Graphic Programming System
Software Version 6
Drilling/Boring and Milling
Part 1: Programming Examples

User’s Guide
User Documentation

Valid for

Control Software Version
SINUMERIK 840C/CE as from SW 4
(Standard/Export Version)

07.97 Edition
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>6FC5198–6AA20–0BP0</td>
<td>A</td>
</tr>
<tr>
<td>09.96</td>
<td>6FC5198–6AA20–0BP1</td>
<td>C</td>
</tr>
<tr>
<td>07.97</td>
<td>6FC5198–6AA20–0BP2</td>
<td>C</td>
</tr>
</tbody>
</table>

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 1997. All rights reserved.

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

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

Subject to change without prior notice.
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 “Graphic Programming System User’s Guide Part 1” describes

basic graphic programming principles with the help of two simple practical examples.

These examples are useful not only as an introduction but also for later reference.

All the keys that you will require for programming the examples are shown as icons.

Refer to Section 5 when programming multi-plane machining.

When you program the examples, it is very important that you refer to the notes in Part 2.

Note

Before using the functions provided by the Graphic Programming System in practice, you should have worked through the programming example 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 1: Programming Examples” 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: Objective
Section 3 describes the objectives of the programming examples in Sections 4 and 5.

Section 4: Programming Example 1
Section 4 illustrates the procedure applied for graphic programming by using an example without multi-side machining.

Section 5: Programming Example 2
Section 5 illustrates the procedure applied for graphic programming by using an example with multi-side machining.

Section 6: Appendix
The appendix contains fold-out worksheets and workpiece sketches for the programming examples.

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 Part 2 Section “Operation” 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”.

## Contents

1. Introduction ........................................................................................................ 1–1
2. Help Function ...................................................................................................... 2–1
3. Exercise Objectives ............................................................................................. 3–1
4. Programming Example 1 .................................................................................... 4–1
   4.1 Graphic programming ...................................................................................... 4–2
   4.2 Tools ............................................................................................................... 4–4
   4.2.1 Creating tools in tool master data .............................................................. 4–4
   4.2.2 Change tools ............................................................................................. 4–12
   4.2.3 Save tool master data ............................................................................... 4–14
   4.2.4 Create new – copy user magazine ............................................................ 4–15
   4.3 Graphic programming of geometry ............................................................... 4–19
   4.3.1 Blank contour ....................................................................................... 4–19
   4.3.2 Standard geometry/rectangle ................................................................. 4–24
   4.3.3 Views – 4 windows ................................................................................. 4–27
   4.3.4 Views – plan view .................................................................................... 4–29
   4.3.5 Finished part contour ............................................................................... 4–29
   4.3.6 Standard geometry/rectangle ................................................................. 4–31
   4.3.7 Screen – select viewport ......................................................................... 4–33
   4.3.8 Construction geometry .......................................................................... 4–37
   4.3.9 Creating contour construction geometry elements .................................. 4–41
   4.3.10 Edit construction geometry .................................................................. 4–45
   4.3.11 Create contour construction geometry elements .................................... 4–46
   4.3.12 Contour definition .................................................................................. 4–55
   4.3.13 Element transition .................................................................................. 4–57
   4.3.14 Edit programmed contours ................................................................... 4–61
   4.3.15 Oriented geometry ................................................................................. 4–62
   4.3.16 Delete contour ....................................................................................... 4–63
   4.3.17 Create contour using oriented geometry ............................................... 4–64
   4.3.18 Holes – hole box ................................................................................. 4–72
   4.3.19 Hole as cutter insert point ...................................................................... 4–75
   4.4 Graphic programming of drilling and milling ................................................. 4–77
   4.4.1 Machining ............................................................................................... 4–78
   4.4.2 Machining, milling ............................................................................... 4–81
   4.4.3 Tool selection ....................................................................................... 4–81
   4.4.4 Solid milling of an island ........................................................................ 4–83
   4.4.5 Solid milling of a pocket ........................................................................ 4–85
   4.4.6 Machining, drilling .............................................................................. 4–89
   4.4.7 Machining simulation ............................................................................ 4–94
   4.4.8 Create part program ............................................................................. 4–96
   4.4.9 Exit programm. WOP function .............................................................. 4–101
1 Introduction

This User’s Guide describes operator actions and programming functions for the Graphic Programming System.

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 can 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 surfaces perpendicular to the design plane]

Fig. 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.

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

Fig. 2.1 Graphic help for a help form

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.

---

**Fig. 2.2 Graphic help**
3 Exercise Objectives

You will work step by step through sections 4 and 5 to fulfill the exercise objectives of:

- Creating a tool magazine for boring and milling tools
- Entering the workpiece contour on the milling level or surface “front”\(^1\) and “right”\(^1\)
- Programming the machining of a workpiece using boring and drilling technologies and
- Creating a part program.

A fold-out workpiece sketch for this example is contained in the Appendix. This enables you to work through the programming example page by page with the workpiece sketch folded out.

Note

If you want to stop before the end of the exercise, proceed as follows:

- Make a note of the programming level (e.g.: `Geometry/Create_new/Blank`)
- Save the exercise in its present state (vertical softkey `Save`).

The interrupted programming example can be continued by

- Starting the graphic programming system with the workpiece, for example, “PART1”
- Calling the softkey functions at the noted programming level.

\(^1\) Only if multi-side machining option is available
4 Programming Example 1

This section describes

• How to create a tool and a tool magazine
• How to enter the workpiece geometry (contour)
• How to program machining of the workpiece (technology), and
• The main operating control and display elements.

A fold-out workpiece sketch for this example is contained in the Appendix. This enables you to work through the programming example page by page with the workpiece sketch folded out.

The keys correspond to the symbols on the operator panel.

The Environment Description contains the key symbols that differ from those shown on the operator panel.
4.1 Graphic programming

Starting point

You have created a workpiece directory called “PART1”.

How to do this is explained step by step in the section “Programming Examples” (e.g., “Graphic Programming System on the PC”) of the corresponding Environment Description.

The Graphic Programming System is loaded. You start off from this operator interface:

![Graphic Programming System, basic level](image)

Fig. 4.1 Graphic Programming System, basic level

Note

You now have seven vertical softkeys in addition to the seven horizontal softkeys.

The horizontal softkeys take you into the programming of your workpiece “PART1”.

The vertical softkeys are used for screen and system control and contain general programming utilities which are available at any time during graphic programming.

The RECALL key enables you to abort a softkey function and to go back in the menu.

- Before you activate the Machining softkey, a geometry must be available.
- If new tools are required, it is recommended to generate the tools before creating the workpiece geometry. You will practise this procedure in the “Programming tools” subsection below.
Notes regarding the WOP basic level

Milling plane\(^1\) is always the default setting in graphic programming (see Fig. 4.2). The current surface can be seen in the top right of the graphics area.

Use the Select surface\(^1\) function to select a particular plane from the submenu under the Define surface\(^1\) function.

This plane can be a surface of the box (see “User’s Guide Part 2: Operator/Programming Functions”) or any surface in the space.

The string “Milling plane” appears to the right above the graphics area immediately you select the Geometry function.

\(^1\) Only if multi-side machining option is available
4.2 Tools

Tools exercise (duration approx. 45 mins.)

Tools are organized in a master data catalog or in user magazines.
See the User’s Guide “Part 2: Operator/Programming Functions” for programming tips and definitions for the master data or user magazine.

Now fold out the workplan in the appendix (see section 6.2).

4.2.1 Creating tools in tool master data

You need tools to machine the workpiece programmed in the following exercise.

Objective
Add a new tool to the tool master data as described below.

Starting point
The Graphic Programming System is loaded

Press the Tools Material softkey in the horizontal softkey menu to start tool programming.

Tool programming is then automatically active and the following display appears:

![Image of tool programming interface]

Fig. 4.3 Tools Material

Note
You add new workholders or tools to a master data catalog.

You copy the individual workholders or tools from this catalog into a user magazine.

To add new tools to the master data, first load the tool master data in the programming system.
Press the **Masterdata/magazine** softkey.

Press the **Load masterdata** softkey.

While loading, you will see the following display:

![Diagram](image)

**Fig. 4.4 Load masterdata**

While operating, the programming system provides you with information. This information is shown in the line below the graphics area. The programming level in which you are currently working is shown in the line above the graphics area.

A moving bar shows you that the programming system is active.

Having loaded the tool master data, you can start to add tools.

Press the **Tool** softkey and then the **New tool** softkey.

The interactive screenform for **Tool input** appears (see Fig. 4.5).
You require the following tools for machining your workpiece (see Section 6.2):

- Two end mills (5 mm and 20 mm),
- A twist drill with a diameter of 10 mm and
- A solid drill with a diameter of 10 mm.

**Note**

SIEMENS AG supplies a catalog containing several predefined tools with the program package “Graphic Programming System Drilling/Boring and Milling” for SINUMERIK 840C.

The tools have been prepared for this exercise.

An end mill (20 mm), the twist drill and the solid drill are already contained in the master data catalog.

Create an end mill (5 mm) for the purposes of this exercise.

Select type of machining **milling**

by pressing the **INPUT** key.

The cursor bar is now on the tool type **end mill**.

Now press the **INPUT** key.
The **General tool data** interactive screenform appears for entering the tool data. Enter as parameters the values that are already given in the following figure.

**Fig. 4.6  General tool data**

Use the alphanumeric keys and the cursor keys for entering parameters.

The cursor is in one line of the interactive form.

Move this to the **Description** line

and enter “SAF 5/16/R HM P20–P35”.

Move the cursor to the **ID number**

and enter “7100050”.

The **ID number** is a type designation. The ID number must be unique. The user codes for example: manufacturer code

The T No. for the magazine location and the D No. for the storage location are not defined in the tool master data. In this exercise, you determine these values in the user magazine.
Move the cursor to the input field \textit{mm/min}.

The existence of a continuation screenform is indicated by a black triangle next to the input field.

Open the continuation screenform and a selection menu appears.

Using the cursor, select \textit{mm/tooth} and

Confirm the selection with the \textbf{INPUT} key.

Close the interactive form with the \textbf{INPUT} key.

The system now puts the \textbf{End mill} interactive form on the screen.

Using the cursor keys and alphanumeric keys enter the values shown in the above \textit{End mill} figure (Fig. 4.7). You do not have to enter the decimal points or the zeros behind the decimal points. These are generated automatically by the system.
**Note**

Some key functions for controlling within an interactive screenform are explained in the table below.

<table>
<thead>
<tr>
<th>Operator panel 840C</th>
<th>Function when controlling an interactive screenform</th>
</tr>
</thead>
<tbody>
<tr>
<td>![Arrow]</td>
<td>With this key, you put the cursor in the first input field and accept the parameters that have been input.</td>
</tr>
<tr>
<td>![Arrow]</td>
<td>With the end key, you go from input field to input field <strong>backwards</strong> and accept the parameters that have been input.</td>
</tr>
<tr>
<td>![Arrow]</td>
<td>With these keys, you control the cursor when entering parameters within a field.</td>
</tr>
<tr>
<td>![Arrow]</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>![Delete]</td>
<td>The delete key deletes the character to the left of the cursor.</td>
</tr>
<tr>
<td>![Arrow]</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>![Screen Form]</td>
<td>If a black triangle can be seen to the right of an input field, open a continuation screenform with this key.</td>
</tr>
<tr>
<td>![Alphanumeric]</td>
<td>Use the alphanumeric keys to enter parameters.</td>
</tr>
<tr>
<td>![Recall]</td>
<td>The <strong>RECALL</strong> key interrupts interactive form input and the system ignores all inputs.</td>
</tr>
<tr>
<td>![Input]</td>
<td>With the <strong>INPUT</strong> key, you end screenform inputs and accept all entries.</td>
</tr>
<tr>
<td>![Calculator]</td>
<td>You can call the <strong>calculator function</strong> with this key if you are in an interactive screenform and the cursor is positioned in a numerical input field.</td>
</tr>
<tr>
<td>![Text Selection]</td>
<td>If the cursor is positioned on a text selection field, you can insert the texts into this field one after the other with this key.</td>
</tr>
</tbody>
</table>

With the **INPUT** key, transfer the parameters that you have entered into the interactive screenform.
The programming system computes the tool geometry and displays it schematically in the graphics area (see Fig. 4.8).

![Graphical representation of end mill](image)

**Fig. 4.8** Graphical representation of end mill

**Note**

If the tool is not quite what you want, page up the previous interactive screenforms using the **Edit** softkey and change the parameters once again for your tool.

When the end mill represents the tool that you require then press the softkey **Accept**.
Now enter the cutting values in the interactive screenform that now appears (see Fig. 4.9). The cutting values refer to the material of the workpiece to be machined with this mill. The material is displayed in the second column of the screenform.

![Fig. 4.9 Cutting values](image)

For details relating to this interactive screenform, please refer to the User’s Guide “Part 2: Operator/Programming Functions”.

Using the alphanumeric keys, enter the values shown above (Fig. 4.9) into the interactive screenform.

Begin with a cutting velocity $V = \text{90 m/min}$ and then enter the feed values (FT1, FR and FTR).

Place the cursor on Accept, and accept the cutting values of the end mill for this material.

The Programming System offers the following material (No. 2) with its cutting values.

---

1) Derived from cutting velocity, time and cutter diameter.
For this exercise, one material is enough.
Put the cursor on OK.

You conclude the whole function with OK.
The function OK does not accept any parameters.

Then conclude parameter input.

You have now created a new tool within the tool master data by means of the programming system.

4.2.2 Change tools

Objective
Before learning in the following exercise (next section)
- How to save the new tool in the tool master data and then
- To copy it into a new user magazine
check whether the tool you have created is actually in the tool master data.

Procedure
To do this, press the Change tool softkey and then All.

The cursor is now on Tool class End mill.

On pressing this key, the selection form Tool description opens for the tools of the End mill type.
In this display you can see the tool that has just been created.

Place the cursor on the tool description

SAF 5/16/R HM P20–P35

After pressing the INPUT key, you can select the parameters that are to be modified using the functions

- All
- T and D number
- Geometry data
- Technology.

No modifications are required for the tool used in this exercise, so you can now quit this function by pressing the RECALL key.
4.2.3 Save tool master data

Always save the new data after concluding tool generation.

Objective
The following section describes how you
- Save the new tool in the tool master data, and then
- Copy into a new user magazine.

Initial situation
You are at the programming level Tools_Material/MasterdataMagazines/Tools.

If you create a workpiece geometry without then saving it, this data will be lost.

Do not forget:
1. Load the tool master data or magazine,
2. first program the tool,
3. then save the tool master data or magazine.

Procedure

Press the RECALL key to get into the next higher programming level,

and press the softkey Save masterdata in order to save the tool master data.

In the following section you will create the user magazine for the machining of the workpiece “PART1”.

You copy the tools from the master data to the user magazine.
4.2.4 Create new – copy user magazine

Objective
You create a user magazine ("Exercise") with the tools
- End mill (5 mm, 20 mm),
- Twist drill (10 mm diameter),
- Solid drill (10 mm diameter).

You copy these tools from the tool master data.

You are now at this programming level:

![Fig. 4.11 Master data magazine](image)

Procedure

Press the **New magazine** softkey.

The vertical softkey function **Program info** tells you, for example, which magazine is active.

At the moment, the current tool data are the master data. You have already saved these in the exercise.

You answer **yes** to the question **Existing tool data will be overwritten** put by the programming system.

Press the **INPUT** key.

**Note**

When you **Save magazine**, you save the new user magazine under a name (see Fig. 4.12, Page 4–18).
You can copy tools from the tool master data to the new user magazine.

**End mill for the stock outside of the island**

First you copy the end mill for the area outside of the island.

Position the cursor on **End mill** and press the **INPUT** key.

You are now in the directory **End mill** and move the cursor to the **End mill SAF 20/32/R HM P20–P35**.

Confirm the selection.

Check the values for the **General tool data** and change them if necessary (e.g.: T No. or D No.).

Copy the end mill (20 mm) to the magazine.

**End mill for the pocket**

Now copy the end mill for the pocket.

Position the cursor on **End mill** and press the **INPUT** key.

Move the cursor to the **End mill SAF 5/16/R HM P20–P35** and confirm the selection.

Check the values for the **General tool data** and for the purposes of this exercise change the T No. and D No. to “20”.

Then copy the end mill (5 mm) to the magazine.
**Twist drill for the holes**

For the four holes, you need a drill with a diameter of 10 mm.

Position the cursor on **Twist drill**.

Press the **INPUT** key.

Select the drill with diameter 10 mm.

The name of the drill in the **Tool description** is **SPB 10.0/31/118/R HSS**.

With the **INPUT** key, select the drill with diameter 10 mm, enter a T No. or a D No. if needed and conclude the copying function for the twist drill.

---

**Solid drill for the relief hole when pocket milling**

You need a solid drill which puts a relief hole in a pocket for the milling cutter.

Position the cursor on **Solid drill**.

Press the **INPUT** key.

Position the cursor on the tool description **VOB 10/42/R VHM**.

With the **INPUT** key select the solid drill with diameter 10 mm, enter a T No. or a D No. if needed and conclude the copy function for the solid drill.

You exit the function **New magazine** with the **RECALL** key.

and then press the **Save magazine** softkey.
The program displays the interactive screenform **Data access**.

![Data access](image)

Fig. 4.12 Data access

Type in the name “Exercise” on the alphanumeric keyboard.

Save the new tool user magazine under the name “Exercise” by pressing the **INPUT** key.

You have now created the user magazine.

Now press the **RECALL** key twice to start generating the geometry.
4.3 Graphic programming of geometry

Steps involved in the exercise “Create blank and finished part geometry” (duration approx. 120 min.)

Objective
The following sections describe how you graphically program the finished part contours.

Initial situation
You have created a user magazine and now wish to define the geometry.
- You are in workpiece directory “PART 1”.
- The graphic programming system is active.

4.3.1 Blank contour

When programming, we advise you to start with the blank contour.

The programming system guides you in your workpiece programming through a menu until NC blocks have been generated.

Objective
This section describes how you graphically program the blank contour of the workpiece “PART 1” as a standard contour.

Initial situation
- You are in workpiece directory “PART 1”.
- The graphic programming system is active.

Procedure

Now unfold the workpiece sketch in the Appendix (see Section 6.1).

Begin your graphic programming of the blank contour by pressing the Geometry softkey in the horizontal softkey menu.

Then press the softkey Create new. The following display appears.
In the first exercise, you will program the blank contour shown in the dotted background in the sketch (Fig. 4.14).

Press the Blank softkey and enter in the interactive screenform Blank the parameters described below.
The name of the blank is to be “Blank1”.

Confirm the name entered and move the cursor to the input field **Select material**.

Open the continuation form in the input field.

The **Material list** selection form appears.

The material of the blank could be ST 50–2.

Confirm the material group for this material.

Move the cursor to the input field **Workpiece ref. plane**.
The Workpiece ref. plane has a sign whereas the Material thickness as depth does not.

The workpiece reference point (see sketch in the Appendix or Fig. 4.17) is at the top edge of the workpiece. Since the top edge is zero, accept the default value Workpiece ref. plane ZM = 0.0.

The programming system programs the geometry in the
- X and Y axes (XY plane) and
- The depth in the Z axis.

![Diagram of Workpiece reference plane and material thickness](image)

Fig. 4.17 Workpiece reference plane and material thickness

Enter the value “35” (mm) as the Material thickness.

Your interactive screenform should now look like this:

![Screenshot of screenform](image)

Fig. 4.18 Parameters for the blank

Set the cursor to the input field global contour transitions.

This field is initialized with "no", which corresponds to the majority of cases.

If you want to define geometries with global contour transitions for oriented and construction geometries, you must select "yes".

Proceed in the same way when editing contours.

Conclude input in the interactive screenform for the first blank parameters.

The Graphic Programming System inserts a new menu in the horizontal softkey row.
Note The current programming level is displayed in the line above the graphic area.

You will now define the rectangular blank contour using the **Standard geometry/rectangle**.
4.3.2 Standard geometry/rectangle

In the Graphic Programming System there are three functions for defining a blank:

- Oriented geometry
- Construction geometry
- Standard geometry

In the following section you will define a rectangle with the function Standard geometry.

**Objective**
Enter all parameters of the standard geometry rectangle.

**Initial situation**
You are in programming level Geometry/New_Create/Blank.

**Procedure**

Press the Standard geometry softkey and then press the Rectangle softkey to define the blank contour.

The interactive screenform Rectangle is offered by the Graphic Programming System.

The parameters Workpiece reference plane and Material thickness have been stored by the system.

If these values are to be changed again, go into the higher-level menu and call up the blank data once again.
Have a look at your sketch and then you can enter all parameters needed for your rectangle.

![Sketch of a rectangle with dimensions 150 (mm) in X and 100 (mm) in Y, with the reference point at the bottom left.]

**Fig. 4.21 Reference point, bottom left**

The cursor is located at the input field **Reference point**.

Open the continuation screenform and a selection menu appears for the reference point of the rectangular contour to be programmed.

Using the cursor control, select the reference point **Bottom left (2)**,

![Selection form for reference point with options for different points like Center point, Bottom left, Top right, etc.]

**Fig. 4.22 Selection form, reference point**

and accept this.

The reference point is entered.

If you are not familiar with some of these parameters, call up a graphic help screen by pressing the help key.

This form describes the parameters by means of a sketch.

Place the cursor on **Length in X**.
Using the cursor keys and the numeric keys, enter the parameters for

**Length in X** = “150” (mm) and

**Length in Y** = “100” (mm), as shown in the next figure.

![Rectangle parameters](image1)

**Fig. 4.23 Blank/rectangle parameters**

Conclude entry in the interactive screenform for the rectangle parameters.

![Blank contour rectangle](image2)

**Fig. 4.24 Blank contour rectangle**

You have now programmed the blank contour.
4.3.3 Views – 4 windows

Objective

While programming, you can display at any time several views of the contour on the screen.

In this example, select the representation of the four standard views on the screen.

Procedure

Press the softkey Views

and select a view.

Press the softkey Activate 4 views.

You now see your blank on the screen in four different views:

- Plan view
- Two side views and
- Isometric perspective.
The vertical softkey menu has returned to its initial level. Every function offered in the horizontal and vertical softkey menus can be activated.
### 4.3.4 Views – plan view

**Objective**
For your example, again select plan view.

**Procedure**
Press the Views softkey then the Plan view softkey, and the following display appears:

![Plan view display](image)

*Fig. 4.27 Blank, plan view*

### 4.3.5 Finished part contour

**Objective**
This section describes how you
- Generate a rectangular pocket with the standard geometry, and
- Zoom a viewport.

The blank contour has been defined. Now program the rectangular pocket indicated by the dotted area in the sketch below.
Initial situation

You are at the programming level *Geometry/New_Create*.

Procedure

Press the *Finished part* softkey.

An interactive screenform appears in which you must define the location of the contour in the view that you have selected, i.e. in the Z plane.

First give the contour the name "Contour1".

Then enter for the *Start plane* the Z value = "–15" (mm) and for the *End plane* the Z value = "–25" (mm).

End interactive screenform input for the contour parameters.
4.3.6 Standard geometry/rectangle

Objective
You will program a rectangular contour (pocket).

Procedure

Press the **Standard geometry** softkey.

Now program the rectangular pocket.
Press the **Rectangle** softkey.

Consider your sketch and enter all necessary parameters for your rectangle.

Note
The angle to the X axis is “150°” or “–30°”. Please enter “–30°” for the angle.
End interactive screenform input for the rectangle parameters. You have now programmed your rectangular pocket.

![Diagram of a rectangular pocket](image)

Fig. 4.33 Contour rectangular pocket

Now operate the function **Save** to avoid losing (e.g. in the event of a power failure) the last contour that you have programmed.

**Note**

The graphic programming system saves the geometry you enter at regular intervals.

The maximum amount of data that will be lost is that which was entered over the previous 3–5 minutes.

If you want to open the file following a loss of data (e.g.: power failure), operate the function **Read** followed by

the function **Read Auto**.

The **Oriented Geometry** function does not have the automatic save feature.

The next section describes how to zoom in on a part of the screen.
4.3.7 Screen – select viewport

**Objective**
An interesting function in the vertical softkey menu is the Set and zoom or reduce viewport function.

**Initial situation**
You are at the programming level Geometry/New_Create.

**Procedure**

Press the **Zoom** softkey.

![Fig. 4.34 Softkey function: Zoom](image)

Then select the **Select viewport** softkey.

![Fig. 4.35 Select viewport](image)

The cursor is automatically positioned on **Zoom**.

**Note**
If you want to reduce the size of the viewport, put the cursor on **Enter reduction factor (2–9)** and enter a factor. In this example, you will practice zooming.

Leave the cursor on **Zoom** and press the **INPUT** key.
A crosshair appears. You can control this crosshair with the cursor keys.

When zooming, you call up a rectangle with the crosshair. The contents of this rectangle is then the zoom. Begin with the first corner point.

Move the crosshair to the first corner of your zoom.

---

**Fig. 4.36  Zoom viewport 1**

**Fig. 4.37  Zoom viewport 2**
and confirm.
The information line guides you when operating.
It requests you to specify the second corner of your zoom.
Move the crosshair to the second corner (see next figure)

Fig. 4.38 Zoom viewport 3

and confirm by pressing the INPUT key.
You now have a new viewport.

Fig. 4.39 Viewport zoomed
Note

The zoom is retained only if you press the Set viewport softkey under the Screen softkey.

For the next exercise, generation of a user-defined geometry, you need the plan view of the complete workpiece.

Procedure

Change the zoomed display back to its initial state in the way that you practised in View/Plan view.

Set the plan view back to the usual size.
4.3.8 Construction geometry

**Objective** Using the construction geometry function, you will
- Create the construction elements point, line and circle
- Edit these elements
- Place a contour on them, and
- Generate the element transitions rounding and chamfer.

You will be working under the menu path **Contour/Construction geometry**.

You will have been able to program the blank contour and rectangular pocket with standardized contours.

**Initial situation** You are at the programming level **Geometry/New_Create**.

![Fig. 4.40 Contour – geometry, any (island)](image)

Next, you will program the contour indicated by dots in the sketch above.

This is a user-defined contour.

Create this contour by means of generated construction geometry elements.
You start the exercise from this representation.

![Geometry representation]

**Procedure**

Press the **Finished part** softkey.

Now refer to your sketch with the dimensions.

![Parameter representation]

**Fig. 4.42 Parameters “Start plane”, “End plane” (island)**

The island of the user-defined contour is given the name “Contour2”. Enter the name and then enter the position of “Contour2” in the Z plane.

The figure below shows the values that are to be entered as well as the name.
Accept the parameters and conclude the interactive screen form.

You must now define your user-defined contour. You can create your contour with the functions:

- Oriented geometry (s. Section 4.3.15, Page 4–62)
- Construction geometry.

Now you can practice creating construction geometry. Please press the **Construction geometry** softkey.
In the line above the graphics area you see in which programing level you are now working. You **cannot** exit this programming level with the **RECALL** key.

You exit this programming level with the **Abort** and **OK** softkeys.

The **RECALL** key has no effect here.

For this contour, you first define the construction geometry elements.
4.3.9 Creating contour construction geometry elements

Objective

You will create the construction geometry elements

- Point
- Line
- Circle.

Initial situation

You are at the programming level Geometry/New_Create/Finished part/Construction geometry.

Procedure

Press the Const. geo. elements softkey.

Construction geometry elements

The programming system offers various construction geometry elements, e.g.:

- Point
- Line
- Circle.

For these elements, there are various design possibilities, and these are described in detail in the User’s Guide “Part 2: Operator/Programming Functions”.

Notes

- The workpiece contour is created by means of the construction geometry elements listed above. You must join these to define a contour.
- Lines are always infinite lines. They have no end limit. Circles are always full circles.
- The sequence in which the construction geometry elements are programmed does not have to be the same as the contour sequence.
- You define the direction of the contour later with Contour definition.
The elements drawn with dotted lines in the following figure are required as construction geometries in order to program your user-defined contour in accordance with the workpiece sketch.

Fig. 4.45 Construction geometry elements

Procedure

**Line 1**

1. Start off with Line 1.
2. Press the Line softkey.
3. A crosshair appears. You are requested to select the first reference elements.
4. You control the movement of the crosshair with the cursor keys.

At the beginning of every construction geometry design, the X and Y axes are available to you in the coordinate system as well as the zero point as reference point.

5. Line 1 is parallel to the X axis at a distance of 20 mm. Thus, you need the X axis as reference element for the design.
6. Move the crosshair to the X axis reference element as shown in the figure below.
You do not have to accurately position the crosshair in order to select lines and circles. The programming system automatically finds the element which is closest to the crosshair.

The coordinates of the crosshair can be seen above the vertical softkey bar.

Confirm selection of the reference element by pressing the **INPUT** key.

You enter the distance by pressing the softkey **Numeric input**.

Enter the **distance** "20" (mm)

in the interactive screenform which now appears on the screen, accept the value and close the interactive screenform.

The programming system calculates two solutions (green), a line above and a line below (which cannot be seen in the viewport) the X axis. You need the line above and therefore you should now select it.

Move the crosshair to the upper line

and confirm the selection.
Line 1 has now been designed.

Fig. 4.47 Construction geometry line 1
4.3.10 Edit construction geometry

Objective
If you have entered an incorrect parameter for the distance of construction line 1,
you can change the value with the function Edit const. geo.

Initial situation
You are at the programming level Geometry/New_Create/Construction geometry/Const. geo. elements.

Procedure

Press the softkey Edit.

Fig. 4.48 Edit programming level

For this exercise, press the softkey Edit parameter.

With the softkey function Edit parameter only the construction geometry elements can be changed that you have designed with a numerical value (e.g. distance, radius).

The system automatically detects that you have only constructed one element and displays the interactive screenform Distance.

Fig. 4.49 “Distance” interactive screenform
The value “20” (mm) can now be changed. For your example, however, this is the correct value and therefore you will accept it and continue with your design.

Press the RECALL key to return to the .../construction geo. elements programming level.

Note
If you have designed several elements, use the cursor keys to select the line you have just constructed, and confirm your selection.

4.3.11 Create contour construction geometry elements

Initial situation
- You are at the programming level Geometry/New_Create/Finished part/Construction geometry/Construction geo. elements.
- Continue programming (see Page 4–44) by designing the remaining construction geometry elements.

Procedure
To design Circle 1 you need Line 1 and Line 2. Lines 1 and 2 are tangential to circle 1.

![Diagram of construction geometry elements]

The parallel reference line for Line 2 this time is the Y axis.

You have now had some practise in operating the graphic programming system. You know the key symbols. Now proceed in the operating sequence described below.
**Line 2**

Press the **Line** softkey.

Select the Y axis as reference element using the crosshair.

Press the **INPUT** key.

Press the **Numeric input** softkey.

Enter **Distance** “85” (mm) as parameter.

Press the **INPUT** key.

Select the required **Line 2** with the crosshair. This is the line to the right of the Y axis in the sketch to the left of the circle.

Press the **INPUT** key.

You have now designed the second construction line.

---

*Fig. 4.51 Construction geometry line 2*
Now design Circle 1 with Construction lines 1 and 2.

From the workpiece sketch you can see that

- Lines 1 and 2 are tangential to circle 1
- The radius is 15 mm.

![Construction geometry elements](image)

**Fig. 4.52 Construction geometry elements**

Now proceed in the sequence described below.

**Circle 1**

Press the **Circle** softkey.

Use the crosshair to select **Line 1** as first reference element.

Press the **INPUT** key.

Use the crosshair to select **Line 2** as second reference element.

Press the **INPUT** key.
Press the **Numeric input** softkey.

![Figure 4.53 Interactive screenform: Radius](image)

The programming system displays the interactive screenform **Radius**.

In the line above the graphics display you can see that you are in the programming level **Geometry/Create_new...Circle**.

Enter the radius value

![...](image)

**Radius** “15” (mm),

and press the **INPUT** key.

There are now four different possibilities for constructing the circle.

You need the circle on the right above the reference line point of intersection.

![Figure 4.54 Select element](image)

Move the crosshair into the proximity of the circle that you require for your construction geometry

and select it with the **INPUT** key.

Your Graphic Programming System has constructed the circle.
You now construct Line 3 on Circle 1.
From the workpiece sketch you can see that

- **Line 3** is tangential to **Circle 1**
- The angle is 45°.

**Line 3**

Press the **Line** softkey.

Using the crosshair select **Circle 1** as first reference element,

press the **INPUT** key.

Press the **Numeric input** softkey.

The system recognizes that
line 3 is tangential to circle 1 under a specific angle.
The interactive screenform **Angle** appears.

Key in

**Angle** “45” (°).
as a parameter for the angle (see workpiece sketch).

**Fig. 4.55 Construction geometry elements**

**Fig. 4.56 Interactive screenform: Angle**
Press the **INPUT** key.

Using the crosshair select **Line 3** above the circle.

Press the **INPUT** key.

**Fig. 4.57 Construction geometry, line 3**

**Line 3** has been constructed.

Next, you will construct **Lines 4, 5 and 6** and also **Point 1**.

**Fig. 4.58 Construction geometry elements**
Please continue in the following order:

**Line 4**

You will now construct line 4 parallel to line 1.

Press the **Line** softkey.

Using the crosshair select **Line 1** as reference element.

Press the **INPUT** key.

Press the **Numeric input** softkey.

Enter **Distance** “45” (mm) as parameter.

Press the **INPUT** key.

Select the required **Line 4** above using the crosshair.

Press the **INPUT** key.

**Point 1**

You will use **Point 1**

- As reference point for **Line 5** and
- As initial and end point for the contour.

Press the **Point** softkey.

Select **Line 4** as first reference element using the crosshair.

Press the **INPUT** key.
Select Line 3 as second reference element using the crosshair.

Press the INPUT key.

Line 5

You will construct line 5 in point 1 as line 3 at a specific angle of 90°.
Press the Line softkey,

Select Point 1 as first reference element using the crosshair. You have to locate exactly on the point with the crosshair.

Press the INPUT key.

Select Line 3 as second reference element using the crosshair.

Press the INPUT key.

Enter Angle “90” (°) as parameter.

Press the INPUT key.

Line 6

Construct line 6 parallel to line 3.
Press the Line softkey,

Select Line 3 as first reference element using the crosshair.
Press the **INPUT** key.

**Numeric input**

Press the **Numeric input** softkey.

Enter **Distance** “40” (mm) as parameter.

**Press the INPUT key.**

Select the required **Line 6** below using the crosshair.

**Press the INPUT key.**

![Construction geometry element](image_url)

**Fig. 4.59 Construction geometry element**

All construction geometry elements have been constructed for the user-defined contour.

In the next programming step, you will define the contour based on these elements.

Press the **RECALL** key to get to the next-higher programming level.
4.3.12 Contour definition

Objective
You will define the contour for the island based on the generated construction geometry elements.

Initial situation

- You are at the programming level Geometry/New/Create/Finished part/Construction geometry.
- You have defined the construction geometry elements for the island (see Fig. 4.59, Page 4–54).

Procedure

Press the Contour definition softkey.

Note

You will program the transitions in the section Element transition below (see Section 4.3.13, Page 4–57).

Note

At this programming level, using the cursor keys to control the crosshair, you will define your contour with the help of the construction elements by pressing the INPUT key.

Fig. 4.60 Contour definition
Every contour begins with an initial point and finishes with an end point.
In the case of closed contours, the initial point must always be identical with the end point.
In the case of open contours, the initial point is not the same as the end point.
You must locate the crosshair exactly on the desired point.
If you select a circle on a line, the circle element closest to the line will be allocated to it.

Your contour is a closed contour (from Point 1 to Point 1). The direction in which you allow your contour to develop is optional. In our exercise, the contour follows the elements
Point 1 – Line 5 – Line 6 – Line 1 – Circle 1 – Line 3 – Point 1.

If you have selected an incorrect element, the contour element last selected can be deleted by means of the CLEAR key.

When contour definition has been completed, leave this level with the OK softkey or abort the function by pressing Abort.
Follow the instructions for crosshair control given in the information line below the graphics area.

Procedure
Proceed as follows:
1. Select Point 1 as initial point of the contour
2. Select Line 5 as contour
3. Select Line 6 as contour
4. Select Line 1 as contour
5. Select Circle 1 as contour
6. Select Line 3 as contour
7. Select Point 1 as end point of the contour
Conclude contour definition by pressing the OK softkey.

Fig. 4.62  User-defined contour (island)

The contour has now been defined.

4.3.13 Element transition

Take a look at your workpiece sketch. Two roundings and one chamfer are still missing in the contour.

_initial situation_ You are at the programming level Geometry/New_Create/Finished part/Construction geometry.

Procedure

Press the Element transition softkey.
1st rounding (20 mm)

Press the Rounding softkey.

Select corner for the first rounding using the crosshair,

and confirm.

Enter “20” (mm) as transition radius,

and confirm.
2nd rounding (5 mm)

Select corner for the second rounding,
and confirm.
Enter "5" (mm),
and confirm.
You have programmed the roundings, now press the RECALL softkey.

Chamfer (5 * 45°)

Press the Chamfer softkey.
Select chamfer corner,
and confirm.
Enter "5" (mm),
and confirm.

Note
You can delete the parameters for the transition elements using the Delete transition softkey.
Exercise

Change from the **Element transition** programming level to the next-higher level.

Press the **RECALL** key twice.

You have completed your user-defined contour.

Press the **OK** softkey.

On quitting the construction geometry, the graphic programming system saves the created geometry as **Auto Save**.

The programming system clears the construction geometry elements from the screen.

If you want to add to the contour or modify it, call up the function **Geometry/Edit**.

An exercise now follows.

Press the **RECALL** key to get into the programming level **Geometry**.
4.3.14 Edit programmed contours

**Objective**

The next section describes how you can, for example, modify a contour that you have created under the function **Construction geometry**.

**Initial situation**

You are at the programming level **Geometry**.

![Fig. 4.64 Edit contours](image)

Press the **Edit** softkey.

All geometry elements that you define with parameters in an interactive screenform can be redefined under the function **Edit**.

Select the programmed user-defined contour.

Accept this contour to change its parameters.

The interactive screenform **Contour** appears. You have access to the parameters.

Conclude the interactive screenform **Contour** by confirming the parameters.
Now you are back at this programming level:

You do not wish to change anything. Press the OK softkey to return to the Geometry programming level.

You will now practise generating this user-defined contour (island) with the function Oriented geometry.

4.3.15 Oriented geometry

**Objective**

The user-defined geometry constructed in the Construction geometry section will now be programmed in this section with the Oriented geometry function.

This section describes how you

- Delete the user-defined geometry programmed under the Construction geometry function, and
- Regenerate it with the Oriented geometry function.

**Initial situation**

You are at the programming level Geometry.
4.3.16 Delete contour

You are now at the next programming level as shown below.

![Diagram showing the delete geometry process]

Fig. 4.66  Delete geometry

**Procedure**

Press the **Delete** softkey.

Select the user-defined contour which is represented by the shaded figure in the sketch (Fig. 4.67).

![Diagram showing user-defined geometry]

Fig. 4.67  Contour – user-defined geometry (island)

Confirm the selection and delete the geometry.
4.3.17 Create contour using oriented geometry

Objective
In the following, you will again create the user-defined geometry with the function **Oriented geometry**.

Initial situation
You are at the programming level **Geometry**. You start off at this programming level.

Procedure
Press the softkey **Create new** and

then press the softkey **Machined part**.

Call the new contour, for example, “Oriented contour”. Enter the parameters for the start and end planes as shown in the illustration below.

Press the **INPUT** key.
Press the softkey *Oriented geometry*.

You will program the transitions later.

![Diagram showing initial point and parameters](image)

**Fig. 4.70 Parameters, initial point**

Take a look at your workpiece sketch and you will be able to calculate from the dimensions the initial point for creating the geometry.

Enter the values

- \(X = 100\) (mm)
- \(Y = 20\) (mm)

in the interactive screenform *initial point cartesian* that appears on the screen.

When defining an element, you choose whether you wish to enter a contour element in Cartesian or polar coordinates in an interactive screen form. Only when you delete the contour element or all the parameters of the coordinate system you have chosen, can you enter a different coordinate system.

and confirm the input.
You see the initial point.

The programming system logs your inputs for the element in the information line beneath the graphics area.

**Sketch**

Take a look at your workpiece sketch.

You begin to program the user-defined contour in clockwise direction.

The first element is the arc in the clockwise direction (given in the sketch with a radius of 15 mm).

Press this softkey function **Arc cw** (cw – clockwise, ccw – counterclockwise).

**Note**

Enter known values into this screenform. The other fields remain blank.

EX means end point X absolute.

You can get explanations for these values by pressing the Help key.

The entries in the interactive screenform **Arc of circle Cartesian** are as shown in the figure below.
Enter in the input fields for the centre point of the arc to be constructed (circle)

for the radius \( RD = "15" \) (mm)

and for the coordinates

\( CX = "100" \) (mm)
\( CY = "35" \) (mm).

Confirm the parameter inputs.

**Sketch**

As the next contour element on your workpiece sketch you can see the user-defined line which is tangential at 45° to the **Arc right** and an end point \( E_Y = "65" \) (mm).

Press the softkey **Line any** and enter the known parameters

for the end point of the element in the Y axis \( E_Y = "65" \) (mm),

for the inclined line (tangential to the circle) \( AS = "45" \) (°) and,

for the transition arc-line \( AT = "0" \) (°)

in the interactive screenform **Line Cartesian**.

When you have put the cursor in an input field for the end point, you will see the following display.

<table>
<thead>
<tr>
<th>Line</th>
</tr>
</thead>
<tbody>
<tr>
<td>End point ( e_X )</td>
</tr>
<tr>
<td>( e_Y )</td>
</tr>
<tr>
<td>Length</td>
</tr>
<tr>
<td>Angle ( AS )</td>
</tr>
<tr>
<td>Transition behaviour</td>
</tr>
<tr>
<td>Angle ( AT )</td>
</tr>
<tr>
<td>Radius ( R )</td>
</tr>
<tr>
<td>Cham.</td>
</tr>
<tr>
<td>Full parameter set</td>
</tr>
</tbody>
</table>

**Fig. 4.73** Help, Line Cartesian

Confirm the entries.
You can see the constructed geometry elements.

- Initial point
- Arc cw
- Line cw

For the next contour element (see sketch), again press the softkey **Line any** and enter the known parameters

For the length of the lines $L = "40"$ (mm),

or the transition line-line $AT = "-90"$ ($^\circ$) und

for the element transition as chamfer $B = "5"$ (mm),

in the interactive screenform **Line Cartesian**.

**Note**

The rounding and the chamfer refer to the transition to the last element.

Confirm the entry.
The figure below shows the design that should also be displayed in the graphics area.

For the next user-defined line, you know three parameters:

- Element transition \( \text{AT} = -90^\circ \),
- Rounding \( \text{R} = 5 \text{ (mm)} \),
- End point \( \text{E}_Y = 20 \text{ (mm)} \).

For the purpose of this exercise, however, you should enter only two parameters in order to see how the programming system behaves for partially defined elements.

Press the softkey Line any and enter the known parameters for the transition line-line \( \text{AT} = -90^\circ \) and for the element transition as rounding \( \text{R} = 5 \text{ (mm)} \), in the interactive screenform Line Cartesian.

Press the INPUT key.

The programming system can not yet offer a solution as the length of the line has not yet been entered.

With the next element, the programming system computes the contour details from this.

The number of programmed elements can be read off in the information line below the graphics area.
Press the softkey **Line left ←** for the last geometry element. The programming system already enters some parameters.

**Note**

Parameters calculated by the system are identified in the input fields. They have the following meaning:

- “=“ direct calculated value
- “+” value added to another
- “−” rounded down value

These marked fields cannot be overwritten.

The input fields can be defined only when you delete the parameter from which the system has calculated these resulting values.

Now add the known parameters for the end point of the last element and the element transition (rounding).

The graphic programming system allows you to enter the geometry elements from a **simple parameter set**.

If, however, you need still more parameters in order to define the geometry element, then use this key to open the interactive screenform containing all the parameters from the **full parameter set** list box.

Enter the end point

\[ E_X = "100" \text{ (mm)} \]
\[ E_Y = "20" \text{ (mm)} \]

and the rounding

\[ R = "20" \text{ (mm)} \]

into the interactive screenform **Line Cartesian**, and confirm the entries.

The programming system computes the contour details. In the figure below you can see the constructed user-defined contour.
When the cursor is positioned on a geometry element, you can:

- Use the **INPUT key** to call up the relevant interactive screenform for the element and edit its parameters.
- Delete the element using the **Delete key**.

End graphic programming of the geometry elements and exit this level with this key.

The island is programmed and therefore you exit this programming with the softkey **OK**.

On quitting the input of oriented geometry, the graphic programming system saves the defined geometry as **Auto Save**.

**Exercise**

Next, you will practise how to graphically program holes.

The programming system defines the holes as geometry definition.
**4.3.18 Holes – hole box**

**Objective**

The section below describes how you create
- Holes as a hole box and
- A single hole.

You will program the four holes shown in the sketch with a diameter of 10 mm.

**Procedure**

**Initial situation**

You are at the programming level **Geometry/New_Create**.

**Press the softkey Holes.**

Enter the parameters which have already been entered in the illustration below.

**Confirm your entries.**

**Press the softkey Hole box to define the location of the four holes that you want.**

![Sketch of holes](image)
The interactive screenform **Hole box** appears. With the help of the drawing below, enter the various parameters from your workpiece sketch into the interactive screenform.

![Diagram of hole box with parameters](image)

**Fig. 4.79** Parameters, start point, 1st side, 2nd side

The figure below already includes the parameters that are to be entered.

![Selection of start points](image)

**Fig. 4.80** Selection of start points, 1st side, 2nd side

After pressing the **INPUT** key, the hole box has been constructed.
You have now constructed the geometry of your workpiece.
4.3.19 Hole as cutter insert point

**Objective**

This section describes how you generate a single hole.

**Initial situation**

You are at the programming level **Geometry/New_Create**.

In this exercise, you will program an insert point (relief hole) for the rectangular pockets (shown by the dotted area in the sketch).

This hole is provided to relieve the face end of the cutter when machining in solid material.

This hole can be created immediately before programming the milling operation.

Press the **Holes** softkey.

Enter the parameters in the interactive screenform **Holes**.

The next figure already contains the parameters that are to be entered.
Confirm the entry.

Press the **Single hole** softkey.

In the “Parameters, insert point 1” drawing (see Fig. 4.82) you will find the values $X = \text{“15” (mm)}$ and $Y = \text{“60” (mm)}$.

Enter the values that you can already see in the following figure (Fig. 4.84).

**Fig. 4.84 Parameters, insert point 3**

Confirm the parameters by pressing the **INPUT** key.

**Fig. 4.85 Single hole**

The single hole has been programmed.

In the next steps of the exercise you will learn how to program the machining functions drilling and milling.

First go into the programming level shown in the next figure.
4.4 Graphic programming of drilling and milling

Steps involved in the exercise, Create user magazine (duration approx. 25 min.)

Now unfold the work schedule in the Appendix (see Section 6.2).

Objective

You will machine the programmed geometry by drilling and milling.

The following section describes how you

- Mill an island
- Select a tool to do this
- Set a construction hole for the pocket
- Mill a pocket
- Machine holes
- Generate part programs, and
- Simulate machining.

Initial situation

- You have created the tools (see work schedule Section 6.2) in user magazine Exercise
- You have designed workpiece “PART 1” (see workpiece sketch Section 6.1)
- You are at the basic level of the Graphic Programming System.
4.4.1 Machining

For the next exercise, you start off with this display:

![Diagram of machining setup]

Program the functions **drilling** and **milling** under the function **machining**.

Press the softkey **Machining**.

You always need tools for machining.

You load **one** magazine for each machining operation.

You can always extend or alter the loaded magazine under the function **Tools Material** (see User’s Guide “Part 2: Operating/Programming Functions”).

The vertical function **Program steps** has a submenu called **edit** that enables you to load a different user magazine or tool master data (file “WKZ”).

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

In the exercise “Tools” (see Section 4.2, Page 4–4) you created a magazine called **Exercise**. First load this tool magazine.

Move the cursor to the input field **Load magazine**.
Open a selection screenform with the INPUT key.

Select the magazine **Exercise** from the list of tool magazines offered,

and load the magazine for the machining operation with this key.

The interactive screenforms with parameters offered in this exercise mostly have default values. An empirical value file exists for this purpose. This makes it easier to enter parameters when programming.

The **Beginning of program** interactive screenform that comes onto the screen (see Fig. 4.87 on the following page) requests you to enter the program type and program No.

These two items of information lead to the program name of the part program.

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

This and other information appears as commentary in your **Part program**.

The entries put into the interactive screenform **Beginning of program** are default values for the calculation which follows in the **Create part prog.** function.
The entries in the interactive screenform could look like this:

<table>
<thead>
<tr>
<th>Beginning of program</th>
</tr>
</thead>
<tbody>
<tr>
<td>Program type: Main program</td>
</tr>
<tr>
<td>Program No.: 1</td>
</tr>
<tr>
<td>Drawing No.: 5/00001302.00</td>
</tr>
<tr>
<td>Date: 30 Sept 1996</td>
</tr>
<tr>
<td>Machine: WMP-M</td>
</tr>
<tr>
<td>Programmer: Anne Cetisch</td>
</tr>
<tr>
<td>Comment:</td>
</tr>
<tr>
<td>20 rotary table centre-work, 2P</td>
</tr>
<tr>
<td>1st axis: 0.0000</td>
</tr>
<tr>
<td>2nd axis: 0.0000</td>
</tr>
<tr>
<td>Mounting position (angle) of the side parallel to the 1st axis in the workpiece coordinate system 0.0000°</td>
</tr>
</tbody>
</table>

Fig. 4.87 Parameters: Beginning of program

Enter the required parameters in the usual way.
Accept these parameters.
4.4.2 Machining, milling

Objective
The following exercise steps describe how you

- Initially mill the island from the blank,
- Select a milling cutter for this, and
- Start milling with the function Solid milling.

You require the following milling tools

- For the island, the 20 mm end mill,
- For the pocket as a relief hole, the 10 mm solid drill and the 5 mm end mill.

4.4.3 Tool selection

Your workpiece contains drilling and milling operations. Programming can always be done in any sequence. Start with the milling.

Press the Milling softkey.

![Milling programming level](image)

Press the Tool selection softkey.

The programming system offers you the choice of tools that you created in the tool magazine Exercise.

You want to be able to choose from all the existing milling tools. Select the All function.
There are two tools:

![Image of tool class]

To mill the island, select the **SAF 20/32/R HM P20–P35** end mill.

In the interactive screenform that is now on the screen, you will see the tool description that you have programmed.

![Image of tool description]

**Note**

At this point you can again check the values for the tool and offset memory number and alter these to suit the machine.

Accept the end mill.
4.4.4 Solid milling of an island

The solid milling function machines islands and pockets. The un-machined contours can be entered in any sequence.

You will first machine the area indicated by dots in the sketch.

![Fig. 4.91 Blank contour and island](image)

Press the **Solid milling** softkey.

**Note**

The programming system guides you in the information line while you are programming.

Now select the blank contour and accept it.

Select the island (any contour) and accept it.

When selecting contours, you must always include the island(s) that must be considered.

End selection for the area by pressing the **INPUT** key.

Confirm.

You do not have to modify the parameters in the interactive screenform **Solid milling** that is shown on the screen. The default values in the input fields are contained in an empirical values file.
Fig. 4.92  Solid milling interactive screenform

The start and end plane of this area have been determined from the geometry and entered by the system.

Confirm.

The programming system now automatically calculates the cutter path and the cut segmentation for the programmed geometry definition.

Note

The programming system always recognizes that the tool has freedom to move outside of the area to be milled.

The milling cutter infeeds outside of the blank contour.
4.4.5 Solid milling of a pocket

Objective

The section below describes how you

- Place a relief hole for the end mill, and
- Machine a rectangular pocket with solid milling.

In the “Geometry” section you programmed a relief hole for the milling tool.

Before executing the function “Solid milling”, you will first drill this cutter insert hole.

Press the RECALL key to go to programming level Machining.

Relief hole

Press one after the other the softkeys Drill,

Centre drill and

Tool selection.

Select the All function.

Select the solid drill VOB 10/42/R VHM.

Open the tool description for the solid drill.

If you have no changes to make, confirm this tool.

To execute the drill function, press the softkey Value input.

Select the single hole in the rectangular pocket.

End selection.
The interactive screenform **Drilling/centering L81** that then appears on the screen contains parameters taken from the empirical values file.

Move the cursor to the input field **Include tip length in calcul.** and open a selection form.

Choose **No (1)** and confirm the selection.

"0" has been entered for the programmed depth. This means that the system determines the depth of the hole from the geometry (the default depth of the hole is 10 mm).

Confirm the screenform parameters.

The system works out the drill path.

To execute the function "Solid milling", go into the programming level **Machining** and press the softkey **Milling**.

Change the tool.

Select the **All** function.

For this pocket, select the end mill **SAF 5/ 16/R HM P20–P35**.

Confirm the screenform parameters.
The next contour is the rectangular pocket of your workpiece (indicated by dots in the sketch).

Fig. 4.94 Pocket and insert hole

Press the softkey **Solid milling** once again.

Select the rectangular pocket and accept it.

Select the insert hole for the milling cutter and confirm it.

As there is no other contour to machine, end the selection.

You do not need to change the parameters (from the empirical values file) in the interactive screenform **Solid milling** that appears on the screen.

The programming system obtains the start and end planes from the geometry parameters.

Confirm this.

The programming system now automatically calculates for the programmed geometry definition the cutter path and the cut segmentation for this pocket.

You machine the contour start from the insert point.

When calculation has been completed, take a look at your workpiece in the **4-window view** (see Section 4.3.3, Page 4–27) and continue programming in this view.
You have now programmed machining by solid milling for your exercise.
4.4.6 Machining, drilling

**Objective**

Finally, you will program machining of the holes indicated by dots in the sketch.

![Fig. 4.96 Holes](image)

When creating the geometry, the holes are stored in the form of a point box. The programming system identifies this relationship and machines the holes in one operation.

Press the **RECALL** key to get into programming level **Machining**

and press the softkey **Drill**.

The following display appears.

![Machining/Drill](image)
Process the drilling matrix with “Drilling cycle, drilling”.
Press the softkey Centre drill,

and select via the Tool selection softkey.

Select the All function.
The programming system accesses the tool magazine Exercise and displays the programmed tool.
The holes have a diameter of 10 mm. The drill offered with its description:
“SPB 10.0/31/118/R HSS” is correct.

Select the twist drill and confirm the choice of tool and then

the tool description.

To select the holes to be machined, press the softkey Value input.
The information line requests you to select the contour, i.e. the holes.
Select the four holes that have been programmed together and end selection.
The island has a height of 15 mm. The drill path passes close to the island.

To prevent a the drill colliding with the island, you must adjust the withdrawal distance of the drill to suit the given conditions.

Place the cursor on the input field **Approach distance DS**, and open a selection form.

Select **Withdrawal distance CR**, and end selection.

Put the cursor on the input field **Withdrawal distance CR** and,

enter “18” (mm).
For the drilling depth, it is advisable to program a depth allowance (AD).

Place the cursor on the input field **Input drill depth direct (1)**

and open a selection form.

Select **Depth allowance (2)**

and confirm the selection.

You can see that the line **Prog. depth L1+** now includes the additional comment **allowance AD**.

Move the cursor to this input field **Prog. depth L1 + allowance AD**.

and enter “1” (mm) for the allowance.

Move the cursor to **Include tip length** and toggle “Yes (2)”. 
The programming system works out the travel movements and displays them in the graphics area.

You see construction lines.

*Note* You see the paths in the front view very clearly.

*Fig. 4.101 Travel paths, front view*
4.4.7 Machining simulation

Objective

The following section describes how you graphically simulate the programmed machining operations.

Initial situation

- You are in workpiece directory “PART 1” (see workpiece sketch Section 6.1).
- You have completed programming of the workpiece (Geometry and Machining).

Press the RECALL key twice to get into programming level Machining.

Select Top view for this function.

For simulation, press the softkey Machining simulation.

Accept the parameters.
The functions on the horizontal softkey menu affect the display on the screen during simulation (see Fig. 4.103).

Abort simulation by pressing the **RECALL** key.
4.4.8 Create part program

Objective
This section describes how you
- Create a part program with the graphic programming system, and
- Edit a part program.

Initial situation
- You are in workpiece directory “PART 1” (see workpiece sketch Section 6.1).
- You have completed programming of the workpiece (Geometry and Machining).
- You are at the basic level.

To obtain an executable DIN program, press the softkey Create part prog.

Accept the parameters in the screenform Main program end that appears on the screen.

The programming system creates the part program for your example.

Note
During graphic programming, you have programmed in the XY plane and the depth in the Z axis.
The part program that has been created has the following contents:

```
(MPF1)
(Drawing No.: 5700801302.00)
(Create date: 28. Sept. 1995)
(Machine: WOP)
(Programmer: Anne Cetlieh)
(Exercise program)
N10 G26 S6000
(Approach reference point)
N15 G00 G53 X250
N20 G53 Y250
N25 G53 Z250
N30 G54 G90
N35 G00 B0
(Machining, top)
N40 G59 X0 Z0 Y0
(Station no. 23)
N45 (SAF 20/32/R HM P20–P35)
N50 D0 M05 M09
N55 G53 Z200
N60 G53 X140 Y110
N65 T23 M06
N70 G54 D23
N75 T30
(Solid milling blank oriented contour)
N80 G58 X0 Z0
N85 G94 F214.859 S1432 M03 M08
N90 G00 X–19 Y0
N95 Z10
N100 G64 Z5
N105 G01 Z–15 F214.8
N110 X0 F429.6
N115 Y100
N120 X150
N125 Y0
N130 X0
N135 X14 Y14
N140 Y86
N145 X136
N150 Y71.728
N155 X140 Y62.071
N160 X136 Y71.728
N165 X129.293 Y78.435
N170 G03 X112.322 Y85.464 I–16.971 J–16.971
N175 G01 X105.251
N180 G03 X88.281 Y78.435 I0 J–24
N185 G01 X72.423 Y62.577
N190 G03 X67.137 Y14 I27.577 J–27.577
N195 G01 X75.505 Y10
N200 X67.137 Y14
N205 X14
N210 X27.6 Y27.6
N215 Y72.4
N220 X63.012
N225 X71.703 Y76
N230 X63.012 Y72.4
N235 X62.806 Y72.194
N240 G03 X47.923 Y27.6 I37.194 J–37.194
```
N245 G01 X27.6
N250 X39.6 Y57.913
N255 Y60.4
N260 X40.603
N265 G03 X39.6 Y57.913 I59.397 J–25.4
N270 G01 X37.6 Y40.001
N275 X39.6 Y57.913
N280 G00 Z8
N285 X150 Y20
N290 Z5
N295 G01 Z–15 F214.8
N300 X139.567 Y10.4 F429.6
N305 G03 X139.6 Y10.431 I–27.495 J29.6
N310 G01 Y10.4
N315 X139.567
N320 G00 Z8
N325 X112.5 Y0
N330 Z5
N335 G01 Z–15 F214.8
N340 X100 Y10 F429.6
N345 G02 X82.322 Y52.678 I0 J25
N350 G01 X98.18 Y68.536
N355 G02 X105.251 Y71.464 I7.071 J–7.071
N360 G01 X112.322
N365 G02 X119.393 Y68.536 I0 J–10
N370 G01 X140.607 Y47.322
N380 G01 X133.284 Y18.787
N385 G02 X112.071 Y10 I–21.213 J21.213
N390 G01 X100
N395 Z8 F214.8
N400 G00 Z10
N405 G60
( Station no. 30 )
N410 ( VOB 10/42/R VHM )
N415 D0 M05 M09
N420 G53 Z200
N425 G53 X140 Y110
:430 T30 M06
N435 G54 D30
N440 T20
(Centre/drill insert hole )
N445 G58 X0 Z0
N450 G95 F0.09 S3183 M03 M08
N455 G00 X15 Y60
N460 Z–5
N465 G81 R2=–10 R3=–25 R10=–7
N470 G80
N475 Z–5
( Station no. 20 )
N480 ( SAF 5/16/R HM P20–P35 )
N485 D0 M05 M09
N490 G53 Z200
N495 G53 X140 Y110
:500 T20 M06
N505 G54 D20
N510 T13
(Solid milling contour1 insert hole )
N515 G58 X0 Z0
N520 G94 F51.662 S5729 M04 M08
N525 G00 X15 Y60
N530 Z–5
N535 G64 Z–10
N540 G01 Z–25 F51.61
N545 X9.599 Y57.345 F1031.22
N550 G02 X27.099 Y87.655 I2.165 J–1.25
N555 G01 X70.401 Y62.655
N560 G02 X71.316 Y59.24 I–1.25 J–2.165
N565 G01 X56.316 Y33.26
N570 G02 X52.901 Y32.345 I–2.165 J1.25
N575 G01 X9.599 Y57.345
N580 G02 X12.215 Y59.876
N585 G01 X26.215 Y84.124
N600 X67.785 Y60.124
N605 X53.785 Y35.876
N610 X12.215 Y59.876
N615 X16.996 Y61.157
N620 X27.496 Y79.343
N625 X63.004 Y58.843
N630 X52.504 Y40.657
N635 X16.996 Y61.157
N640 X21.778 Y62.438
N645 X28.778 Y74.562
N650 X58.222 Y57.562
N655 X51.222 Y45.438
N660 X21.778 Y62.438
N665 X26.559 Y63.719
N670 X30.059 Y69.781
N675 X53.441 Y56.281
N680 X49.941 Y50.219
N685 X26.559 Y63.719
N690 X31.34 Y65
N695 X48.66 Y55
N700 X31.34 Y65
N705 Z–7 F515.61
N710 G00 Z–5
N715 G60
( Station no. 13 )
N720 ( SPB 10.0/31/118/R HSS )
N725 D0 M05 M09
N730 G53 Z200
N735 G53 X140 Y110
.:740 T13 M06
N745 G54 D13
( Centre/drill hole 1 )
N750 G58 X0 Z0
N755 G95 F0.16 S954 M03 M08
N760 G00 X15 Y10
N765 Z–10
N770 G81 R3—–39.004 R10=3
N775 X135
N780 Y90
N785 X15
N790 G80
N795 D0 M05 M09
N800 G53 Z200
N805 G53 X140 Y110
N810 M30 (End of program)
4.4.9 Exit programm. WOP function

Objective
You will be provided with information for saving the files which you generated with the Graphic Programming System.

The operations for exiting the Graphic Programming System are described step by step in the section “Programming Examples” of the corresponding Environment Description (e.g.: “Graphic Programming System on PC”).

Workpiece geometry and machining
If you change to a new workpiece, the programming system enquires whether you wish to save the workpiece geometry and machining that you have created.

Apart from that, with the vertical softkey Save you can save your workpiece geometry and machining at the time of Graphic Programming.

The Graphic Programming System stores the workpiece geometry file in the workpiece directory where you cannot see it.

Note
In the Graphic Programming System, you are always working under workpiece names (e.g. “PART1”)

Part program
After ending the function Create part prog., the Graphic Programming System stores the part program in your workpiece directory (e.g. “PART1”) it is not necessary to save the program separately.
5 Programming Example 2

Programming Example 2 is an example (see sketch in section 6.3) involving the definition of surfaces and multi-side machining.

5.1 Geometry

Steps involved in the exercise create blank geometry and surfaces (duration approx. 45 mins.)

Objective The following sections describe how to
- program the blank contour graphically and
- then create holes and surfaces.

Initial situation You have created a new workpiece directory called “PART2”. The Graphic Programming System is loaded. You start from this user interface:

Some geometry must be present before you select the Machining softkey.
If you need some new tools, we recommend you create the tools before creating your workpiece geometry. You learned how to do this in the first programming example.
You can use the existing master data magazine as the tool magazine.

1) Only if multi-side machining option is available
First create the workpiece geometry.

5.1.1 Blank contour

Objective

This section describes how you graphically program the blank contour of the workpiece “PART 2” as a standard contour.

Initial situation

- You are in workpiece directory “PART2”.
- The graphic programming system is active.

Now unfold the workpiece sketch in the Appendix (see Section 6.3).

Begin your graphic programming of the blank contour by pressing the Geometry softkey in the horizontal softkey menu.

Then press the softkey Create new. The following display appears.

The plane that is to be programmed with WOP is the default milling plane used by the system.

This can be seen in the top right of the graphics area in Fig. 5.2.
Press the **Blank** softkey and enter the parameters described below in the interactive screenform **Blank**.

In Fig. 5.3, the data for the blank has already been entered.

![Blank parameters](image)

*Fig. 5.3  Data for the blank*

The name of the blank is to be “Dice”.

Confirm the name entered and move the cursor to the input field **Select material**.

Open the continuation form in the input field. The **Material list** selection form appears.

![Material list](image)

*Fig. 5.4  Material list*

The material of the blank could be ST 50–2.

Confirm the material group for this material.

Move the cursor to the input field **Workpiece ref. plane**.

The workpiece reference point (see sketch in the Appendix or Fig. 5.5) is at the top edge of the workpiece. Since the top edge is zero, accept the default value **Workpiece ref. plane ZM = 0.0**.

The programming system programs the geometry in the

- **X** and **Y** axes (XY plane) and
- the depth in the **Z** axis.
Enter the value “40” (mm) as the Material thickness.

Conclude input in the interactive screenform for the first blank parameters.

The Graphic Programming System inserts a new menu in the horizontal softkey row.

Display line for the menu level (programming level)

Note The current programming level is displayed in the line above the graphic area.

You will now define the rectangular blank contour using the Standard geometry softkey.
Press the **Rectangle** softkey.

The interactive screenform **Rectangle** is offered by the Graphic Programming System.

Have a look at your sketch and then you can enter all parameters needed for your rectangle.

The **Reference point** for the blank is at the bottom left (2).

Use this key to select **Reference point bottom left (2)** from the list box.

Using the cursor keys and the numeric keys, enter the parameters for our example

- **Length in X (INC) DX** = "40" (mm) and
- **Length in Y (INC) DY** = "40" (mm).

In the following interactive screenform, the parameters are entered in the individual input fields.

![Rectangle interactive screenform](image)

Fig. 5.7 **Rectangle** interactive screenform

Once you have entered parameters in the input fields, terminate the input of the interactive screenform for the rectangle parameters using the **INPUT key**.

![Blank contour](image)

Fig. 5.8 **Blank contour**

You have now programmed the blank contour.
5.1.2 Hole box at milling level

Objective
This section describes how you create a hole box at the milling level.

Initial situation
- You are at the programming level Geometry/New_Create.
- The blank contour is created (see Fig. 5.8).

Now program the hole box shown in grey in the following sketch.

![Hole box diagram](image)

Fig. 5.9 Hole box

Press the softkey Holes.

First, give the holes the name “Hole box”.

... ... ...
Use the cursor keys and the numeric keys to enter the parameters for the first hole box.

![Diagram of a hole box with parameters](image)

**Fig. 5.10**

The figure below already includes the parameters that are to be entered.

<table>
<thead>
<tr>
<th>Holes</th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Menu:</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Holebox</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Start plane</td>
<td>PS</td>
<td>0.0000</td>
</tr>
<tr>
<td>Depth</td>
<td>L1</td>
<td>20.0000</td>
</tr>
<tr>
<td>Diameter</td>
<td>D1</td>
<td>5.0000</td>
</tr>
</tbody>
</table>

**Fig. 5.11 Parameters, holes**

Confirm by pressing the **INPUT** key.

Press the softkey **Hole box** to define the location of the four holes that you want.

![Diagram of hole box with start and side points](image)

**Fig. 5.12 Parameters, start point, 1st side, 2nd side**
The figure below already includes the parameters that are to be entered.

![Point box diagram]

**Fig. 5.13** Selection of start point, 1st side, 2nd side

Use the cursor keys and the numeric keys to enter the hole box parameters.

After pressing the **INPUT** key, the hole box has been constructed.

![Geometry/Create_new screen]

**Fig. 5.14** Hole box

“Hole” exercise finished

**Note**

Select the function **Save** to prevent any possible loss of the most recently programmed contour (e.g.: as the result of a power failure).

Go to the **Geometry** programming level to define the plane “front” with a single hole in the next section.
5.1.3 Create surface – “front”

**Objective**  
The following section describes how
- To create a side as a “front” surface and
- Make individual holes on the “front” surface.

**Initial situation**  
- You are at the programming level **Geometry**.
- The **milling level** is activated.

Surfaces on the blank or finished part that are perpendicular to the design plane (see Fig. 5.15) can always be created with "Surface" from Geometry.

Fig. 5.15  Finished part with surfaces

To define the surface “front”, press the softkey **Create surface**.

**Note**  
The Graphic Programming System provides three ways of creating a surface:
- **Surface from geometry** and
- **Lateral surface** and
- **Free definition**  
If the surface is to be a circle, you must define it again under **lateral surface**.  
As the surface corresponds to a straight geometry element, create the surface from the geometry.  
With free definition, a surface can be defined in the space as desired.
Select the surface to be machined from the blank geometry.

Select **Surface from Geometry**

![Diagram showing selection of surface from geometry]

**Fig. 5.16** "Front"

Use these cursor keys to select the side (surface) of the blank geometry indicated in the sketch (Fig. 5.16).

Make a single hole on this side.

The contour of the selected surface is shown in **red**.

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

![Screenform for plane definition]

**Fig. 5.17** Plane definition

Enter the name “Front” in the input fields of the interactive screenform **Plane Definition**.

After entering the name, close the interactive screenform by pressing the **INPUT key**.
The Graphic Programming System shows a top view of the surface created on the front side.

**Note**

If you are not programming at the milling level, the Graphic Programming System creates a new coordinate system for this surface.

In the following exercise steps you will create a single hole.

**Objective**

This section describes how to make a single hole (shown in grey in Fig. 5.19).
Create New

Holes

Fig. 5.20 Parameters for single hole

Confirm by pressing the INPUT key.

Single hole

The input parameters are (see Fig. 5.19):

\[
\begin{align*}
X &= 20 \text{ (mm)} \\
Y &= 8 \text{ (mm) (own coordinate system)}
\end{align*}
\]

Confirm the parameters by pressing the INPUT key.

Continue graphic programming using the function **Create New**.

Press the softkey **Holes**.

Enter the parameters (see sketch in Appendix) in the interactive screenform **Holes**.

The figure below already includes the parameters that are to be entered.

Press the softkey **single hole**.

The input parameters are (see Fig. 5.19):

\[
\begin{align*}
X &= 20 \text{ (mm)} \\
Y &= 8 \text{ (mm) (own coordinate system)}
\end{align*}
\]

Confirm the parameters by pressing the INPUT key.

Fig. 5.21 Single hole
The single hole is programmed.
Next, you will make another single hole in the surface “Right”.
To do this, first enter the programming level **Geometry**.

### 5.1.4 Create surface – “right”

**Objective**

The steps in the exercise in the following section describe how to
- Create a side as surface “right” and
- Make a single hole in this surface.

**Initial situation**

- You are at the programming level **Geometry**.
- The blank contour for the milling part is created.

Select the function **Select surface** and

Select the **Milling level**.

To create the surface “right”, press the softkey **Create surface**.

Select **Surface from Geometry**.

Use these cursor keys to select the surface “right” indicated in the sketch (Fig. 5.16).
The contour of the selected surface is shown in **red**.
Confirm your selection by pressing the **INPUT key**.

Enter the name “Right” in the input fields of the interactive screenform **Plane Definition**.

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

![Graphical representation of plane definition](image)

*Fig. 5.23 “Right” side as a created surface*

The Graphic Programming System shows a top view of the surface created on the right.
Single hole

Objective

This section describes how to make a single hole on the surface “right” (shown in grey in Fig. 5.24).

![Fig. 5.24 Single hole “right”](image)

Continue graphic programming using the function Create New.

Press the softkey Holes.

Enter the parameters (see sketch in Appendix) in the interactive screenform Holes.

The figure below already includes the parameters that are to be entered.

![Fig. 5.25 Parameters for single hole “right”](image)

Confirm the inputs.

Press the softkey Single hole.
The input parameters are
\[ X_P = "20" \text{ (mm)} \]
\[ Y_P = "15" \text{ (mm)} \]
Confirm the parameters by pressing the **INPUT** key.

![Diagram of a single hole](image)

*Fig. 5.26 Single hole*

The single hole is programmed.

**Exercise finished**

**Note**

Now select the function **Save**.

The surfaces created do not relate to the finished part and must be redefined if the finished part is modified.

The next section covers how to program the **machining** of Programming Example 2.

Enter the programming level **Geometry** of the Graphic Programming Systems by pressing the **RECALL key**.
5.1.5 Machining

In the following section you will program the machining of the milling part “Programming Example 2”.

Objective

In accordance with the work schedule in section 6.4, page 6–7, you will program

- The hole matrix ("milling plane"),
- The single hole “front” and
- The single hole “right”

in the above sequence.

Initial situation

You have defined the milling level, the surfaces “front” and "right" and created the geometry for Programming Example 2 (see sketches in section page ).

- The milling level is selected
- You are at the basic level of the Graphic Programming System and
- The Graphic Programming System is displaying a top view of the workpiece (see Fig. 5.14, page 5–8).

Note

The individual key functions are not shown in this section.

The work steps are shown in tabular form (see work schedule).

The operations for the Drilling and Milling function and illustrations of the keys are described in Programming Example 1.

After selecting the Select Surface function,

- Load the tool master data,
- Fill in the interactive screenform Program start,
- Enter the coordinates for the point the tool is to approach before the level change in the interactive screenform Change of Level and
- Select the Milling level from the menu.
After pressing the softkey **Display** and reducing the view, the Graphic Programming System displays the workpiece in the graphics area as follows.

![Diagram of workpiece](image)

*Fig. 5.27 Programming Example 2 All levels on*
<table>
<thead>
<tr>
<th>Work Step</th>
<th>Key Function</th>
<th>Instructions</th>
</tr>
</thead>
<tbody>
<tr>
<td>Hole matrix (1)</td>
<td>Machining</td>
<td>Press the softkey <strong>Machining</strong>.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Select the <strong>Twist Drill SPB 5.0/20/118R HSS</strong> tool and then execute the <strong>Enter Values</strong> function:</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• Select the hole matrix</td>
</tr>
<tr>
<td></td>
<td></td>
<td>The Graphic Programming System calculates the clamping method.</td>
</tr>
<tr>
<td>Surface “front” (2)</td>
<td></td>
<td>Select created surface “front” from the menu.</td>
</tr>
<tr>
<td></td>
<td></td>
<td><strong>Note:</strong></td>
</tr>
<tr>
<td></td>
<td></td>
<td>The interactive screenform Change of level describes the tool change point</td>
</tr>
<tr>
<td></td>
<td></td>
<td>for this level.</td>
</tr>
<tr>
<td>Surface “right” (3)</td>
<td></td>
<td>Select created surface “right” from the menu.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>You have already selected the tool in work step 2.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Execute the <strong>Enter Values</strong> function</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• Select the individual hole.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>The Graphic Programming System calculates the path.</td>
</tr>
<tr>
<td>Machining simulation</td>
<td></td>
<td>For the <strong>machining simulation</strong>, select <strong>milling level</strong> from the menu.</td>
</tr>
<tr>
<td>Work Step</td>
<td>Key Function</td>
<td>Instructions</td>
</tr>
<tr>
<td>-----------</td>
<td>--------------------</td>
<td>-----------------------------------------------------------------------------</td>
</tr>
</tbody>
</table>
| Machining simulation | At the basic level | If you want to simulate all the programmed machining steps, press the vertical softkeys 
**Show/All Levels On/Off** followed by 
**Views/DIN Isometrics**. |
|           | Show               | in sequence. The **Machining Simulation** function enables you to follow all the work steps on screen. |
|           | DIN Isometrics     |                                                                             |
|           | Machining simulation |                                                                             |
| Create part program | At the basic level | To create an executable DIN program, press the softkey **Create Part Program** |
|           | Create part program |                                                                             |

The exercise for Programming Example 2 is now complete.

Press the softkey **Save** to prevent loss of data (e.g.: in the event of a power failure).
6 Appendix

6.1 Workpiece sketch example 1

Front view

Top view

Isometric view
6.2 Work schedule example 1

1. Solid milling of an island
2. Relief drilling
3. Solid milling of a pocket
4. Drilling the hole matrix
6.3 Workpiece sketch example 2
6.4 Work schedule example 2

1. Drill hole matrix milling level
2. Drill single hole surface “front”
3. Drill single hole surface “right”
7 Index

B
Blank, 4–19, 5–2

C
Centre drill, 4–90
Chamfer, 4–59
Change of level, 5–19
Circle, 4–48
Construction geometry, 4–37
   contour definition, 4–55
   edit, 4–45
   elements point, line, circle, 4–41
Contour
   delete, 4–63
   edit, 4–61
Contour definition, 4–55
Create part prog., 4–79
Create plane, 5–13
Create surface, 5–9

D
Drilling, 4–77, 4–89

E
Edit
   construction geometry, 4–45
   contour, 4–61
Element transition, 4–57
End plane, 4–30
Exercise Objectives, 3–1

F
Finished part contour, 4–29

G
Geometry, 4–19, 5–1
Geometry, from, 5–10
Graphic programming, 4–2

H
Help function, 2–1
Hole box, 4–72, 4–73
Hole box at milling level, 5–6
Holes, 4–72
   hole box, 4–72, 4–73
   single point, 4–75

L
Line, 4–42

M
Machining, 4–2, 4–78, 5–1
   simulation, 4–94
Magazine
   tool master data, 4–78
   user magazine, 4–78
   WKZ, 4–78
Master data, 4–4
   copy, 4–15
   save, 4–14
Masterdata, load, 4–5
Masterdata/magazine, 4–5
Material list, 4–21, 5–3
Material thickness, 4–22, 5–4
Milling, 4–77, 4–81

O
Oriented geometry, 4–62
   create, 4–64
   input fields, 4–70

P
Part program, create, 4–96
Plan view, 4–29
Point, 4–52
Program, info, 4–15
Programming Example 1, 4–1
Programming Example 2, 5–1
Programming Example 2 (with C axis), machining, 5–17
Programming WOP function, exit, 4–101

R
Rectangle, 4–31
S
Screen, 4–33
   select viewport, 4–33
Simulation, 4–94
Single hole, 5–11, 5–15
Solid milling
   island, 4–83
   pocket, 4–85
Standard geometry, 4–23, 5–4
   rectangle, 4–24
Start plane, 4–30

T
Tool, 4–5
   change, 4–12
Tool data, 4–7
Tool master data, 4–78
Tool selection, 4–81
Tools, 4–4
   change, 4–12
   create, 4–4
Tools material, 4–4

U
User magazine, 4–78
   create new, 4–15

V
Value input, 4–90
Viewport, select, 4–33
Views, 4–27, 4–29
   activate 4 views, 4–27
   plan view, 4–29

W
Work schedule, 6–3, 6–7
Workpiece sketch, 6–1, 6–5

Z
Zoom, 4–33