<tr>
<td>--------</td>
<td>-------------</td>
</tr>
<tr>
<td>135</td>
<td></td>
</tr>
<tr>
<td>136 to 155</td>
<td>Position internal/external cannot be defined</td>
</tr>
<tr>
<td>156</td>
<td>Block headers in TEXT.TXT and STEUER.TXT different</td>
</tr>
<tr>
<td>157 to 169</td>
<td>Cannot open TEXT.TXT</td>
</tr>
<tr>
<td>170</td>
<td>Unknown block header in TEXT.TXT</td>
</tr>
<tr>
<td>171</td>
<td>Cannot open STEUER.TXT</td>
</tr>
<tr>
<td>172</td>
<td>Error on reading of TEXT.TXT</td>
</tr>
<tr>
<td>173</td>
<td>Error on reading of STEUER.TXT</td>
</tr>
<tr>
<td>174</td>
<td>Syntax error at beginning of block header</td>
</tr>
<tr>
<td>175</td>
<td>Syntax error at end of block header</td>
</tr>
<tr>
<td>176</td>
<td>Error on converting tool catalog</td>
</tr>
<tr>
<td>177</td>
<td>Syntax error in subblock header</td>
</tr>
<tr>
<td>178</td>
<td>Incorrect line number</td>
</tr>
<tr>
<td>179</td>
<td>Not enough memory space</td>
</tr>
<tr>
<td>180</td>
<td>Error in {...}</td>
</tr>
<tr>
<td>181</td>
<td>Entered data too long</td>
</tr>
<tr>
<td>182</td>
<td>Access refused; line number does not exist</td>
</tr>
<tr>
<td>183</td>
<td>Cannot reduce memory space</td>
</tr>
<tr>
<td>184</td>
<td>Wrong end of line</td>
</tr>
<tr>
<td>185</td>
<td>Unexpected end of file (EOF)</td>
</tr>
<tr>
<td>186</td>
<td>Unknown error</td>
</tr>
<tr>
<td>187</td>
<td>Tool master data not available</td>
</tr>
<tr>
<td>188</td>
<td>Machining not possible</td>
</tr>
<tr>
<td>189</td>
<td>Error on creating of work schedule</td>
</tr>
<tr>
<td>190 to 200</td>
<td>Error on create tool list</td>
</tr>
<tr>
<td>201</td>
<td>Error on creating tool catalog</td>
</tr>
<tr>
<td>202</td>
<td>No cutting values for current material</td>
</tr>
<tr>
<td>Number</td>
<td>Description</td>
</tr>
<tr>
<td>--------</td>
<td>-----------------------------------------------------------------</td>
</tr>
<tr>
<td>207</td>
<td>Max. clamping diameter too small</td>
</tr>
<tr>
<td>208</td>
<td>Min. clamping diameter too large</td>
</tr>
<tr>
<td>209</td>
<td>Sleeve too small</td>
</tr>
<tr>
<td>210</td>
<td>Sleeve stroke too small</td>
</tr>
<tr>
<td>211</td>
<td>No blank available</td>
</tr>
<tr>
<td>212</td>
<td>Field for vocabulary data L/_INT too small</td>
</tr>
<tr>
<td>213</td>
<td>Unknown machining</td>
</tr>
<tr>
<td>214</td>
<td></td>
</tr>
<tr>
<td>215</td>
<td>No memory available</td>
</tr>
<tr>
<td>216</td>
<td>Wrong call for obtaining memory</td>
</tr>
<tr>
<td>217</td>
<td>Wrong call for releasing memory</td>
</tr>
<tr>
<td>218 to 250</td>
<td></td>
</tr>
</tbody>
</table>
Appendix

5.6 Error messages
6 Index

A
Absolute, 4–41
All, 4–29, 4–74, 5–22
Angle data, 4–43
Angle definition, 4–49
Annular groove, 5–17
Any hole pattern, 5–18
Approach distance, 4–81
Approach strategy, 4–80
Autozoom, 5–30

B
Beginning of program, 4–73
Blank, 5–13
Bore to DIN, 5–25

C
Cartesian, 4–42
Center drill, 5–25
Chamfering, rounding, 4–43
Change tools, all, T–D–number, geometry data, technology, 4–29
Circle definition, 4–55
Circle of holes, 5–18
Circular groove, 5–17
Colors, 4–44
Construction geometry, 4–35, 4–48, 5–14 angle definition, 4–49
circle definition, 4–55
line definition, 4–52
point definition, 4–50
Continuation form, 3–2
Contour, 5–16
delete, 5–18edit, 5–18
Contour milling, 4–77, 5–23
construction geometry, 4–78
contour direction, 4–78
oriented geometry, 4–78
part contours, 4–89
standard geometry, 4–78
standard start point, 4–78
Contour transition element, 4–43
Coordinate zero point, 4–17
Copy, 4–68, 5–18
Countersink, 5–25
Cutting values, 4–31, 4–79

D
D number, 4–24, 4–29, 4–74, 5–22
Data from magazine, 4–34
Deep hole, 5–25
Diameter, values, input of, 4–42
DIN block, insert, 4–79
DIN isometry, 5–29
Drilling, 5–25

E
Edit
contour, 5–18
elements, 4–46
Editing, contour elements, 4–46
Elongated hole, 5–17
Empirical values, 4–2
End plane, 4–85
Error messages, 5–44
EX (end point X), EZ (end point Z), 4–42

F
File environment, 4–1
Free definition, 5–19
Front view, 5–29

G
Geometry, 4–35, 5–2, 5–13
create new, 5–13
turning, 5–8
Geometry data, 4–29, 4–74, 5–22
Global contour transitions, 4–38, 4–49
Graphic programming, 4–16
Groove, 4–67, 5–16
annular groove, 5–17
circular groove, 5–17
elongated hole, 5–17
n corner, 5–17
radial groove, 5–17
rectangular groove, 5–17

H
Help function, 2–1
Index

Hole box, 5–18
Hole pattern, define, 4–66
Holes, 4–64, 4–65, 5–17
    creating, 4–65
    skipping, 4–66

I
Identification number, 4–25, 4–74, 5–22
Incremental, 4–41
Infeed depth IF, 4–85
Info, 5–32
    angle/spacing, 5–32
    construction element, 5–32
    Contour dist./angl, 5–32
    Contour element, 5–32
Information fields, 4–42
Information line, 3–6, 4–44
Initial point, 4–39
Input fields, 4–42
Insert block, 5–26
Insert blocks, 4–107
    unassigned DIN code, 4–108
Interactive screenform control, 3–2
Island height, 4–96

K
Key function
    interactive screenform control, 3–4
    selection control, 3–7, 3–8, 3–9
Key functions, other, 3–10

L
Line definition, 4–52
Lower, 4–93

M
Machine and help functions, 4–106
Machine help functions, 5–26
Machining, 4–70, 5–2, 5–20, 5–22
    drilling, 4–104
    insert blocks, 4–107
    Machine and help functions, 4–106
    milling, 4–76
        contour milling, 4–77
        manual milling, 4–97
        solid milling, 4–91
        surface milling, 4–101
    simulation, 4–110
Magazine
    load, 5–5
    new, 5–5
    save, 5–5
    tool master data, 4–72
    user magazine, 4–72
Manual milling, 4–97, 5–24
Master data, 4–23, 4–72
    extend, 4–26
    load, 5–5
    save, 5–5
Master data magazine, 5–5
Material, data, 5–7
    change, 5–7
    save, 5–7
Materials, 4–31
Menu tree
    blank, 5–9
    construct. geometry, 5–9
    construction geometry, 5–10
    create new, 5–9
    drilling, 5–20
    edit elements, 5–9
    finished part, 5–9
    geometry, 5–8
    groove, 5–9
    holes, 5–9, 5–12
    horizontal, 5–1
    machining, 5–20
    milling, 5–20
    oriented geometry, 5–9
    standard geometry, 5–9, 5–11
    tools and material, 5–4
    vertical, 5–27
Milling, 4–76, 5–22
    contour milling, 4–77
    manual milling, 4–97
    solid milling, 4–91
    surface milling, 4–101
Mirror, 5–18
N
N corner, 5–16
Narrow point offset, 4–87

O
Operation, 3–1
Oriented geometry, 4–36, 5–13
   edit elements, 4–46
   insert elements, 4–44
   select variants, 4–46

P
Parallel contour, 5–18
Part program, 4–108, 4–114, 5–3, 5–26
part program, 4–108
Peripheral surface, 4–20, 5–19
Plan view, 5–29
Pocket calculator function, 4–128
Point definition, 4–50
Point elements, define, 4–66
Polar, 4–42
Program info, 5–32
Program steps, 4–115, 5–33
   delete, 4–122, 5–33
   edit, 4–118, 5–33
   info, 4–124
   insert, 4–120, 5–33
   move, 4–123, 5–33
   recalculate, 4–125, 5–33
   skip, 4–126, 5–33
Programming functions, 4–1
   empirical values, 4–2
   geometry, 4–36
   holes, 4–64, 4–65
   machining, 4–70
   tools, 4–23

R
Radial grooves, 5–17
Raise, 4–94
Read, 5–28
   auto save, 4–127, 5–28
   new, 5–28
   original, 5–28
Rectangle, 5–16
Rectangular groove, 5–17
Refresh, 5–30
Reverse mirror, 5–18
Rotate (copy), 5–18
Rotate isometry, 5–29
Row of holes, 5–18

S
Safety distance, 4–80, 4–81
Safety plane, 4–81
Save, 4–127, 5–28
Save setup, 5–31
Scaling, 5–18
Screen
   enter viewport, 5–30
   move viewport, 5–30
   refresh, 5–30
   representation, 5–31
      All planes on/off, 5–31
      dimensioning on/off, 5–31
      direction arrows on/off, 5–31
      Display plane select, 5–31
      save setup, 5–31
      select viewport, 5–30
      set viewport, 5–30
Select control, 3–6
Selection form, 3–3
Side view, 5–29
Simulation, 4–110, 5–3, 5–26
Single hole, 5–18
Skip, NC blocks, 4–106
Solid milling, 4–91, 5–23
   parallel to the axis, 4–96
Standard geometry, 4–35
   blank/contour, 5–16
   groove, 5–16
Start plane, 4–85
Start point, contour milling, 4–78
Strategy, path milling (contour milling), 4–87
Surface
   change, 4–22
   create, 4–18, 5–19
   delete, 4–22, 5–19
   edit, 4–22, 5–19
   from geometry, 4–19, 5–19
   modify, 5–19
   select, 4–22, 5–2, 5–19, 5–26
Surface milling, 4–101, 5–23
   parallel to the axis, 4–103

T
T number, 4–24, 4–29, 4–74, 5–22
Technology, 4–29
Thread, 5–25
Tool, 5–5
  change, 5–6
classes, 4–30
copy, 4–28, 5–6
delete, 5–7
new, 5–6
Tool magazine, tool master data catalog, 4–23
Tool position, 4–86
Tool selection, 4–74, 5–22
  functions, 4–74
Tool types, 5–34
  drill, 5–38
  boring rod 1, 5–42
  boring rod 2, 5–42
  boring rod 3, 5–43
  boring rod 4, 5–43
  centre drill, 5–38
  counterbore with pilot, 5–41
  counterbore without pilot, 5–40
  countersink, 5–41
  metric ISO fine tap, 5–40
  metric ISO regular tap, 5–39
  NC spot drill, 5–38
  reamer, 5–41
  solid drill, 5–39
  twist drill, 5–38
  Whitworth tap, 5–40
milling cutters, 5–34
  angle mill (A), 5–36
  angle mill (B), 5–36
  circular mill, 5–37
  corner mill (90 degrees), 5–36
  end mill, 5–34
  face cutter, 5–34
  face mill, 5–35
  side mill, 5–37
  slotting mill, 5–35
  T–slot cutter, 5–35
Tools, 4–23
  cutting values, 4–31
  materials, 4–31
Tools and material, 5–2, 5–4, 5–5

Translate, 5–18

U

User magazine, 4–23, 4–72
  create new, 4–28

V

Value input, 5–25
Variants, select, 4–46
Vertical softkey menu, 5–27
Viewport
  enter, 5–30
  move, 5–30
  select, 5–30
  set, 5–30
Views, 5–29
  activate 4 views, 5–29
  DIN isometry, 5–29
  front view, 5–29
  plan view, 5–29
  rotate isometry, 5–29
  set 4 views, 5–29
  side view, 5–29

W

Withdrawal distance, 4–81
WOP, exit, 4–134
Work schedule, 4–115
Workholder, classes, 4–30

Z

Zoom, 5–30
  autozoom, 5–30