SIEMENS

SINUMERIK 802S/C

Manual Machine

Operator's Guide

Valid for

Software	*Version*
SINUMERIK 802S/C	3.1.0 or higher

04/2005 Edition

SINUMERIK® Documentation

Printing history

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

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

Status code in the "Remarks" column:

A .... New documentation.
B .... Unrevised reprint with new order number
C .... Revised edition with new status

Edition	Order No.	Remarks
04/2005	---	A

Trademarks
SIMATIC®, SIMATIC HMI®, SIMATIC NET®, SIROTEC®, SINUMERIK® and SIMODRIVE® are registered trademarks of Siemens AG. Other names used in this publication may be trademarks, which, if used by third parties for their own means, may violate the rights of their owners.

Additional information is available in the Internet at:
http://www.siemens.com/motioncontrol

This publication has been produced using WinWord V 8.0, Designer V 7.0 and the DocuTool AutWinDoc.

Other functions not described in this documentation may 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 the contents of this manual for agreement with the hardware and software described. Nevertheless, differences might exist and we cannot, therefore, guarantee that they are completely identical. However, the data in this manual are reviewed regularly and any necessary corrections included in subsequent editions. Suggestions for improvement are welcome.

Technical data subject to change.

VICPAS Aktiengesellschaft
HMI Parts Center

Preface

SINUMERIK documentation

The SINUMERIK documentation is organized in 3 parts:

- General documentation

- User documentation

- Manufacturer/Service documentation

Please contact your local Siemens office for more detailed information about other SINUMERIK 802D publications.

An overview of publications that is updated monthly is provided in a number of languages in the Internet at:
http://www.siemens.com/motioncontrol
Follow menu items "Support" → "Technical Documentation" → "Overview of Documents".

The Internet version of the DOConCD (DOConWEB) is available at:
http://www.automation.siemens.com/doconweb

Target readership

This Manual is intended for machine-tool users. This publication provides detailed information that the user requires for operating the SINUMERIK 810D and 840D controls.

Standard scope

This Operator's Guide describes the functionality of the standard scope.
Extensions or changes made by the machine tool manufacturer are documented by the machine tool manufacturer.
Other functions not described in this documentation may be executable in the control. This does not, however, represent an obligation to supply such functions with a new module or when servicing.

VICPAS
HMI Parts Center

Hotline

If you have any questions, please get in touch with our Hotline:
A&D Technical Support
Phone.: +49 (0) 180 / 5050 - 222
Fax: +49 (0) 180 / 5050 - 223
E-mail: mailto:adsupport@siemens.com
Internet: http://www.siemens.com/automation/support-request

If you have any queries (suggestions, corrections) in relation to this documentation,
please fax or e-mail us:
Fax: +49 (0) 9131 / 98 - 63315
E-mail: mailto:motioncontrol.docu@siemens.com

Fax form: See the reply form at the end of the document.

SINUMERIK Internet address

http://www.siemens.com/motioncontrol

Safety notes

This Manual contains information which you should carefully observe to ensure
your own personal safety and the prevention of material damage. The notices are
highlighted by a warning triangle The warnings are shown below in decreasing
order of danger.

Danger

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

Warning

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

Caution

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

Caution

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

Notice

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

If several hazards of different degrees occur, the hazard with the highest degree must always be given preference. If a warning note with a warning triangle warns of personal injury, the same warning note can also contain a warning of material damage.

Qualified personnel

Start-up and operation of the device/equipment/system in question must only be performed using this documentation. The device/system must only be commissioned and operated by **qualified personnel**. Qualified personnel as referred to in the safety guidelines in this documentation are those who are authorized to start up, earth and label units, systems and circuits in accordance with the relevant safety standards.

Proper use

Note the following:

Warning

The device must only be used for the applications described in the Catalog and only in combination with the equipment, components and devices of other manufacturers as far as this is recommended or permitted by Siemens. It is assumed that this product be transported, stored and installed as intended and maintained and operated with care to ensure that the product functions correctly and properly.

Other information

! **Important**

This notice indicates important facts that must be taken into consideration.

Note

This symbol always appears in this documentation where further, explanatory information is provided.

Machine manufacturer

This pictorial symbol always appears in this document to indicate that the machine manufacturer can affect or modify the function described. Note the machine manufacturer's instructions.

■

vi SINUMERIK 802S/C BA_MM (BA) – 04/2005 Edition

Contents

VICPAS
HMI Parts Center

1

1 Overview of Controls

1.1 Operator panel

Fig. 1-1: Operator panel

1.1.1 Key functions

◀ ▲ ▼ ▶   Move cursor in the direction of the arrow

⬦   Enter key (accept entry values)

←   Delete character (from right to left)

⊖   Acknowledge alarm

>   Extended menu (only if the corresponding symbol appears
                 on the screen)

∧   Back key
                 (only if the corresponding symbol appears on the screen)

⊜   Monitor key, used exclusively for switching to the standard
                 operator interface (Sinumerik 802S/C base line).

M	Key not used for machine operation
[VAR]	Toggle key for handwheel valuation
Jog	JOG key for selecting "JOG" mode

1.1.2 Controls on the machine

(if provided by the machine manufacturer)

- NC start key
- NC stop key
- 4-way switch or axis direction keys (X+/X-/Z+/Z-) for axis movement
- Rapid traverse override switch or key for axis movement
- Spindle joystick or spindle direction keys for spindle control (clockwise or counterclockwise rotation)
- Jog spindle key
- Handwheel for X and Z axis
- Emergency Stop button

■

2

2 Sequence of Operations

2.1 Powering-up the control

Unless the machine manufacturer has set it up differently, the controller powers up automatically when the power supply is switched on. Internal test routines are performed during the power-up process. Once they have been successfully completed, the controller opens and displays the Reference Point screen form.

2.2 Reference point approach

Once the controller has powered up, the Reference Point screen form appears:

Fig. 2-1: Reference point approach

!   **Note**

Before starting the reference point approach, be sure to move the machine slide to a position from which the reference point can be approached in a positive axis direction.

2.2.1 Preparations for reference point approach

Before you can approach the reference points for the two axes, the following conditions must be met:

Prerequisite

- There must be no NC alarms pending!
  → Clear any that are pending by pressing the "Reset" function key.

- All machine axes must be in a position from which the reference point can be reached in a positive traversing direction.
  → In manual mode, use the handwheels to move the axes to the appropriate position ahead of the reference point.

2.2.2 Moving the axes ahead of the reference point

- Exit the Reference Point screen form by pressing the "End Refpkt" function key.

The following prompt appears:
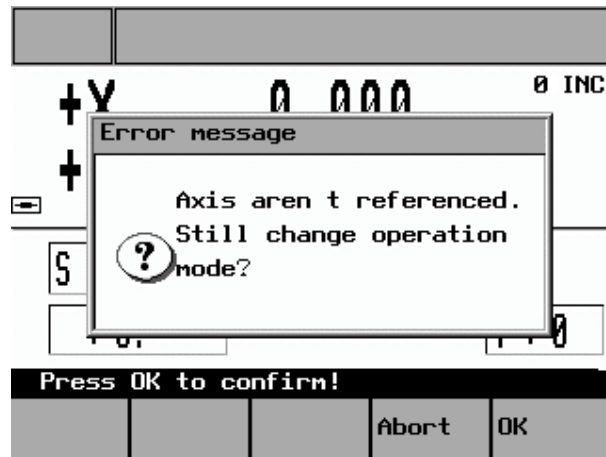
Fig. 2-2: Exiting the Reference Point Approach screen

- Press "OK" to acknowledge the message and return to the main service functions menu.
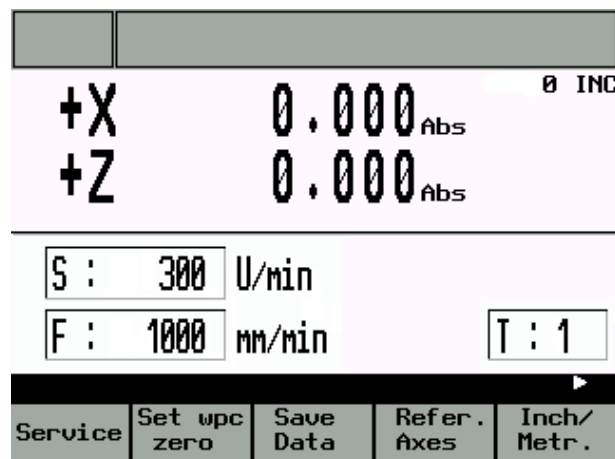
Fig. 2-3: Main Service Functions screen

- Press the "JOG" key once (otherwise, it will not be possible to switch to handwheel valuation).

- Then use the toggle key to set the handwheel valuation. The current setting will appear on the top-right of the screen (e.g., 100 INC).

- Then use the handwheel to move the axes to a position from which they can approach the reference point in a positive direction.

Caution

In this operating mode the axes can only be approached using the handwheel. Use of the 4-way switch or axis direction keys to move the axes is blocked. In addition, the spindle cannot be started in this mode.

- Press the "Refer. Axes" function key to switch back to "Reference point approach" mode.

2.2.3 Reference point approach

Displays

○   Axis has not yet been referenced

✛   Axis has already reached the reference point (is referenced)

Once all the conditions have been met (→ Preparations for reference point approach), reference point approach can be started for each axis as follows:

- Using the 4-way switch or axis direction key, start the X axis in the positive direction (X+). The reference point for the X axis is then approached automatically.

- At the end of this process the axis stops automatically and the symbol is displayed in front of the axis name.

- Once the X axis has reached its reference point, the same process can be repeated for the Z axis (Z+).

! **Note**

The axes must be referenced in this sequence, i.e. the X axis first, followed by the Z axis. Any other sequence will not be accepted by the controller.

Caution

If the machine manufacturer has not fitted an axis limit switch, there is a risk of a mechanical collision if the axis is on the wrong side of the reference point cam before the start of the reference point approach.
→ Preparations for reference point approach.

- Press the "End Refpkt." function key once both axes have been referenced to return to the main "Manual Machining" screen.

2.3 Manual Machining

The main Manual Machining screen looks like this:

Fig. 2-4: Main "Manual Machining" screen

Manual workpiece machining is carried out in this operating mode.

Here the cross-slide is controlled either
- By means of the handwheels for the X and Z axis (→ Machining with the handwheels) or

- By means of the 4-way switch or the axis direction keys (X+/X-/Z+/Z-)
  (→ Machining with the 4-way switch/axis direction keys)

The spindle is controlled by means of
- The spindle joystick or the individual spindle advance and spindle reverse keys
  (→ Spindle advance/reverse).

Machining technology data such as spindle speed, cutting speed and feedrate
must be set via special screen forms.
→ Machining technology data.

!   **Note**

Generally speaking the relevant machining technology data must be entered
before starting manual machining.

Caution

The maximum permissible spindle speed, which depends on the chucking device
fitted, must be entered in the Machining Technology Data screen form!
Failure to pay sufficient attention to this point can lead to serious damage as a
result of the chucking device speed being exceeded.
→ Machining technology data.

2.3.1 Displays on the main "Manual Machining" screen:

Actual-value display:	The current actual position for the X and Z axis is shown here. If the abbreviation "Abs" appears after the position value, the display shows the absolute machine position. If the abbreviation "Rel" appears here, however, it is a relative position reflecting the distance traversed since the display was last reset. → Toggling the display.
Traversing direction:	If the axis is moving, the current traversing direction is indicated by the appropriate sign (+/-) in front of the axis letter (X/Z).
S value:	The programmed value for either the spindle speed (Rev/min) or the cutting speed (m/min) is shown here. The display corresponds to the settings for the machining technology data. → Machining technology data.
F value:	The programmed feedrate is displayed here in either "m/min" or "mm/Rev", depending on the settings for the machining technology data. → Machining technology data.
T value:	The tool offset used by the controller is shown here. → Tool change.
INC value:	Indicates the handwheel pulse weighting currently set. → Machining with the handwheels

VICPAS
HMI Parts Center

2.3.2 Toggling the display

The radius turning function is accessed from the main "Manual Machining" screen (Fig. 4) as follows:

- Change the function key bar by pressing the "Extended menu" key [>] twice:
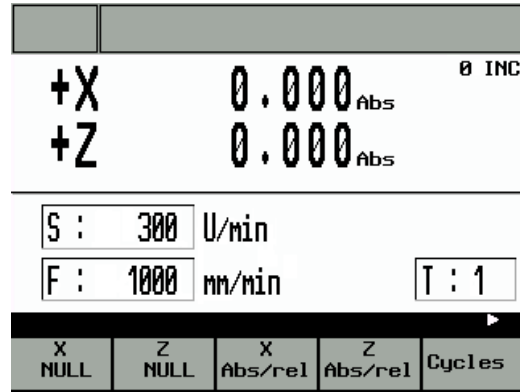
Fig. 2-5: Main service "toggling the display"

Press the function keys shown to alter the display as follows:

"X NULL":	Press the "X=0" or "Z=0" key to change the actual value display to "relative position display" and reset the display in the corresponding axis.
"Z NULL":	Press the "X=0" or "Z=0" key to change the actual value display to "relative position display" and reset the display in the corresponding axis.
"X Abs./rel.":	Press the "abs./rel." key to toggle the actual value display between "absolute position display" and "relative position display" in the corresponding axis.
"Z Abs./rel.":	Press the "abs./rel." key to toggle the actual value display between "absolute position display" and "relative position display" in the corresponding axis.

2.3.3 Machining with the handwheels

The handwheels for the X and Z axis are not mechanically connected to the feed screws. Electronic pulse generators mounted on the handwheels generate the information needed by the controller to execute the required traversing movement.

The handwheels are only active when the 4-way switch is in the zero position and the individual axis control keys are disabled.

The distance traversed per handwheel pulse depends on the handwheel valuation setting.
→ Handwheel valuation setting.

!   **Note**

If the handwheel valuation is set to "0" or if the feedrate override switch is in the "0" position, the handwheels are disabled.

2.3.4 Handwheel valuation setting

Press the "Toggle key for handwheel valuation" in the main "Manual Machining" screens to set the handwheel valuation.
The current setting will appear on the top-right of the screen (e.g., 100 INC).

If you are unable to adjust the handwheel valuation setting, it will be because the controller's internal mode is incompatible with this process. Press the "JOG" key once to resolve the problem.

Caution

An incorrect handwheel valuation setting can result in damage to the workpiece, tool and machine.

2.3.5 Machining with the 4-way switch/axis direction keys

The axis can be traversed in the desired direction by turning the 4-way switch or pressing the appropriate axis direction key. The feedrate at which the axis is traversed depends on the settings in the Machining Technology Data screen form.

The axis feedrate is also influenced by the feedrate override switch setting and, depending on the option selected in the Machining Technology Data screen form (Rev. feedrate/cutting speed), by the spindle override switch.

VICPAS
HMI Parts Center

If rapid traverse overlay is also actuated, the axis is moved at the maximum possible speed unless a different value has been set at the feedrate override switch.

! **Note**

If the feedrate override switch is set to "0", any type of axis movement is blocked. With the "Rev. feedrate" and "cutting speed" settings, the feed is blocked until the spindle reaches the setpoint speed.

2.3.6 Spindle advance/reverse

The spindle is started in the appropriate direction (spindle advance/reverse) by moving the spindle joystick or actuating the corresponding spindle direction key, provided that the chuck guard switch is enabled (chuck guard closed).

! **Note**

The spindle cannot be started unless the chuck guard switch is enabled.
→ Close the chuck guard.

Caution

Handling the chuck guard/chuck guard switch can lead to injury or mechanical damage and should be avoided.

When the spindle is switched off it brakes and comes to a halt. If a spindle brake is fitted, it is applied. If there is no spindle brake or it is switched off, the spindle can be rotated freely once it has stopped.

If "Spindle Positi." is selected in the "Spindle Positioning" screen form, the spindle is moved to the specified position after being switched off.
The spindle brake, if fitted, is then applied.
→ Spindle positioning.

The programmed spindle speed can be controlled by means of an appropriate spindle override switch setting (e.g. 50%).

2.3.7 Jog spindle

The jog spindle key (if provided by the machine manufacturer) can be used to rotate the spindle at low speed, for example to move a workpiece to a specific angular position.

VICPAS
HMI Parts Center

2.3.8 Tool change

In the case of conventional/cycle-controlled machines of this order of magnitude, with just a few exceptions, a manual tool change will generally be required. For this reason, the correct tool number (tool compensation) must be notified to the control by hand once the real tool change has taken place at the machine.

This is done as follows:
- Press function key "T" in the main Manual Machining screen form.

- The entry field for the T value is highlighted.

- Use the numeric keypad to enter the required tool number (tool offset) (permissible values 1-15).

- Press "Enter" to complete the process.

If no further messages appear, the controller has completed the tool change.

! **Note**

The tool number (tool offset) can only be changed, if all axes and the spindle are stationary.

Caution

The tool number (tool offset) entered in the T value field must correspond to the tool fitted in the machine. Otherwise, the tool will have to be re-calibrated ($\rightarrow$ Tool calibration). An uncalibrated or incorrectly calibrated tool can lead to dimensional errors or to incorrect cutting values.

If machining technology data settings for the tool number (tool offset) to be activated are found in the "Tool Data" screen form ($\rightarrow$ Entering tool technology data), the following message appears (relating to T4, in this example):

VICPAS
HMI Parts Center

Fig. 2-6: Tool technology data acceptance message

The question can be answered by pressing one of these function keys:

"OK"   The tool offsets for the X and Z axis corresponding to the last tool
              calibration operation are applied. The actual-value display changes,
              if necessary.
              The previous machining technology data is also overwritten by the
              tool technology data (which has been saved for this tool) (feedrate,
              spindle speed, cutting speed)
              → Entering tool technology data.

"Abort"   The tool offsets for the X and Z axis corresponding to the last tool
              calibration operation are applied. The actual-value display changes,
              if necessary.
              The previous machining technology data is retained.

2.3.9 Machining technology data

Machining technology data, such as spindle speed, cutting speed and feedrate,
defines the settings that the controller needs for the machining of workpieces.

! Note

The machining technology data can only be changed, if all axes and the spindle
are stationary.

There are two ways of changing the machining technology data:

- Simple changes to the spindle and feedrate values (S value/F value) can be
  made directly in the "Manual Machining" screen form.
  → Changing the spindle/feedrate value

- A separate screen form is provided for entering the full set of machining technology data.
  → Entering machining technology data

2.3.10 Changing the spindle/feedrate value

This method of adapting technology data can be used to change values quickly during machining.
The values are changed using the appropriate function keys in the "Manual Machining" screen form.
However, it is not possible to switch between "m/min" and "rev/min" (spindle) or "mm/min" and "mm/rev" (axis) here (→ Entering machining technology data).

Changing the S value:	Press function key "S" to highlight the display field showing the current programmed spindle value. The programmed value can now be changed via the numeric keypad. Then press Enter to accept (activate) the change. The entry field is deselected and the process is complete. To abort the change, press function key "S" or the Alarm reset key instead of Enter. In this case the entry field is again deselected, the new value is cleared and the previously programmed spindle value is restored. You cannot switch between "m/min" and "rev/min" here.
Changing the F value:	Press function key "F" to highlight the display field showing the current programmed feedrate value. The programmed value can now be changed via the numeric keypad. Then press Enter to accept (activate) the change. The entry field is deselected and the process is complete. To abort the change, press function key "F" or the Alarm reset key instead of Enter. In this case the entry field is again deselected, the new value is cleared and the previously programmed feedrate value is restored. You cannot switch between "mm/min" and "mm/rev" here.

!   **Note**

The S value or F value can only be changed if all axes and the spindle are stationary.

VICPAS
HMI Parts Center

2.3.11 Entering machining technology data:

To open the screen form for machining technology data, press the "Spindle" function key in "Manual Machining".

The following screen form appears:

```
┌─────────┬──────────────────────────┐
│         │                          │
├─────────┴──────────────────────────┤
│ Const.SpindleSpeed  ████    0 Rev/min│
│ Const.cuttingSpeed        0.0 m/min │
│ Max. spindle speed        0 Rev/min │
│ Time feed                 0 mm/min  │
│ Feedrate per rev.      0.00 mm/Rev  │
│ ○Spindle speed   Time feed          │
│ ○Spindle speed   rev. feedrate      │
│ ○Cutting speed   rev. feedrate      │
├─────────┬───────┬─────────┬─────────┤
│ Spindle │       │ Spindle │         │
│ Positi. │       │ Feed    │   Back  │
└─────────┴───────┴─────────┴─────────┘
```

Fig. 2-7: Machining technology data input

When the screen form appears, the last entry field to have been edited is highlighted and a new value can be entered. To change a value in a different entry field, use the cursor keys to move to that field. Enter the value via the numeric keypad and then press Enter to accept (activate) the value. To abort the change and restore the previous entry value, use the cursor keys to move to a different entry field or exit the screen form (by pressing the "Back" function key), without pressing Enter.

The meanings of the individual entry fields are defined as follows:

Const. Spindle Speed	This entry value defines the programmed spindle speed for machining with "Spindle speed + Time feed" or "Spindle speed + Rev. feedrate". The value is only fully reached if no reduction is specified via the spindle override valuation or the spindle speed limit parameter ("Max. spindle speed").
Const. cutting Speed	Cutting speed entry value for machining with "Cutting speed + Rev. feedrate". The spindle speed is adjusted to the machining diameter of the workpiece so that uniform cutting conditions are achieved. Since the spindle would theoretically have to rotate at an "infinitely high" speed at the rotational center point in this mode, it has to be limited via the "Max. spindle speed" parameter. The constant cutting speed can also be influenced by means of the feedrate and spindle override valuation settings.

VICPAS
HMI Parts Center

Max. spindle speed	It is vital to enter the maximum permissible spindle speed here. The spindle speed is then limited to this value under all circumstances, so that the permissible chucking device speed, for example, cannot be exceeded.
Time feed	If Time feed is selected, the axes are moved at the speed entered here (mm/min) (unless rapid traverse override is activated). It can be influenced by the feedrate override switch setting.
Feedrate per rev.	The value entered here determines the axis velocity in "Spindle speed + Rev. feedrate" or "Const. cutting speed + Rev. feedrate" modes (if no rapid traverse override is set). It is influenced directly by the feedrate override valuation setting and indirectly by the spindle override valuation setting.

Caution

In the entry field for the maximum permissible spindle speed, you must enter the value specified by the manufacturer for the chucking device that is fitted!
Failure to pay sufficient attention to this point can lead to serious damage as a result of the chucking device speed being exceeded.

• To select the desired machining mode, press the "Spindle Feed" function key one or more times:

   "Spindle speed + Time feed" → "Spindle speed + Rev. feedrate" → "Cutting speed + Rev. feedrate"

The symbols displayed next to the text have the following meaning:

Display: ○   Machining mode deselected
                  ◉   Machining mode active (selected)

2.3.12 Spindle positioning

The Spindle positioning function is intended purely for stopping the spindle in a specific position to facilitate fitting of the chuck guard. It is not intended for position-related machining.

If the Spindle positioning function is selected, the spindle is switched off and then moved to the position specified in the "Set spindle pos." entry field. If fitted, a spindle brake is then applied.

To open the "Spindle Positioning" screen form, press the "Spindle Positi." function key in the "Machining Technology Data" screen form.

The following screen form appears:

```
┌───────────┬───────────────────────────────┐
│           │                               │
├───────────┴───────────────────────────────┤
│ Set spindle stop                           │
├────────────────────────────────────────────┤
│ Please turn the spindle onto the           │
│ requested position.                         │
│                                             │
│ Actual spindle pos.               0.000     │
│ Set spindle pos.                 █0.000     │
│                                             │
│      ○Spindle positioning ON                │
│      ◉Spindle positioning OFF               │
├────────────────────────────────────────────┤
│                                             │
├────────┬──────┬──────┬──────┬──────────────┤
│ Enter. │  ON  │ OFF  │      │    Back       │
└────────┴──────┴──────┴──────┴──────────────┘
```

Fig. 2-8: Spindle positioning

When the screen form appears, there are two ways to enter the position:
- Use the numeric keypad to enter the position directly in the highlighted entry field for the "Set spindle pos.". Then press Enter to accept the value. Only values from 0 to 359.9 are permitted; otherwise, an error message appears and the value is not accepted.

- After switching off the spindle, rotate it manually into the required position and press Enter. The current spindle position is displayed in the "Set spindle pos." entry field and automatically accepted. If the spindle has not completed a full rotation since the controller was last powered up, the spindle is not yet synchronized and the position will therefore not be accepted. In this case, a corresponding message is displayed.

Press "ON" to activate spindle positioning and "OFF" to deactivate it.

The current status is indicated as shown below:

Display: ○   Function deselected
                 ◉   Function active (selected)

■

VICPAS
HMI Parts Center

3

3 Tool Data

3.1 Tool list

The controller is supplied with a tool list for 15 tools as standard. However, the individual tools must be calibrated prior to initial use.
→ Tool calibration.

Caution

An uncalibrated or incorrectly calibrated tool can lead to dimensional errors or to incorrect cutting values! If the values entered are very different from the actual tool values, there is a risk that the tool may break or the mechanism or workpiece may be damaged.

To open the "Tool Data Overview", press the "Tool" function key in the main "Manual Machining" screen form.

The following screen form appears:

Fig. 3-1: Tool data overview

The active offsets for each tool, known as the tool offsets for the X and Z axis, are displayed in this screen form. Experienced lathe operators can use this screen form to enter the tool offsets, as on CNC controllers. Other operators should simply refer to this screen form for information about the current offsets. The actual tool measuring process takes place in the "Tool Measuring" screen form.
→ Tool calibration.

Data can be entered in the "Tool Data Overview" as follows:
- Use the cursor keys to select the required field (field is highlighted)

- Select the required tool type (turning tool or drilling tool) by pressing the "Turn-/Boretool" function key

- Enter the values and press Enter to accept. The actual value display on the machine (in "Manual Machining") is now updated to reflect the change. If you do not wish to accept the value entered, either use the cursor key to leave the entry field or switch to another screen form without pressing Enter.

3-30   SINUMERIK 802S/802C BA_MM (BA) – 04/2005 Edition

3.2 Tool calibration

The supported tool calibration function is described below by reference to the X axis of a turning tool:

- Fit the tool to be calibrated into the machine.

- In the "Manual Machining" screen form select the appropriate tool number.
  → Tool selection

- Switch to the "Tool Data Overview" screen form.

- Keep on pressing the cursor key until the relevant tool number is highlighted.

- Press the "Mess." function key; in the case of turning tools, the following screen form will appear:
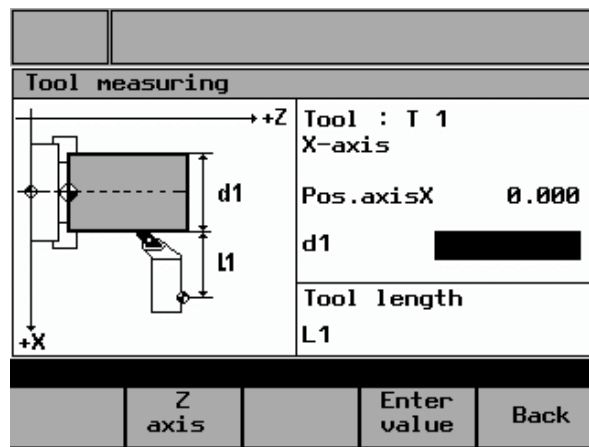

Fig. 3-2: Tool calibration (turning tools)

- Check that the current tool number appears in the display field for the tool, since the calibration operation will relate to this tool. If not, press the "Back" function key to return to the "Tool Data Overview" and select the correct tool.

- With the spindle running, adjust the X handwheel carefully to "scratch" a workpiece and then (without changing the X position) traverse the slide in the Z axis with the handwheel (axial turning).

- Switch off the spindle.

- Enter the diameter measured at the workpiece in entry field "d1".

- Press Enter to accept the value. The controller then automatically calculates the corresponding tool offset (in the radius) and displays this as value "L1" in the screen form.

! **Note**

If you exit the screen form at this point, the new offset will not take effect!

- Press the "Enter value" function key to apply the new tool offset for the selected tool in the X axis. Provided that the "scratch position" in the X axis has not been moved, the measured diameter is now displayed as the actual position in the "Manual Machining" screen form.

- Press the "Z axis" function key to change the screen form so that the process can be repeated for the Z axis.

- Press "Back" to return to the higher-level screen form.

3.3 Geometry tool data

In the "Geometry Tool Data" screen form, experienced lathe operators can enter tool offsets, as on CNC controllers, or correct a calibrated tool wear value without having to calibrate the tool again.

Other operators should simply refer to this screen form for information about the current offsets. The actual tool measuring process takes place in the "Tool Measuring" screen form. (→ Tool calibration)
However, here it is necessary to change the tool position "Change position" and enter the "Radius" value to ensure that the cycles are executed correctly.

Press the "Geom. data" function key in the "Tool Data Overview" screen form to open the following screen form for turning tools:
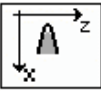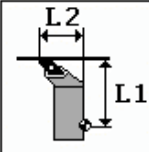
VICPAS
HMI Parts Center

Fig. 3-3: Geometry tool data (turning tools)

The displays for "Length 1" always relate to the X axis, those for "Length 2" to the Z axis.

If the geometry tool data for a drilling tool is invoked, the following screen form appears. The use and meaning of the values displayed are the same as those shown in the "Geometry Tool Data" screen form for turning tools, except that some of the values are not available for drilling tools.
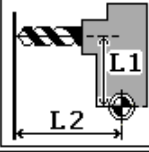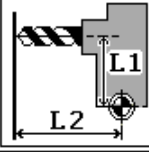

Fig. 3-4: Geometry tool data (drilling tools)

3.4 Entering tool technology data

This function allows the machine operator to store the cutting technology data for a tool, known as the tool technology data, together with the tool offset. The advantage of this function is that when the tool is selected in "Manual Machining", the machining technology data does not have to be changed; it can simply be copied from the tool technology data memory. The decision lies with the machine operator.

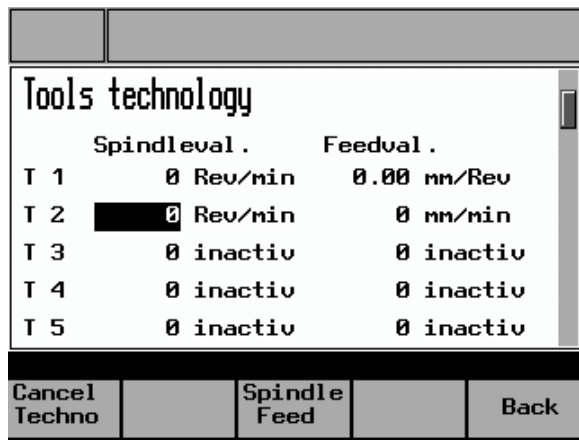Press the "Technology" function key in the "Tool Data Overview" screen form to open the following screen form:

Fig. 3-5: Tool technology

The following data can be entered in the "Tool Technology" screen form:

Spindle value	Depending on which operating mode is selected (see below), you can enter either the spindle speed in "Rev/min" or the cutting rate in "m/min".
Feed value	Depending on which operating mode is selected (see below) this parameter requires the desired axis speed to be entered in either "mm/min" or "mm/Rev".
Inactive	"Inactiv" indicates that no tool technology data has been entered yet for the corresponding tool.

The tool technology data should be entered as follows:

• Use the cursor keys to select an entry field for the tool in question (spindle or feedrate entry field is irrelevant).

• To select the desired machining mode, press the "Spindle Feed" function key one or more times:
  (for a more detailed description: → Entering machining technology data)

• "Spindle speed + Time feed"   ("Rev/min" + "mm/min")   →
  "Spindle speed + Rev. feedrate"   ("Rev/min" + "mm/Rev")   →
  "Cutting speed + Rev. feedrate"   ("m/min" + "mm/U")

• Use the numeric keypad to enter data in the entry fields for the spindle and feedrate value. Then press Enter to accept the data. If you leave the entry field or change the screen form without pressing Enter, the original value will be retained (the new value is not saved).

3.5 Deleting tool technology data

Data in the "Tools Technology" screen form can be deleted as follows:

- Use the cursor keys to select an entry field for the tool in question (spindle or feedrate entry field is irrelevant).

- Press the "Cancel Techno" function key

The following prompt appears:

Fig. 3-6: Deleting tool technology

The question can be answered by pressing one of these function keys:

"OK"   The tool technology data for the tool in question is deleted (spindle and feedrate value) and the system returns automatically to the "Tools Technology" screen form. The display "Inactive" indicates that the spindle and feedrate values have been deleted.

"Abort"   The tool technology data for the tool in question is retained (spindle and feedrate value) and the system returns automatically to the "Tools Technology" screen form. The existing tool technology data is retained.

■

4

4 Functions

4.1 Limit stops

Limit stops are used to stop the axes in a specific position. If an axis stops in the limit stop position, it cannot be moved again until the triggering limit stop is reset. This function can be used in "Manual Machining" mode to turn simple steps (including tapers) without the need for any further cycle parameterization.
→ Turning against mechanical stoppage

Boundary conditions

- The trip position is always an absolute dimension, i.e. it always corresponds to the position of the absolute actual value display in "Manual Machining". A relative limit stop position is not possible.

- A limit stop position can only be entered or accepted when the axes are stationary; otherwise, an error message appears.

4.1.1 Setting limit stops

The screen form for setting trips is opened from the main "Manual Machining" screen (Fig. 2-4) as follows:

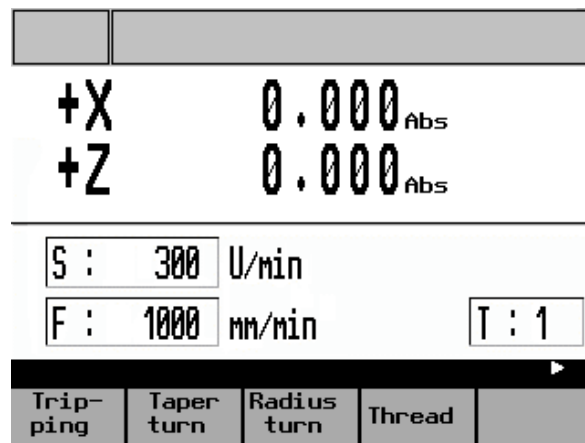- Change the function key bar by pressing the "Extended menu" key $\boxed{>}$:

Fig. 4-1: Extended "Manual Machining"

- Press the "Tripping" function key. The following screen form appears:

Fig. 4-2: Setting limit stops

The displays in the "Tripping" screen form have the following meanings:

ON   The limit stop shown in this line is switched on, so it is active.

OFF   The limit stop shown in this line is switched off, so it is inactive.

-X   Negative limit stop for the X axis. If the limit stop is active, the axis
          stops automatically when the absolute limit stop position (entered in
          this line) is reached by traversing the axis in the negative direction.

+X   Positive limit stop for the X axis. If the limit stop is active, the axis
          stops automatically when the absolute limit stop position (entered in
          this line) is reached by traversing the axis in the positive direction.

-Z   Negative limit stop for the Z axis. If the limit stop is active, the axis
          stops automatically when the absolute limit stop position (entered in
          this line) is reached by traversing the axis in the negative direction.

+Z   Positive limit stop for the Z axis. If the limit stop is active, the axis
          stops automatically when the absolute limit stop position (entered in
          this line) is reached by traversing the axis in the positive direction.

When the screen form appears there are two ways to enter the limit stop position:

- Direct position entry: Use the cursor keys to highlight the entry field for the
  appropriate trip. Now use the numeric keypad to enter the absolute position you
  require. Then press Enter to accept the value. If you do not wish to accept the
  value entered, either leave the entry field by pressing the cursor key or change
  the screen form by pressing the "Back" function key, without pressing Enter. In
  both cases the original entry value is retained.

- Accepting the current actual position: Use the cursor keys to highlight the entry field for the appropriate trip. Now press Enter to copy the position of the corresponding axis into the entry field.

! **Note**

A limit stop position can only be entered or accepted when the axes are stationary; otherwise, an error message appears.

! **Note**

The trip position is always an absolute dimension, i.e. it always corresponds to the position of the absolute actual value display in "Manual Machining". A relative limit stop position is not possible.

4.1.2 Activating/disabling limit stops

Trips can be activated or disabled by means of the function keys in the "Tripping" screen form, provided that all axes are stationary when the function key is pressed. Otherwise, an error message appears.

The function keys have the following functions:

"Single ON-OFF"   Pressing this function key activates or disables the limit stop whose entry field is highlighted. Use the cursor keys to select the appropriate entry field. If a trip is shown as "OFF" in the display, it can be activated by pressing the "Single ON-OFF" function key. It will then be shown as "ON" in the display. It can be disabled in the same way: If a trip is shown as "ON", pressing the function key changes it to "OFF". It is now disabled.

"All ON"   Pressing the "All ON" function key activates all trips, regardless of their previous status. The display is then set to "ON" for all trips.

"All OFF"   Pressing the "All OFF" function key disables all trips, regardless of their previous status. The display is then set to "OFF" for all trips.

VICPAS
HMI Parts Center

4.1.3 Turning against mechanical stoppage

The example below is intended to clarify how the limit stops work. Machining can generally also be carried out using the handwheel (rather than the 4-way switch or axis direction keys as described here):

A step of 100 mm in the Z direction and a 50 mm end diameter in the X direction is required, with a final machining allowance of 0.2 mm. The blank diameter is 70 mm and the end face is at 0 mm in the Z direction ($\rightarrow$ Tool calibration).

- Position the axes in front of the workpiece (e.g., X +75 mm/Z +5 mm).

- Check the machining technology data.

- Set the limit stops: -X at 50.4 mm/-Z at –99.8 mm (because of the final machining allowance), +Z at +5 mm. Disable the limit stop for +X; it is not required.

- Start spindle.

- Using the handwheel, infeed to the 1st depth of cut in the X direction.

- Start machining in the Z axis in the negative direction, using the 4-way switch or the axis direction key.

- When the -Z trip position at –99.8 mm is reached, the Z axis stops automatically and the "-Z trip reached" message appears.

- Switch off the 4-way switch or the axis direction key.

- Retract with the handwheel.

- Using the 4-way switch or axis direction key and rapid traverse overlay, traverse in front of the workpiece in the positive Z direction until the axis stops and the "+Z trip reached" message appears.

- Switch off the 4-way switch or the axis direction key.

- Using the handwheel, infeed to the next depth of cut in the X direction.

- Start machining in the Z axis in the negative direction, using the 4-way switch or the axis direction key.

Repeat the process until the roughing depth is reached (the "-X trip reached" message appears as the infeed motion is executed). Once this cut has been completed, adjust the limit stops to the finished dimension, provided that the axes are ahead of the workpiece.

- Set the stops to the finished dimension: -X to 50.0 mm/-Z to –100.0 mm

- Using the handwheel, infeed in the X direction until the "-X trip reached" message appears.

VICPAS
HMI Parts Center

- Start machining in the Z axis in the negative direction, using the 4-way switch or the axis direction key.

- When the -Z trip position at –100,.0 mm is reached, the Z axis stops automatically and the "-Z trip reached" message appears.

- Switch off the 4-way switch or the Z direction axis direction key and start in the positive X direction (finishing the end face).

- Switch off the 4-way switch or the X direction axis direction key as soon as the tool tip leaves the workpiece.

- Machining is now complete and the limit stops can be disabled again.

4.2 Taper turning

The Taper turning function is intended for the simple production of tapered workpieces. The procedure is basically the same as for "Manual Machining" apart from the addition of an angle (angle of taper). The angle input rotates the controller's internal coordinate system according to the angle value. When the 4-way switch or an axis direction key is actuated, the controller then uses the angle input to interpolate (and traverse) the X and Z axis accordingly.
The programmed axis feed then applies to the path being traversed and not to the corresponding axis.

If tapers with defined end points are to be turned, the use of limit stops is a helpful addition to this function.   Limit stops

---

! **Note**

A taper can only be traversed using the 4-way switch or the axis direction keys. Handwheel traversing is always a paraxial traverse and will not produce a taper.

! **Note**

Taper turning can only be selected when the spindle is running; otherwise, an error message appears.

The Taper turning function is accessed from the main "Manual Machining" screen (Fig. 2-4) as follows:

- Change the function key bar by pressing the "Extended menu" key ⊳.

VICPAS
HMI Parts Center

- Press the "Taper turn" function key. The following screen form appears:
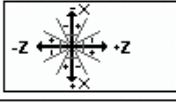
Fig. 4-3: Taper turning

As soon as the screen form appears, the entry field for the angle of taper is highlighted to enable the angle to be entered using the numeric keypad. A positive angle rotates the coordinate system towards the spindle when viewed in the X+ traverse direction, a negative angle rotates it towards the tailstock. Press Enter to accept the value. The taper angle remains active until the "Taper Turning" screen form is closed by pressing the "End Taper" function key.

The diagrams below show the relationship between the sign of the angle and the rotation of the coordinate system.
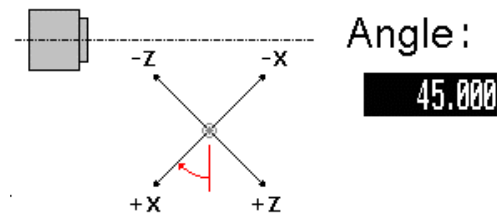
Positive angle:

Fig. 4-4: Rotation of coordinates with a positive angle
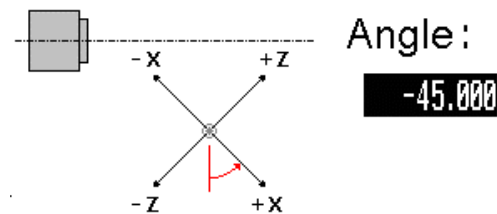
Negative angle:

Fig. 4-5: Rotation of coordinates with a negative angle

4.3 Radius turning

The radius turning function is intended for the simple production of inside and outside radii. All the technology data needed for the machining operation is automatically taken from "Manual Machining". The positions of the axes at the time the NC start key is pressed form the starting point for the radii to be traversed.

> **!** **Note**
>
> The required radius is executed by pressing NC start and can be interrupted at any time by pressing NC stop.
> The 4-way switch or the axis direction keys and the handwheels are always disabled when using this function.

> **!** **Note**
>
> Radius turning can only be selected when the spindle is running; otherwise, an error message appears.

The Taper turning function is accessed from the main "Manual Machining" screen (Fig. 2-4) as follows:

- Change the function key bar by pressing the "Extended menu" key $\boxed{>}$.

- Press the "Radius turn" function key. The following screen form appears:

Fig. 4-6: Radius turning

The entry fields have the following meanings:

x1 This entry value describes the position of the circle end position in the X axis. The value should be the "relative linear measurement" (in the radius) whose starting point is the circle start position (= current X axis position in the diameter).

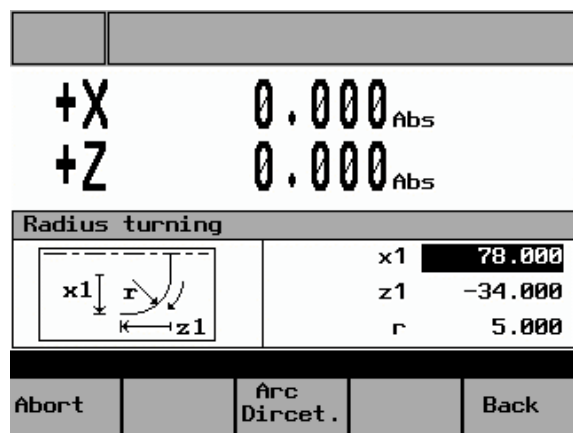**z1** This entry value describes the position of the circle end position in the Z axis. The value should be the "relative linear measurement" whose starting point is the circle start position (= current Z axis position).

r This entry value describes the radius to be traversed.

The function key assignments are as follows:

Arc Direct. This key is used to specify whether a clockwise or counterclockwise circle should be traversed. The chosen direction is indicated by a diagram in the screen form.

Abort This key can be used to abort a non-executable radius machining operation without having to leave the "Radius turning" screen form. The relevant entry value can then be corrected. For the operation to be aborted successfully, the only requirement is that the spindle must have been switched off before the function key is pressed. If not, a corresponding message will appear and the function will be disabled.

Caution

Omitting or using the wrong sign for the entry values (x1, z1) or entering the wrong arc direction ("Arc Direct." function key) can lead to a collision and may destroy the tool or the workpiece.

Once all the entry values have been entered correctly, radius turning can be started by pressing the NC start key. Since the starting point for the radius is always the current axis position, a kind of "surface texture" can be achieved by pressing the NC start key repeatedly.

Any radius execution can be interrupted immediately by pressing the NC stop key. Execution can only be continued by pressing the NC Start key before exiting the "Radius turning" screen form, since exiting the screen form terminates the function. If the screen form is opened again, the system assumes that you wish to execute a new radius starting from the current axis position (in this case, the point of interruption) and calculates it accordingly. An interrupted radius will not be completed.

4.4 Drilling - centered

The "Drilling" function is designed to produce deep-hole drill holes in the vicinity of
the turning center. Before the start of the cycle, the tool should be positioned in the
turning center so that the hole can be machined by infeeding in the longitudinal
axis only.

The technology data needed for the machining operation is automatically taken
from the entry fields in "Manual Machining".

!   **Note**

"Drilling" can only be selected when the spindle is running, otherwise an error
message appears.

Caution

In drilling operations, a spindle rotating in the wrong direction can cause serious
damage to the tool, workpiece or machine!
→ Check the direction of rotation of the spindle before pressing NC start!

Caution

Before starting the drilling operation, ensure that the tool is correctly positioned in
the turning center; otherwise, the drilling tool or workpiece may be damaged!!
→ Check that the tool is in the turning center before pressing NC start!!

The "Drilling - centered" function is accessed from the main "Manual Machining"
screen (Fig. 2-4) as follows:

- Change the function key bar by pressing the "Extended menu" key ▷.

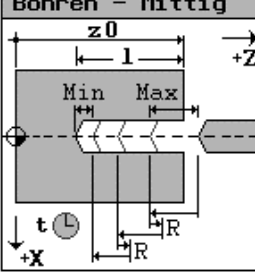- Press the "Drilling" function key. The following screen form appears:

Fig. 4-7: Drilling - centered

The entry fields have the following meanings:

Bezug	z0	Start position for the drill hole in the longitudinal axis (absolute position of the Z axis).
Tiefe	l	Enter the depth of the drill hole to be created, taking the start position for the drill hole ("Refer. z0") as the starting point. The drilling direction is always towards the chuck and cannot be reversed.
Zust.	Max	Maximum infeed value for the 1st infeed in the longitudinal axis.
	Min	Minimum infeed value in the longitudinal axis, which must be observed for the final infeed.
Faktor		Degression factor: the value by which the 2nd and all subsequent infeeds in the longitudinal axis must be multiplied. The following general rule applies: an entry value greater than 1 increases the infeed depth with each infeed, an entry value of less than 1 reduces it with each infeed. To switch off degression, enter 1 (or 0) here.
Zeit	t	Dwell time on reaching the drilling depth.
Rückhub	R	Length of the return stroke in the longitudinal axis for chip breaking. This parameter is not displayed if "deswarfing" is selected.

Meaning of the function keys:

"Spanbr./ Entsp"	This key is used to toggle between machining with "swarf breaking" and with "deswarfing". With "swarf breaking", on reaching the corresponding infeed depth the tool is retracted in the axial axis by a defined value (swarf breaking) before the next infeed. With "deswarfing" on the other hand, the tool is withdrawn from the drill hole on reaching the corresponding infeed depth. The next infeed then continues as usual. The current selection is shown as a graphic in the left-hand section of the screen. If "deswarfing" is selected, no value is shown for the "return".
"Achspos übern."	The current actual position of the relevant axis can be accepted by selecting an input parameter (entry field is highlighted) and pressing this function key. If you try to accept the axis position in entry fields for which this type of value input is not viable, however, an error message appears, which must be acknowledged by pressing the function key.
"Ausführen"	This function key opens the screen form in which the actual machining operation takes place:

Fig. 4-8: Execute drilling

Once all the entry values have been entered correctly and the screen form shown above has been opened, drilling can be started by pressing the NC start key. The spindle must already be rotating in the correct direction and the drilling tool positioned in the turning center.

Machining proceeds as follows:
- Starting from the current axis position, the tool is traversed to the cycle start point in the longitudinal axis. This is calculated internally from the value for the "Referenc z0" parameter (taking into account the clearance distance).

- The first infeed in the axial axis (as defined in the "Infeed Max." parameter) is then performed.

- The subsequent traversing movement in the axial axis depends on whether "swarf breaking" or "deswarfing" has been selected. With "swarf breaking" the tool is retracted in the axial axis by the value set in the "return" parameter; with "deswarfing" the axial axis is positioned at the cycle start point.

- The subsequent infeeds in the longitudinal axis are always calculated in the same way: new infeed value = last infeed value x factor + return stroke value The new infeed value is monitored to ensure that it complies with the value for the "Infeed Min." parameter. If the infeed value is below the minimum infeed, this value is imposed, provided that the drilling depth allows it. The calculation is followed by the infeed in the longitudinal axis.

- Infeed motion and "swarf breaking/deswarfing" then alternate until the drilling depth specified in the "Length I" parameter is reached.

- Once the required drilling depth is reached, the waiting time specified in the "Dwell t" parameter begins.

- At the end of this waiting time, the tool is traversed to the cycle start point in the longitudinal axis.

Machining is now complete and once the spindle has been switched off the "Execution" screen form can be closed by pressing the "Cycle finish" function key.

4.5 Tapping

The "Tapping" function is designed to produce internal threads in the vicinity of the turning center using a compensating chuck.
Before the start of the cycle, the tool should be positioned in the turning center so that the hole can be machined by infeeding in the longitudinal axis only.

The technology data needed for the machining operation is automatically taken or calculated from the entry fields in "Manual Machining", as follows:

Spindle   The cycle takes the spindle speed setting directly from the entry
speed   field for "Const. spind. speed".
              → Entering machining technology data

Pitch   The pitch is formed as follows according to the option selected in the "Machining Technology Data" screen form:

- "Spindle speed + Time feed":
  In this case, the pitch (mm/rev) is calculated as follows from the entry value for the Time feed (mm/min) and the current spindle speed (rev/min):

  The calculation is made when the NC start key is pressed in the "Execution" screen form for tapping. Since the actual speed forms part of the calculation, it is essential that the spindle override valuation is already set to "100%" at this point. Otherwise, a defined pitch cannot be set.

- "Spindle speed + Rev. feedrate":
  Here the pitch is taken directly from the "Rev. feedrate" value (mm/Rev) in the Machining Technology Data screen form.

- "Cutting speed + Rev. feedrate":
  Here the pitch is taken directly from the "Rev. feedrate" value (mm/Rev) in the Machining Technology Data screen form.

! **Note**

"Tapping" can only be selected when the spindle is running, otherwise an error message appears.

Caution

In tapping operations, a spindle rotating in the wrong direction can cause serious damage to the tool, workpiece or machine.
→ Check the direction of rotation of the spindle before pressing NC start.

Caution

Before starting the tapping operation, ensure that the tool is correctly positioned in the turning center; otherwise, the tapping tool or workpiece may be damaged.
→ Check that the tool is in the turning center before pressing NC start.

VICPAS
HMI Parts Center

Caution

If "Time feed" is selected in the "Machining Technology Data" screen form, in order for the pitch to be calculated correctly, the spindle override valuation must be set to "100%". Otherwise, the tapping tool or workpiece may be damaged. Prior to NC start, check that the spindle override valuation is set to 100%!!

The "Tapping" function is accessed from the main "Manual Machining" screen (Fig. 2-4) as follows:

- Change the function key bar by pressing the "Extended menu" key $\boxed{>}$.

- Press the "Gewindebohren" function key. The following screen form appears:
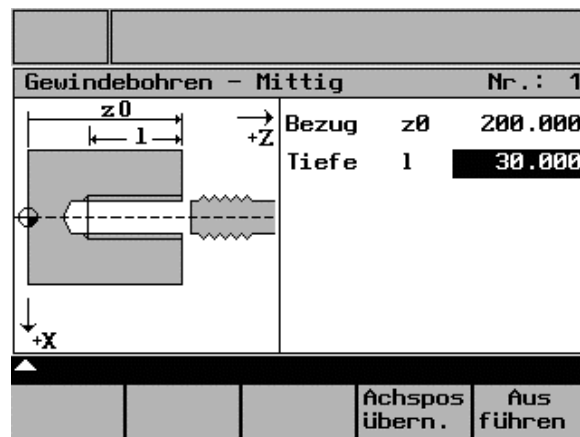
Fig. 4-9: Tapping

The entry fields have the following meanings:

Bezug	z0	Start position for the drill hole in the longitudinal axis (absolute position of the Z axis).
Tiefe	l	Enter the thread length here. The tapping direction is always towards the chuck and cannot be reversed. The choice of "left-hand or right-hand thread" depends on the direction of rotation of the spindle and the thread tapping tool.
"Achspos übern."		The current actual position of the axial axis can be accepted by selecting the input parameter "Referenc z0" (entry field is highlighted) and pressing this function key. If you try to accept the axis position in the "Length l" entry field, however, an error message appears which must be acknowledged by pressing the function key.

"Aus-   This function key opens the screen form in which the actual
führen"   machining operation takes place:

Fig. 4-10: Execute tapping

Once all the entry values have been entered correctly and the screen form shown
above has been opened, machining can be started by pressing the NC start key.
The spindle must already be rotating in the correct direction and the tapping tool
positioned in the turning center.

Machining proceeds as follows:

- Starting from the current axis position, the tool is traversed to the cycle start
  point in the longitudinal axis. This is calculated internally from the value for the
  "Referenc z0" parameter (taking into account the clearance distance).

- The controller then waits (at the cycle start point) for the next marker pulse from
  the spindle encoder in order to start the axis movement in the longitudinal axis
  (defined thread start point).

- When the thread length (end point) is reached, the spindle and longitudinal axis
  change direction and withdraw the tapping tool from the drill hole again.

- The longitudinal axis then stops at the cycle start point and the spindle changes
  direction again. The spindle is now running in the direction in which it was
  originally started.

Machining is now complete and once the spindle has been switched off the
"Execution" screen form can be closed by pressing the "Zyklus ENDE" function key.

VICPAS
HMI Parts Center

4.6 Groove cycles/parting

The "Groove cycles" function is suitable for producing symmetrical grooves on the lateral surface and for parting turned parts. Groove cycles can be used to produce filleted corners or beveled edges (45° chamfer) on the lateral surface.
In addition, the "Multiple execution" function can be used to produce multiple partings and multiple grooves with a uniform offset.

The technology data needed for the machining operation is automatically taken from the entry fields in "Manual Machining".

! Note

"Groove cycles" and "Parting" can only be selected when the spindle is running, otherwise an error message appears.
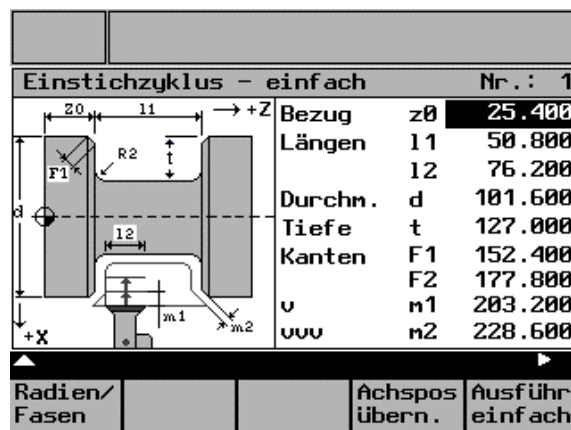
Caution

In machining operations, a spindle rotating in the wrong direction can cause serious damage to the tool, workpiece or machine!!
→ Check the direction of rotation of the spindle before pressing NC start!!

4.6.1 Groove cycle - single

The "Groove cycle - single" function is accessed from the main "Manual Machining" screen (Fig. 2) as follows:

- Change the function key bar by pressing the "Extended menu" key $\boxed{>}$ twice.

- Press the "Cycles" function key to call the main Cycle Selection screen.

- Press the "Grooving" function key. The following screen form appears:

Fig. 4-11: Groove cycle - single

The entry fields have the following meanings:

Bezug   z0   Starting position for the groove. The edge of the groove
               facing the spindle chuck is always specified here. The value
               to be entered is the absolute position in the longitudinal axis
               (Z axis).

Länge   l1   This value is the groove width, which together with the value
               for "Referenc z0" specifies the absolute position of the edge
               of the groove on the side of the groove facing away from the
               spindle. If the groove width setting is the same as the tool
               width ("Length l2") and "0" is assigned to the parameters
               "Edge R/C1" and "Edge R/C2", the parting function is
               activated.

         l2   Width of the grooving tool. The cycle needs this information to
               calculate the individual groove positions for the grooving
               operation.

Durch-   d   Starting diameter for the groove. The value to be entered is
m.   the absolute position in the transverse axis (X axis).

Tiefe   t   This value is the groove depth which together with the value
               for "Diameter d" specifies the absolute position of the base of
               the groove.

Kante   F/R1 Depending on the option selected, this value forms either a
               leading radius (display "R1") or a leading chamfer (display
               "C1") of less than 45° on both sides of the groove. Press the
               function key to toggle between radius and chamfer. An entry
               value of 0.0 switches this function off.

         F/R2 Depending on the option selected, this value forms either a
               radius (display "R2") or a chamfer (display "F2") of less than
               45° on both sides of the groove as the transition to the base
               of the groove. Press the function key to toggle between radius
               and chamfer. An entry value of 0.0 switches this function off.

v   m1   Enter the maximum infeed depth for roughing during
               grooving. The cycle's internal infeed calculation ensures that
               this entry value is not exceeded during machining.

vvv   m2   Final machining allowance perpendicular to the contour.

Meaning of the function keys:

"Radien/Fasen"   This function key is used to select whether the value in the
                      "Kanten" entry fields (R/C1, R/C2) should be interpreted as
                      a chamfer length or a radius. The setting only applies to the
                      entry field selected at the time the function key is pressed
                      (entry field is highlighted). The selection is shown next to
                      the corresponding field.

"Achspos übern."   The current actual position of the corresponding axis can be
                      accepted by selecting an input parameter (entry field is
                      highlighted) and pressing this function key. If you try to
                      accept the axis position in entry fields for which this type of
                      value input is not viable, however, an error message
                      appears, which must be acknowledged by pressing the
                      function key.

"Ausführen"   This function key opens the screen form in which the actual
                      machining operation takes place:

Fig. 4-12: Execute grooving cycle

Once all the entry values have been entered correctly and the screen form shown
above has been opened, machining can be started by pressing the NC start key.
The spindle must already be rotating in the correct direction and the axes must be
in a position from which the groove can be approached without the risk of a
collision.

Machining proceeds as follows:

- Starting from the current axis position, the first calculated groove position is approached (diagonally) in both axes, taking into account the clearance distance and final machining allowance.

- Execute the depth infeeds as a roughing motion in the transverse axis (X axis): each infeed depth is calculated internally so that firstly the setting "m1" is not exceeded and secondly the infeed distance is kept uniform until the base of the groove is reached (taking into account the final machining allowance). After each infeed, the tool is retracted by the clearance distance for chip breaking.

- When the base of the groove is reached for the first time, the tool is withdrawn from the material at the programmed feedrate.

- This is followed by the width infeed in the longitudinal axis: the width offset is calculated internally, taking into account the tool width (length "l2") and the groove width (length "l1") so that the machining is as uniform as possible.

- Depth infeeds are then alternated as roughing motion and width offset until the entire groove contour has been cleared. The only difference between the first depth infeed and the others is that when the base of the groove is reached, the tool is retracted by the clearance distance and then moved out of the groove in rapid traverse.

- Finishing is started immediately after the roughing operation. The entire contour is covered from both sides to the center of the base of the groove at the feedrate specified in the "Technology Data" screen form before the start of the cycle.

- Finally, the original position of the axes before the start of machining is approached diagonally.

Machining is now complete and once the spindle has been switched off the "Execution" screen form can be closed by pressing the "Zyklus ENDE" function key.

4.6.2 Groove cycle - multiple

Note

Since the "Groove cycle - multiple" function is an addition to "Groove cycle - single", the "Groove cycle - single" function must first be parameterized in full!
→ Groove cycle - single

The "Groove cycle - single" function is accessed from the main "Manual Machining" screen (Fig. 4-13) as follows:

- Change the function key bar by pressing the "Extended menu" key [>]; the following screen form appears:
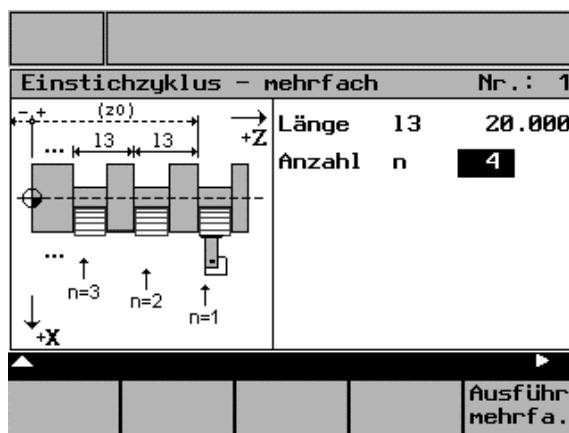

Fig. 4-13: Groove cycle - multiple

The entry fields have the following meanings:

Länge l3   Groove offset in the longitudinal axis (Z axis): the value entered here determines the offset between several identical grooves during production. The direction of the groove offset between the individual grooves is always towards the chuck.

An-   n   Number of grooves to be produced. Entering "0" or "1" here has
zahl   the same effect: a single groove is produced. If a value greater than 1 is entered, the corresponding number of grooves is produced and the necessary offset is formed by the value entered for the "Length l3" parameter.

Caution

In Groove cycle - multiple, ensure that the distance from the starting point "Reference z0" to the spindle is sufficient to enable all parameterized grooves to be produced; otherwise, a collision may occur between the tool and the spindle chuck!
→ Check the viability of the entry values before pressing NC start!

Meaning of the function key:
"Ausführen"   This function key opens the screen form in which the actual machining of multiple grooves takes place:

Fig. 4-14: Execute Groove cycle - multiple

Once all the entry values have been entered correctly and the screen form shown above has been opened, machining can be started by pressing the NC start key. The spindle must already be rotating in the correct direction and the axes must be in a position from which the grooves can be approached without the risk of a collision.

Machining proceeds as follows:
- Starting from the current axis position, the first groove is produced as described under "Groove cycle - single".

- The starting point for the next groove is then approached in the longitudinal axis (X axis), taking into account the clearance distance. The offset is always in the direction of the spindle (chuck).

- Another complete groove is then machined (as described under "Groove cycle - single").

- Groove machining and offset in the axial axis then alternate until the number of grooves specified in the "Number n" parameter has been produced.

- On completion of the final groove, the original position of the axes before the start of machining is approached diagonally.

Machining of the multiple grooves is now complete and once the spindle has been switched off the "Execution" screen form can be closed by pressing the "Zyklus ENDE" function key.

4.6.3 Parting

The "Parting" function is a subfunction of the "Groove cycle - single" function (→ "Groove cycle - single"), the difference being that specific settings for some of the input parameters change the machining cycle:

- Parameter "Length l1" = "Length l2" (= activate "parting")

- Parameter "Edges R/F1" = 0

- Parameter "Edges R/C2" = 0.0

Setting the groove width and the tool width to the same value activates the "parting" function. The radius/chamfer parameters must be set to "0", otherwise the cycle cannot calculate the contour and displays an error message ("Groove form not defined correctly").

Once all values have been entered correctly and the "Execution" screen form has been opened, machining can be started as described in "Groove cycle - single" (→ "Groove cycle - single") by pressing the NC Start key.
The spindle must already be rotating in the correct direction and the axes must be in a position from which the cut-off can be approached without the risk of a collision.

Machining proceeds as follows:
- Starting from the current axis position, the first calculated parting position is approached (diagonally) in both axes, taking into account the clearance distance.

- Execute the depth infeeds as a roughing motion in the transverse axis (X axis): each infeed depth is calculated internally so that firstly the setting "m1" is not exceeded and secondly the infeed distance is kept uniform until the base of the groove (parting depth) is reached. After each infeed, the tool is retracted by the clearance distance for chip breaking.

- When the base of the groove is reached, the tool is withdrawn from the material at the programmed feedrate.

- Finally, the original position of the axes before the start of machining is approached diagonally.

Machining is now complete and once the spindle has been switched off the "Execution" screen form can be closed by pressing the "Cycle finish" function key.

4.6.4 Multiple parting

The "Multiple parting" function is a subfunction of the "Groove cycle - multiple" function (→ "Groove cycle - multiple"), the difference being that specific settings for some of the input parameters change the machining cycle:

- Parameter "Length l1" = "Length l2" (= activate "parting")

- Parameter "Edges R/F1" = 0

- Parameter "Edges R/C2" = 0.0

Setting the groove width and the tool width to the same value activates the "parting" function. The radius/chamfer parameters must be set to "0", otherwise the cycle cannot calculate the contour and displays an error message ("Groove form not defined correctly").

Once all values have been entered correctly and the "Execution" screen form has been opened, machining can be started as described in "Groove cycle - multiple" (→ "Groove cycle - multiple") by pressing the NC Start key.
The spindle must already be rotating in the correct direction and the axes must be in a position from which the cut-offs can be approached without the risk of a collision.

Caution

In multiple parting, ensure that the distance from the starting point "Reference z0" to the spindle is sufficient to enable all parameterized parting motions to be completed; otherwise, a collision may occur between the tool and the spindle chuck.
→ Check the viability of the entry values before pressing NC start.

Machining proceeds as follows:

- Machining takes place in the same way as for machining of "multiple grooves" (→ Groove cycle - multiple). The only difference lies in the fact that this is a "parting" rather than a "grooving" operation (machining cycle → Parting)

Finally, the original position of the axes before the start of machining is approached diagonally.

Machining is now complete and once the spindle has been switched off the "Execution" screen form can be closed by pressing the "Cycle finish" function key.

VICPAS
HMI Parts Center

4.7 Thread cutting

The Thread cutting function provides a number of options for producing, remachining and recutting longitudinal and tapered threads.
They can be single-start or multiple-start threads.
The machining technology data for thread cutting is taken from "Manual Machining", as is the current tool offset.

Note

Any limit stops that are activated should be disabled before starting thread cutting or set to a value outside the traversing range needed for thread cutting.
→ Setting limit stops

4.7.1 Longitudinal thread cutting

The "Tapping" function is accessed from the main "Manual Machining" screen (Fig. 2-4) as follows:

- Change the function key bar by pressing the "Extended menu" key $\boxed{>}$ twice.

- Press the "Cycles" function key to call the main Cycle Selection screen.

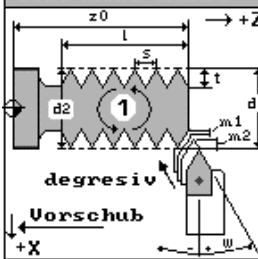- Press the "Thread cutting" function key. The following screen form appears:

Fig. 4-15: Longitudinal thread cutting

As with all other functions, the entry fields are generally selected by means of the cursor keys, data is entered via the numeric keypad and data accepted by pressing Enter. If you do not wish to accept a value, either leave the entry field by pressing a cursor key or change to a different screen form, without pressing Enter.

The entry fields have the following meanings:

Bezug   z0   Start position for the thread in the longitudinal axis (absolute position of the Z axis).

Länge   l   Enter the length of the thread to be created, taking the start position for the thread ("Referenc z0") as the starting point. The thread cutting direction is selected by pressing the "Feed direction" function key and is indicated in the diagram by means of an arrow. The choice of whether to produce a left-hand or a right-hand thread depends purely on the starting direction for the spindle.

Durch m.   d1   Start position for the thread in the transverse axis (absolute position of the X axis in the diameter). This value applies in the reference point.

           d2   End position for the thread in the transverse axis (absolute position of the X axis in the diameter).

Stei-gung   s   Enter the required pitch in mm/rev.

Tiefe   t   This parameter is used to set the thread depth. The infeed direction and infeed depth depend on other parameters ("Intern Extern." and "Linear Degrees." function keys; "Angle" parameter).

Winkel   w   Infeed angle; this specifies the angle of infeed during machining.

v   m1   Enter the maximum infeed depth (perpendicular to the cylinder/taper) for roughing. The internal infeed calculation ensures that this entry value is not exceeded during thread cutting.

vvv   m2   Final machining allowance perpendicular to the cylinder/taper.

Meaning of the function keys:

"Innen / Aussen"   This key is used to select whether an internal or an external thread is required. The selection is indicated by a diagram on the screen.

"Vorschu richt."   This function key is used to select the feed direction. The current selection is indicated in the diagram by means of an arrow below the word "Feed". Machining always starts in the "reference point" of the thread, which is set by means of parameters "z0" and "d1". By contrast, the end position for the thread is calculated from the reference point and thread length, as a function of the "feed direction" on the one hand and by direct input via parameter "d2" on the other.

"Gangzahl"   The number of threads per unit length can be incremented by pressing this function key. The current value is shown in the diagram as a number. When the maximum possible number of threads is reached, the system automatically reverts to a single-start thread (indicated by 1 in the diagram).

"Achspos übern."   The current actual position of the relevant axis can be accepted by selecting an input parameter (entry field is highlighted) and pressing this function key. If you try to accept the axis position in entry fields for which this type of value input is not viable, however, an error message appears, which must be acknowledged by pressing the function key.

"Ausführen"   This function key opens the screen form in which the actual machining operation takes place.

The following function keys can be accessed by pressing the "Extended menu" key
$\boxed{>}$:

"Linear Degrees."   This function key is used to switch between "linear infeed" and "degressive infeed". The current selection is shown in the diagram in plain text. "Linear infeed" means that roughing always takes place at a constant depth of cut, and the internal infeed calculation is designed so that the value for the "max. infeed depth" (m1) is not exceeded during the entire thread cutting operation. "Degressive infeed" on the other hand means that the volume of cut is kept constant throughout the entire thread cutting operation. In this case too, the value for "max. infeed depth" (m1) is observed throughout the entire operation.

"Thread recut."   This function key is used to select remachining or thread recutting ($\rightarrow$ Thread recutting).

VICPAS
HMI Parts Center

Once all the values have been entered correctly, press the "Ausführen" function
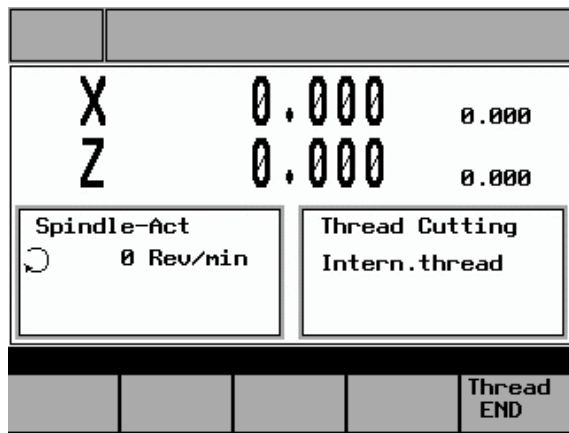key to open the Machining screen form. The following screen form appears:

Fig. 4-16: Execute a thread

Thread cutting can now be started by pressing the NC start key. The spindle must
already be rotating in the correct direction.

Machining proceeds as follows:
• Starting from the current axis position, the start position for the thread (d1/z0) is
  approached in rapid traverse.

• This is followed by infeed by the first depth of cut.

• The controller then waits for the next marker pulse from the spindle encoder in
  order to start the axis movements (longitudinal axis and/or transverse axis)
  (depending on the thread geometry).

• Once the end position for the thread has been reached in both axes, the tool is
  withdrawn from the workpiece in rapid traverse.

• The start position for the thread is then approached in the longitudinal and
  transverse axis in rapid traverse, observing a clearance distance.

• Infeed to the next depth of cut.

• Wait for the next marker pulse from the spindle encoder to start the axes...

This process continues until all cuts have been completed. An additional finishing
cut is then performed to smooth the thread and once the start position for the
thread in the axial and radial axis has been approached, the "Status Ready"
message appears.

There is now a choice of 2 options:
• If the thread is finished and the spindle has been switched off, press the
  "Thread END" function key to close the "Thread END" screen form.

4-64   SINUMERIK 802S/C BA_MM (BA) – 04/2005 Edition

- If you wish to continue machining the thread, e.g. if thread finishing is required, press NC Start again.
  → Thread shaving after thread cutting.

4.7.2 Thread recutting

The "Thread recutting" function is a subfunction of "Thread cutting". It can be used to recut a thread or to continue machining the thread on a workpiece that has been unclamped in between.

In order for "Thread recutting" to proceed correctly, the appropriate values have to be entered in the "Thread Cutting" screen form.
→ Thread cutting.

!   **Note**

The "Thread recutting" function uses the entry values from the "Thread Cutting" screen form. This screen form must therefore have been completed in order for thread cutting to proceed correctly.

The "Thread recutting" function is called by pressing the "Thread recut." function key in the extended "Thread cutting" menu bar.

The following screen form appears:

Fig. 4-17: Thread recutting

The values displayed for start of thread, end of thread and axis position relate to the longitudinal axis (Z axis). All values displayed on this screen form are for information only; it is not, therefore, possible to alter them directly.

The actual thread recutting operation must proceed as follows:
- Appropriate values must already have been entered in the "Thread Cutting" screen form at this point.

- The above screen form appears.

- The spindle must be stationary (switched off) and must already have been synchronized, in other words, it must have been turned through at least one full revolution since the controller was last powered up. Otherwise, an error message appears when the thread angle is accepted.

- Now use the handwheels to traverse the axes until the thread cutting tool can be threaded into the existing thread.

- Carefully introduce the tool into the thread. The "Axis position" shown on the screen must be between the values for "Thread initial p." and "Thread end point".

- Press the "Teach Angle" function key. The current spindle angle is now converted into the appropriate starting angle offset for thread cutting. The starting angle that is now displayed corresponds to the angle that will subsequently be used as the starting angle offset for machining a right-hand or left-hand thread.

- Using the handwheels, move the axes into a position from which the start of the thread can be approached safely.

- Press the "Execution" function key. The following screen form appears:
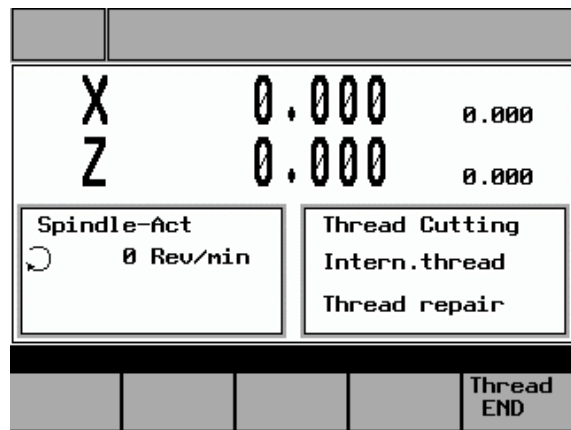
Fig. 4-18: Execute thread recut

The rest of the thread cutting process is exactly the same as that described for "Thread cutting".
→ Thread cutting.

The only difference is that thread cutting is not started with the marker pulse from the spindle encoder but with the angle that was calculated in the previous screen form (Fig. 4-17) as the "start angle offset".

4.7.3 Thread shaving after thread cutting

At the end of each thread cutting operation, you can choose to continue machining the thread, i.e., to perform shaving. Thread shaving can be performed either with or without an additional infeed, in which case it is merely a "smoothing cut".
To activate the function, simply press the NC Start key again at the end of the thread when the "Status Ready" message appears.

The following screen form appears:

Fig. 4-19: Thread shaving query

The "With/Without Feed" prompt can be answered using the function keys, which have the following meanings:

"Without Feed" Answering the prompt by pressing this key means that an additional cut is executed at the same infeed depth as the previous cut when you press NC start (additional "thread smoothing").

"With Feed" Answering the prompt by pressing this key means that next time the NC Start key is pressed, the tool is fed in by the amount entered in the "Val. f. Feed" field. The thread is machined accordingly in the next cut.

"Val. f. Feed" This function key is used to enter the infeed value for subsequent machining. On pressing the key, the corresponding entry field (already highlighted) is displayed in the screen form shown below to enable you to enter the value. Press Enter as usual to accept the value.

Fig. 4-20: Enter infeed

After pressing the "With Feed" or "Without Feed" function key, press NC Start to start machining an additional cut.
If you do not wish to do this, press the "Thread END" function key (with the spindle switched off) to leave the "Execute Thread" screen form, without pressing NC Start.

4.8 Cutting cycles

The cutting cycles (which are preset in the controller) are the easiest way of producing common paraxial cutting contours. They are defined by setting particular input parameters in the appropriate screen forms.
Contour machining can be "internal" or "external", roughing can be "axial" or "radial" and the machining direction can be "towards the chuck" or "towards the tailstock".
To facilitate handling, there are three different sets of cycle parameters (contour-dependent) available:

- Cutting cycle A:
  Simple stepped contour ("step"). Diameter settings and transitions to adjacent faces can include chamfers or radii.

- Cutting cycle B:
  Extended stepped contour with beveling. It is defined by means of 3 Transitions to adjacent faces can again include chamfers or radii.

- Cutting cycle C:
  Extended stepped contour with filleting. It is defined by means of 3 length settings and one radius setting. Transitions with chamfers or radii are not possible in this case.

The machining technology data and the current tool offset for cutting are taken from the entry fields in "Manual Machining".

!   **Note**

"Cutting cycle" can only be selected when the spindle is running, otherwise an error message appears!

Caution

In machining operations, a spindle rotating in the wrong direction can cause serious damage to the tool, workpiece or machine!!
→ Check the direction of rotation of the spindle before pressing NC start!

4.8.1 Cutting cycle A

This cyclic function is used to produce a simple stepped contour (step), with the option of working the transitions to adjacent faces as a radius or chamfer.

The "Cutting cycle A" function is accessed from the main "Manual Machining" screen (Fig. 2-4) as follows:

- Change the function key bar by pressing the "Extended menu" key twice

- Press the "Cycles" function key to call the main Cycle Selection screen.

- Press the "Cutting cycle A" function key. The following screen form appears:

Fig. 4-21: Cutting cycle A

As with all other functions, the entry fields are generally selected by means of the cursor keys, data is entered via the numeric keypad and data accepted by pressing Enter. If you do not wish to accept a value, either leave the entry field by pressing a cursor key or change to a different screen form, without pressing Enter.

VICPAS
HMI Parts Center

The entry fields have the following meanings:

Bezug	z0	Contour start position in the longitudinal axis (absolute position of the Z axis).
Länge	l	Enter the length of the "step" to be produced, taking the contour start position ("Referenc z0") in the axial axis (Z axis) as the starting point.
Durchm.	d1	Outside diameter of the "step" to be machined in the radial axis (absolute position of the X axis in the diameter).
	d2	Inside diameter of the "step" to be machined in the radial axis (absolute position of the X axis in the diameter).
Kante	F/R1	Depending on the option selected, this value forms either a transition radius (display "R1") or a transition chamfer (display "C1") of less than 45° between the end face and the inside diameter of the "step". Press the function key to toggle between radius and chamfer. An entry value of 0.0 switches this function off.
	F/R2	Depending on the option selected, this value forms either a transition radius (display "R2") or a transition chamfer (display "F2") of less than 45° between the inside diameter and the offset end face of the "step". Press the function key to toggle between radius and chamfer. An entry value of 0.0 switches this function off.
	F/R3	Depending on the option selected, this value forms either a transition radius (display "R3") or a transition chamfer (display "F3") of less than 45° between the offset end face and the outside diameter of the "step". Press the function key to toggle between radius and chamfer. An entry value of 0.0 switches this function off.
v	m1	Enter the maximum infeed depth for roughing. The internal infeed calculation ensures that the infeed is as uniform as possible throughout the roughing operation. This entry value represents the maximum value possible and is therefore not exceeded.
vvv	m2	Final machining allowance perpendicular to the contour.

Meaning of the function keys:

"Innen / Aussen"   This key is used to select whether an internal or an external contour is required. The selection is indicated by a diagram on the screen.

"Lage L/R"   This function key is used to select the machining direction for external contours. This can be either "towards the chuck" (default) or, if changed, "towards the tailstock". For internal contours, this setting is irrelevant and the function key is not available. The selection is indicated in the diagram displayed on the screen.

"Radien / Fasen"   This function key is used to select whether the value in the "Edges" entry fields (R/F1, R/F2, R/F3) should be interpreted as a chamfer length or a radius. The setting only applies to the entry field selected at the time the function key is pressed (entry field is highlighted). The selection is shown next to the corresponding field.

"Plan / Längs."   Changes the cutting direction in roughing. The options are: infeed in the longitudinal axis and roughing cut in the transverse axis, or roughing cut in the longitudinal axis and infeed in the transverse axis. The current selection is shown in the form of a diagram on the screen.

"Ausführen"   This function key opens the screen form in which the actual machining operation takes place.

The following function keys can be accessed by pressing the "Extended menu" key $>$:

"Teach Pos."   The current actual position of the corresponding axis can be accepted by selecting an input parameter (entry field is highlighted) and pressing this function key. If you try to accept the axis position in entry fields for which this type of value input is not viable, however, an error message appears, which must be acknowledged by pressing the function key.

"Change mach. type" Contact: This function key is used to set the required machining type. The following options are available: "Complete machining" → "Roughing only" → "Finishing only" The machining type currently selected is shown on the "Cycles" screen form.

Once all the values have been entered correctly, press the "Ausführen" function key to open the Machining screen form. The following screen form appears:

Fig. 4-22: Execute cutting cycle A

Cyclic machining can now be started by pressing the NC start key.

The spindle must already be rotating in the correct direction and the axes must be in a position from which the point on the workpiece that is to be machined can be approached without the risk of a collision.

Machining proceeds as follows:

- Starting from the current axis position, the calculated start position is approached (diagonally) in both axes (taking into account the clearance distance and final machining allowance).

- Infeed in the infeed axis (radial axis or axial axis, depending on whether "Radial" or "Axial" was selected). The infeed is calculated internally so that firstly the setting "m1" is not exceeded and secondly the infeed distance is kept uniform until the unmachined contour is reached (taking into account the final machining allowance).

- Execution of the paraxial roughing motion in the cutting axis until the unmachined contour is reached (taking into account the final machining allowance).

- Withdrawal from the material in the infeed axis by the infeed distance.

- Retraction by the clearance distance below 45° in both axes.

- Return in the cutting axis to the start position calculated in the cycle.

- New infeed in the infeed axis by the infeed depth calculated in the cycle.

- All roughing cuts are performed one after another, as described above. Following the final roughing cut in the cutting axis, withdrawal in the infeed axis, retraction below 45° and return in the cutting axis to the internal start position, finishing is started with the following movements:

- Traverse in the infeed axis to the finished dimension for the contour, corrected by the clearance distance.

- Infeed in both axes (below 45°) to the contour start point.

- Execution of the finishing motion along the parameterized contour.

- Retraction by the clearance distance below 45° in both axes.

- Return in the cutting axis to the start position calculated in the cycle.

- Finally, the original position of the axes before the start of machining is approached diagonally.

Machining is now complete and once the spindle has been switched off the "Execution" screen form can be closed by pressing the "Zyklus ENDE" function key.

4.8.2 Cutting cycle B

This cyclic function is used to produce a simple cutting contour, with an additional interpolation point allowing beveled or tapered contours. Transitions to adjacent faces can again be worked as a radius or chamfer.

The "Cutting cycle B" function is accessed from the main "Manual Machining" screen (Fig. 2-4) as follows:

- Change the function key bar by pressing the "Extended menu" key $\boxed{>}$ twice

- Press the "Cycles" function key to call the main Cycle Selection screen.

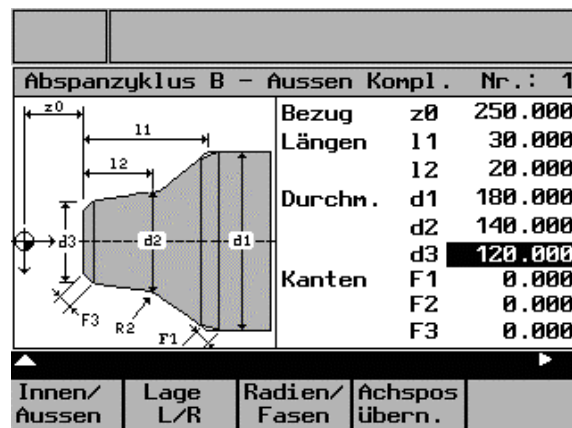- Press the "Cutting cycle B" function key. The following screen form appears:

Fig. 4-23: Cutting cycle B (part 1)

As with all other functions, the entry fields are generally selected by means of the cursor keys, data is entered via the numeric keypad and data accepted by pressing Enter. If you do not wish to accept a value, either leave the entry field by pressing a cursor key or change to a different screen form, without pressing Enter.

The entry fields have the following meanings:

Bezug	z0	Contour start position in the longitudinal axis (absolute position of the Z axis).
Länge	l1	Enter the length of the "step" to be produced, taking the contour start position ("Referenc z0") in the axial axis (Z axis) as the starting point.
	l2	Interpolation point position, which defines the position of the additional contour interpolation point in the longitudinal axis (Z axis).
Druchm.	d1	Outside diameter of the "step" to be machined in the radial axis (absolute position of the X axis in the diameter).
	d2	Interpolation point diameter, which together with the "Interpolation point position l2" parameter defines the position of the interpolation point in the radial axis (absolute position of the X axis in the diameter), allowing beveled faces to be produced within a "step".
	d3	Inside diameter of the "step" to be machined in the radial axis (absolute position of the X axis in the diameter).
Kante	F/R1	Depending on the option selected, this value forms either a transition radius (display "R1") or a transition chamfer (display "F1") of less than 45° between the end face and the first contour element of the "step". Press the function key to toggle between radius and chamfer. An entry value of 0.0 switches this function off.
	F/R2	Depending on the option selected, this value forms either a transition radius (display "R2") or a transition chamfer (display "F2") of less than 45° between first and second contour element of the "step". Press the function key to toggle between radius and chamfer. An entry value of 0.0 switches this function off.
	F/R3	Depending on the option selected, this value forms either a transition radius (display "R3") or a transition chamfer (display "F3") of less than 45° between the second contour element and the outside diameter of the "step". Press the function key to toggle between radius and chamfer. An entry value of 0.0 switches this function off.

VICPAS
HMI Parts Center

Meaning of the function keys in this screen form:

"Innen / Aussen" — This key is used to select whether an internal or an external contour is required. The selection is indicated by a diagram on the screen.

"Lage L/R" — This function key is used to select the machining direction for external contours. This can be either "towards the chuck" (default) or, if changed, "towards the tailstock". For internal contours, this setting is irrelevant and the function key is not available. The selection is indicated in the diagram displayed on the screen.

"Radien / Fasen" — This function key is used to select whether the value in the "Kante" entry fields (R/F1, R/F2, R/F3) should be interpreted as a chamfer length or a radius. The setting only applies to the entry field selected at the time the function key is pressed (entry field is highlighted). The selection is shown next to the corresponding field.

"Achspos übern." — The current actual position of the corresponding axis can be accepted by selecting an input parameter (entry field is highlighted) and pressing this function key. If you try to accept the axis position in entry fields for which this type of value input is not viable, however, an error message appears, which must be acknowledged by pressing the function key.

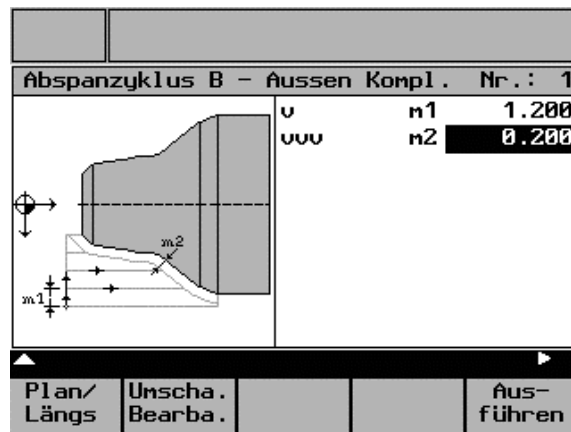The following screen form can be accessed by pressing the "Extended menu" key ▷ :

Fig. 4-24: Cutting cycle B (part 2)

The entry fields have the following meanings:

v	m1	Enter the maximum infeed depth for roughing. The internal infeed calculation ensures that the infeed is as uniform as possible throughout the roughing operation. This entry value represents the maximum value possible and is therefore not exceeded.
vvv	m2	Final machining allowance perpendicular to the contour.

Meaning of the function keys in this screen form:

"Plan / Länge."	Changes the cutting direction in roughing. The options are: infeed in the longitudinal axis and roughing cut in the transverse axis, or roughing cut in the longitudinal axis and infeed in the transverse axis. The current selection is shown in the form of a diagram on the screen.
"Umscha. Bearba."	This function key is used to set the required machining type. The following options are available: "Complete machining" → "Roughing only" → "Finishing only" The machining type currently selected is shown on the "Cycles" screen form.
"Ausführen"	This function key opens the screen form in which the actual machining operation takes place.

! **Note**

The "Cutting cycle B" function requires two screen forms. To ensure that machining proceeds correctly, it is therefore essential to enter the appropriate values in each screen form.

Once all the values have been entered correctly, press the "Execution" function key to open the Machining screen form. The following screen form appears:

Fig. 4-25: Execute cutting cycle B

Cyclic machining can now be started by pressing the NC start key.

The spindle must already be rotating in the correct direction and the axes must be in a position from which the point on the workpiece that is to be machined can be approached without the risk of a collision.

The machining cycle for "cutting cycle B" is identical to that described for "cutting cycle A". → Cutting cycle A.

At the end of machining, the original position of the axes before the start of machining is approached diagonally. Once the spindle has been switched off, press the "Zyklus END" function key to exit the "Execution" screen form.

4.8.3 Cutting cycle C

This cyclic function is used to produce a special cutting contour, with a filleted transition between the inside and outside diameter of the contour. Other chamfers or radii cannot be included.

The "Cutting cycle C" function is accessed from the main "Manual Machining" screen (Fig. 2-4) as follows:

- Change the function key bar by pressing the "Extended menu" key $\boxed{>}$ twice

- Press the "Cycles" function key to call the main Cycle Selection screen.

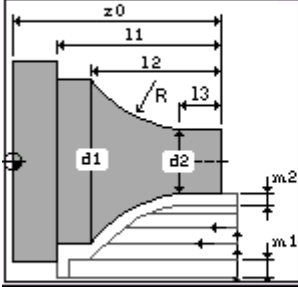- Press the "Cutting cycle C" function key. The following screen form appears:

Fig. 4-26: Cutting cycle C

As with all other functions, the entry fields are generally selected by means of the cursor keys, data is entered via the numeric keypad and data accepted by pressing Enter. If you do not wish to accept a value, either leave the entry field by pressing a cursor key or change to a different screen form, without pressing Enter.

The entry fields have the following meanings:

Bezug	z0	Contour start position in the longitudinal axis (absolute position of the Z axis).
Länge	l1	Enter the end point of the contour in the axial axis here, taking the contour start position ("Referenc z0") in the axial axis (Z axis) as the starting point.
	l2	End point of filleting in the longitudinal axis (Z axis).
	L3	Start point of filleting in the longitudinal axis (Z axis).
Durchm.	d1	Outside diameter of the contour to be machined in the transverse axis (absolute position of the X axis in the diameter).
	d2	Inside diameter of the "step" to be machined in the radial axis (absolute position of the X axis in the diameter).
Radius	R	This entry value determines the size of the filleting, the center of the circle being calculated internally. It is located on the imaginary line that is central and "normal" (90°) to the imaginary connecting line between points "l2/d1" and "l3/d2". The choice of whether the center point is on the side of the contour facing towards or away from the turning center is determined by the "Konvex Konkav" (convex/concave) function key setting. If the radius entered is too small, an error message will be displayed during machining (after NC start) since the contour cannot be produced in this case.
v	m1	Enter the maximum infeed depth for roughing. The internal infeed calculation ensures that the infeed is as uniform as possible throughout the roughing operation. This entry value represents the maximum value possible and is therefore not exceeded.
vvv	m2	Final machining allowance perpendicular to the contour.

Meaning of the function keys:

"Innen / Aussen."	This key is used to select whether an internal or an external contour is required. The selection is indicated by a diagram on the screen.
"Lage L/R"	This function key is used to select the machining direction for external contours. This can be either "towards the chuck" (default) or, if changed, "towards the tailstock". For internal contours, this setting is irrelevant and the function key is not available. The selection is indicated in the diagram displayed on the screen.

VICPAS
HMI Parts Center

"Konvex /   This function key is used to specify on which side of the
Konkav"   contour the circle center point should be located. The circular
                   machining direction, and hence the appearance of the
                   finished contour, is adjusted accordingly.

"Plan / Längs."   Changes the cutting direction in roughing. The options are:
                   infeed in the longitudinal axis and roughing cut in the
                   transverse axis, or roughing cut in the longitudinal axis and
                   infeed in the transverse axis. The current selection is shown
                   in the form of a diagram on the screen.

"Ausführen"   This function key opens the screen form in which the actual
                   machining operation takes place.

The following function keys can be accessed by pressing the "Extended menu" key
$\boxed{>}$:

"Achspos   The current actual position of the corresponding axis can be
übern."   accepted by selecting an input parameter (entry field is
                   highlighted) and pressing this function key. If you try to accept
                   the axis position in entry fields for which this type of value
                   input is not viable, however, an error message appears,
                   which must be acknowledged by pressing the function key.

"Umscha.   This function key is used to set the required machining type.
Bearba."   The following options are available:
                   "Complete machining" → "Roughing only" → "Finishing only"
                   The machining type currently selected is shown on the
                   "Cycles" screen form.

Once all the values have been entered correctly, press the "Execution" function key
to open the Machining screen form. The following screen form appears:

Fig. 4-27: Execute cutting cycle C

Cyclic machining can now be started by pressing the NC start key.

The spindle must already be rotating in the correct direction and the axes must be in a position from which the point on the workpiece that is to be machined can be approached without the risk of a collision.

The machining cycle for "cutting cycle C" is identical to that described for "cutting cycle A".
→ Cutting cycle A.

At the end of machining, the original position of the axes before the start of machining is approached diagonally. Once the spindle has been switched off, press the "Zyklus ENDE" function key to exit the "Execution" screen form.

4.9 Setting the workpiece zero point

The "Setting the workpiece zero point" function can be used to set the required length offset by modifying the clamping tolerances of the unmachined part (when machining multiple parts, for example).

Typical application/procedure

All the machining steps (cycles) for the workpiece are parameterized in relation to a "virtual zero point" (e.g., an end face). The unmachined part is clamped accordingly and the relevant surface, which corresponds to the "virtual zero point", is scratched. The workpiece coordinate system for the parameterized machining operation is then adjusted by means of the "Setting the workpiece zero point" function without the position that has been approached in the longitudinal axis being abandoned.

Additional information

When the "Setting the workpiece zero point" function is selected, the work offset is calculated automatically according to the current axis position in the longitudinal axis (Z) and then entered and activated on the NC memory for the 1st offset (G54). This will also set the position displayed for the longitudinal axis (Z) to 0.000, as this always corresponds to the workpiece coordinate system.

If the workpiece zero point is reset, the value "0.000" will automatically be entered in the NC memory in respect of the first work offset (G54). The workpiece coordinate system display will change to reflect this.

VICPAS
HMI Parts Center

Caution

Setting the "workpiece zero point" affects the absolute machining position of all machining steps that have been parameterized in the controller!
→ All machining steps will now be performed in relation to the zero point that has just been set!
Setting/resetting the workpiece zero point without due care and attention can result in serious damage to the tool, workpiece or machine!

The "Setting the Workpiece Zero Point" screen form is accessed from the main "Manual Machining" screen (Fig. 2-4) as follows:

Change the function key bar by pressing the "Extended menu" key $\boxed{>}$ three times

Press the "Set wkpce zp" function key. The following screen form appears:
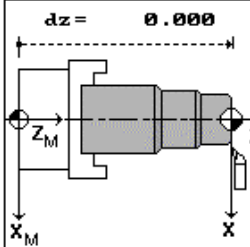
Fig. 4-28: Setting the workpiece zero point

This screen form shows the values that currently apply, as explained below:

"dz =" 　　　　　Value currently entered for the 1st adjustable work offset (G54).

"Werkstück-　　　Current axis position (X, Z axis) in the workpiece coordinate koordinat.:" 　　system, taking into account all active offsets (tool offset and work offset).

"Maschinen-　　　Current axis positions (X, Z axis) in the machine coordinate koordinat.:" 　　system

The possible settings on this screen form can be selected using the following function keys:

"Z-Achse setzen"   This function is used to set the "workpiece zero point".
The workpiece coordinate system for the longitudinal axis (Z) shows the value "0.000", the associated work offset that is required is calculated automatically and entered into the relevant location on the NC (G54).

"Z-Achse löschen"   This function resets the work offset that is currently stored on the NC.
→ The value "0.000" is entered in the G54 memory location. However, all other offsets and the active tool offset remain unchanged.

■

5

5 Service Functions

The operator can use the "Service Functions" screen form to find out about the software version installed, to change the menu language, and to adjust the brightness of the screen to conditions at the machine's place of installation.

The "Service Functions" screen form is accessed from the main "Manual Machining" screen (Fig. 2-4) as follows:

- Change the function key bar by pressing the "Extended menu" key three times.

- Press the "Service" function key. The following screen form appears:
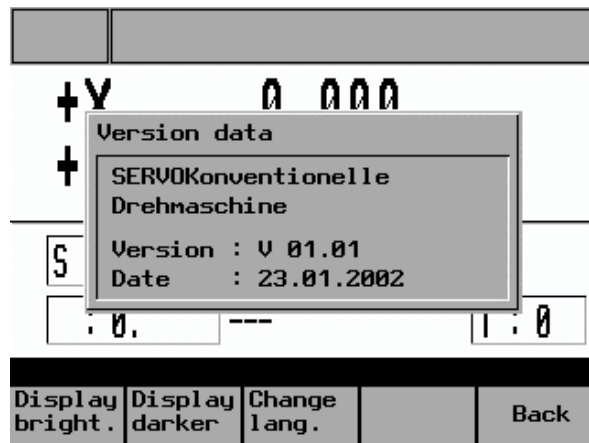
Fig. 5-1:   Service functions

The version number of the software version installed and its development date are automatically displayed in this screen form.

Other setting options in this screen form can be accessed via function keys.

The meanings of the function keys are as follows:

"Display bright."   This function increases the screen brightness incrementally each time the key is pressed. When the screen is set to maximum brightness, it is very difficult to see. The current setting is stored even when the machine is switched off.

"Display darker"   This function reduces the screen brightness incrementally each time the key is pressed. When the screen is set to "minimum brightness" it is very difficult to see. The current setting is stored even when the machine is switched off.

"Change lang."   This function can be used to switch to the "alternative language" installed by the machine manufacturer. The language change takes immediate effect and is retained even when the controller is switched off.

∎

VICPAS
HMI Parts Center

6

6 Saving Data

All the data saved in the controller (tool offsets, technology data, machine cycles, machine data, etc.) is stored in a RAM. This is backed up in a gold-cap capacitor when the system is shut down, although system constraints mean that the backup is designed for "short-term use" only.
If the system is shut down for an extended period (e.g., factory shutdown), as a result of which the backup period is exceeded, the data in the RAM is lost. If the data loss is detected when the controller is next powered up, the data from the last data backup (from the flash memory) is copied to the RAM. A corresponding message also appears in the alarm line.

Note

For data security reasons, we recommend that you back up your data at least once a week. If you intend to take the machine out of service for an extended period (e.g., factory shutdown, etc.), you must back up the data before shutting down the machine.

The procedure for backing up data in the flash memory is described below, starting from the main "Manual Machining" screen (Fig. 2-4):

- Change the function key bar by pressing the "Extended menu" ▷ key three times

- Press the "Save data" function key. The following screen form appears:

Fig. 6-1: Saving data

The question can be answered by pressing one of these function keys:

"YES"   All control data is copied to the flash memory. The current control
         state is now available as a backup in the event of a data loss. Any
         existing data backup is overwritten. Do not shut down the controller
         during the data backup process, since this may lead to a general data
         loss (and the controller will then have to be restarted). A
         corresponding message appears when data backup is in progress.

"NO"   A data backup is not performed and the function is aborted. Any
         existing data backup is retained.

Note

Do not shut down the controller during the data backup process, since this may
lead to a general data loss (and the controller will then have to be restarted).

■

VICPAS
HMI Parts Center

7 Switching to the Standard Operator Interface

7

SINUMERIK 802S/C BA_MM (BA) – 04/2005 Edition

7.1 General

This function can be used to switch to the Sinumerik 802S/C base line standard operator interface for service purposes. All drives (axes and spindle) must be stopped before executing this function.

You can access the function from either the "Reference Point Level" or the main "Manual Machining" screen.

7.2 "Reference Point Level" starting point

The following screen form should appear:
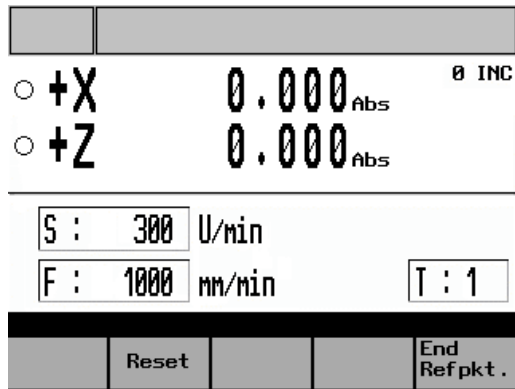
Fig. 7-1: Reference point approach

- Press the Monitor key ⬒; the following prompt will appear:

Fig. 7-2: Switching to the standard interface

The question can be answered by pressing one of these function keys:

"OK"   The operator interface instantly switches from "Manual Machine"
              to the Sinumerik standard "base line" operator interface. Only the
              required screen form should now be highlighted in the function key
              bar displayed.
              → For further information, see:
              "Operator's Guide for Sinumerik 802 S/C base line"

"Abort"   Operator interface does not change; the process is aborted.

Caution

You must only ever attempt to switch the operator interface when the axes and
spindle are stationary!

7.3 "Manual Machining" starting point

The function for switching the operator interface is accessed from the main
"Manual Machining" screen (Fig. 2-4) as follows:

- Change the function key bar by pressing the "Extended menu" key ▷ three
  times. The following screen form will then be displayed:

Fig. 7-3: Extended "Manual Machining" screen

- Press the Monitor key ⬯; the following prompt will appear:

Fig. 7-4: Switching to the standard interface

The question can be answered by pressing one of these function keys:

"OK"   The operator interface instantly switches from "Manual Machine" to the Sinumerik standard "base line" operator interface. Only the required screen form should now be highlighted in the function key bar displayed.
→ For further information, see:
"Operator's Guide for Sinumerik 802 S/C base line"

"Abort"   Operator interface does not change; the process is aborted.

Caution

You must only ever attempt to switch the operator interface when the axes and spindle are stationary!

7.3.1 Switching back to the "Manual Machine" operator interface:

To switch back to the Manual Machine operator interface from the Sinumerik 802 S/C base line standard operator interface, proceed as follows:

- All the axes and the spindle must be stationary before attempting to switch interfaces.

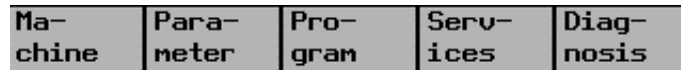- Switch to the main screen, where you should be able to see the following function key bar:

Ma- chine	Para- meter	Pro- gram	Serv- ices	Diag- nosis

Fig. 7-5: "Main Screen" function key bar

- Change the function key bar by pressing the "Extended menu" key $\boxed{>}$ until the following function key bar appears:

	ManMa			

Fig. 7-6: "Manual Machine" function key bar

- Press the "ManMa" function key.

- Switch to the "Manual Machine" operator interface is initiated.

Caution

You must only ever attempt to switch the operator interface when the axes and spindle are stationary!

■

SINUMERIK 802S/C BA_MM (BA) – 04/2005 Edition

8

8 Shutting Down the Controller

The controller can be shut down at any time, provided that a data backup is not in progress!
→ Saving Data.

However, we recommend that you position the axes before shutting down the system so that when the system is switched on again a reference point approach can be performed in both axes.

Caution

Do not shut down the controller during the data backup process, since this may lead to a general data loss (and the controller will then have to be restarted)!

■

9

9 Menu Tree

If the axes have not been referenced since the controller was switched on, the following menu structure applies:
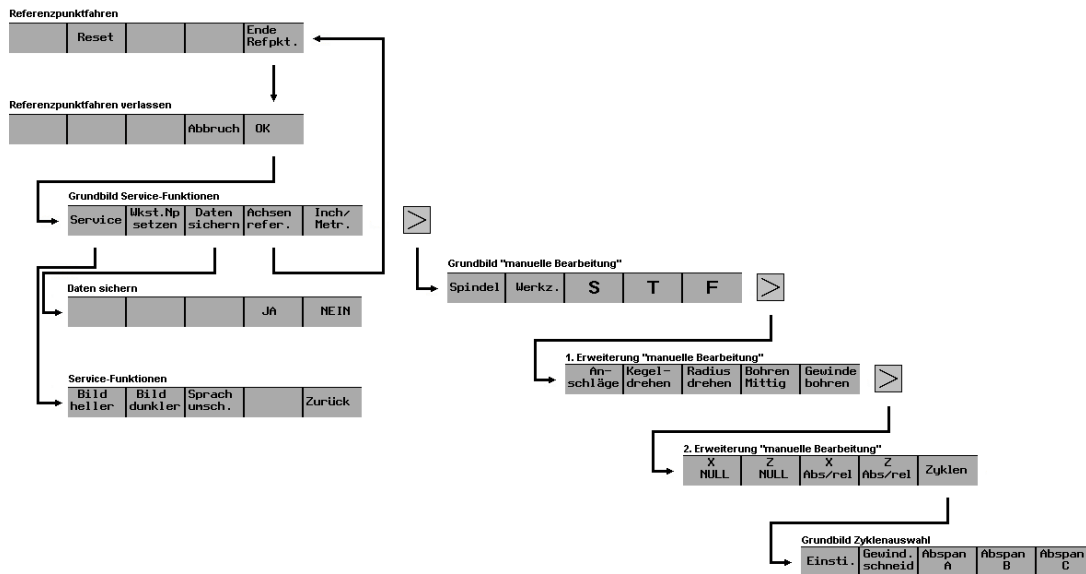

Fig. 9-1: Menu tree without valid reference point

If reference point approach has been completed in all axes since the controller was switched on, the following menu structure applies:
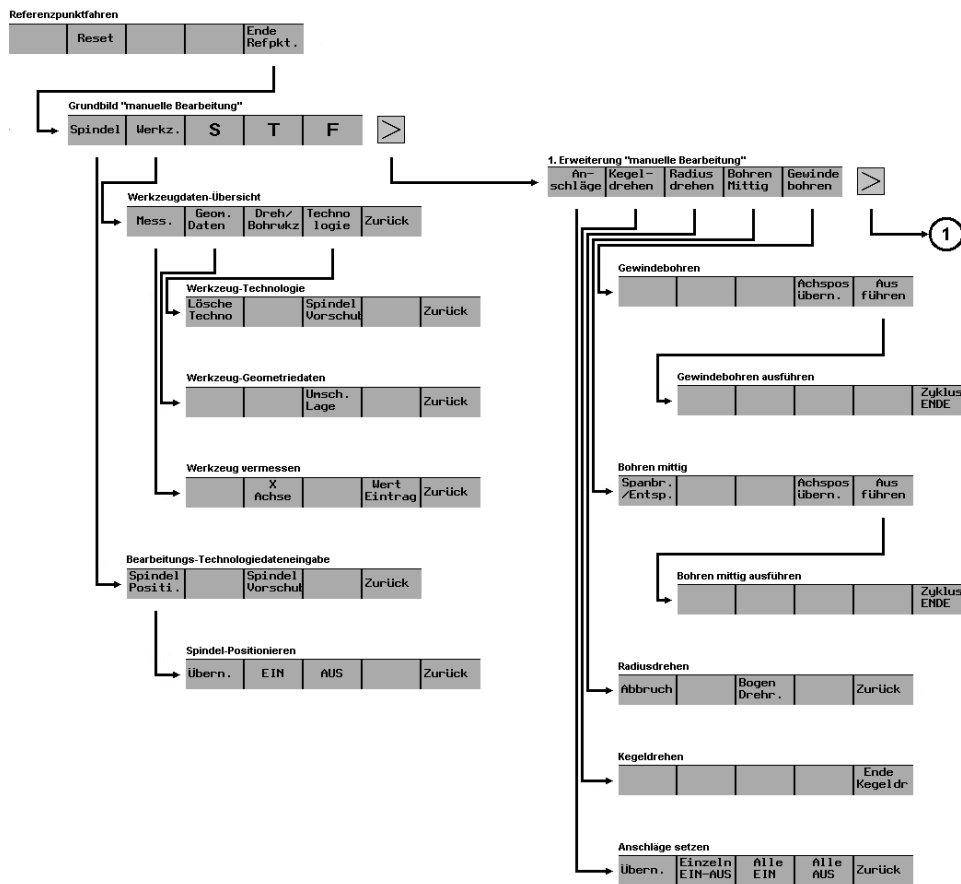
Fig. 9-2: Menu tree part 1 with valid reference point

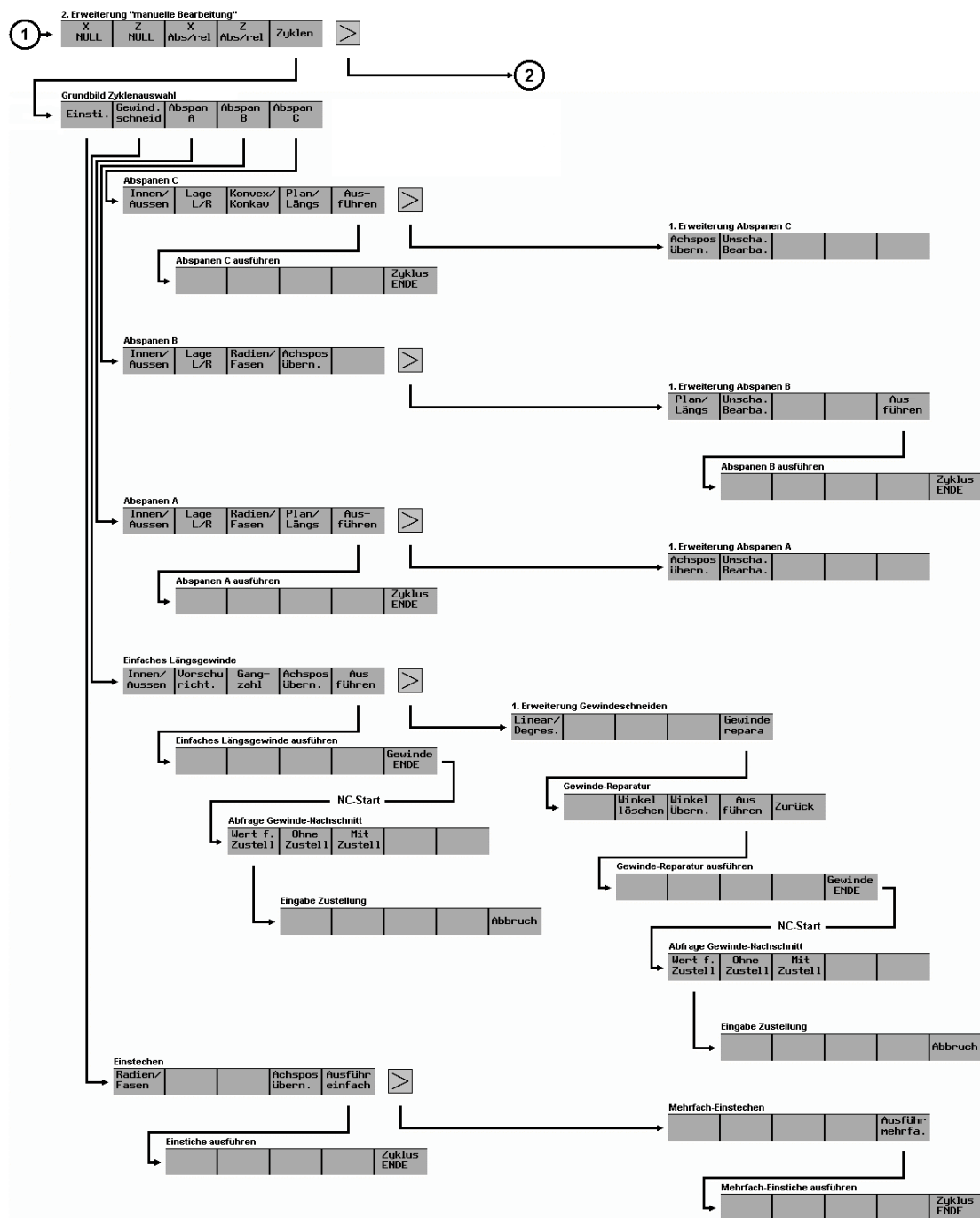SINUMERIK 802S/C BA_MM (BA) – 04/2005 Edition

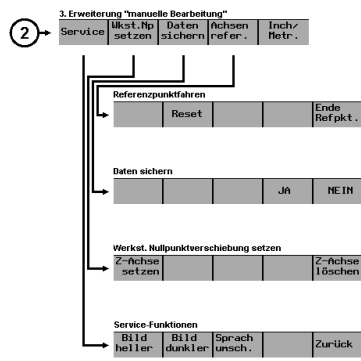Fig. 9-3: Menu tree part 2 with valid reference point

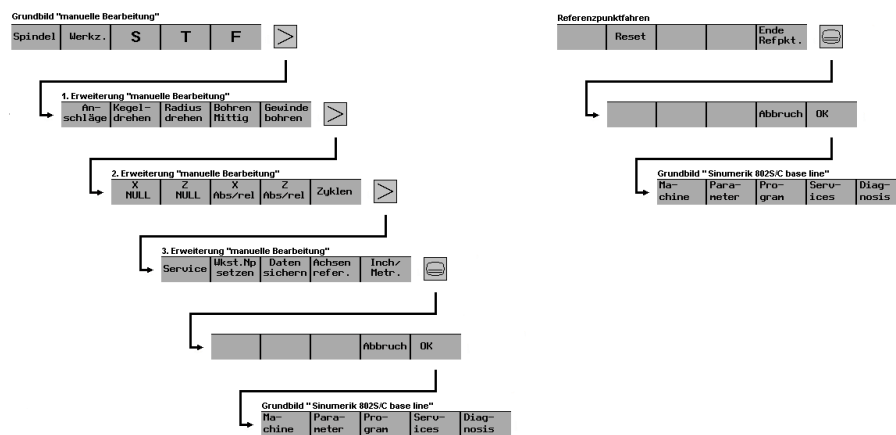Fig. 9-4: Menu tree part 3 with valid reference point

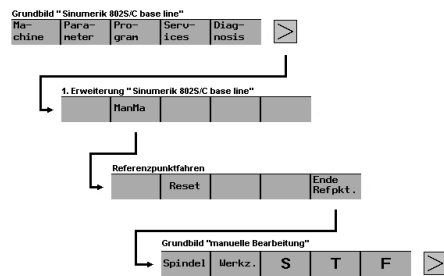Fig. 9-5: Switching to the "Sinumerik 802 S/C base line" standard operator interface

Fig. 9-6: Switching back to the "Manual Machine" operator interface

10

10 Messages

The meanings of the messages listed below differ from those given in the general "SINUMERIK Diagnostics Guide":

10631	+X limit stop reached
10631	+X limit stop reached
10631	-Z limit stop reached
10631	+Z limit stop reached

The following list of PLC messages applies, unless the machine manufacturer has defined an alternative or additional list of errors:

700000	Chucking device guard not closed
700001	Safety door not closed
700002	Reference point on X axis not reached
700003	Reference point on Z axis not reached
700004	Hardware limit switch on X axis not reached
700005	Hardware limit switch on Z axis not reached
700006	Compass switch setting not permitted
700007	Spindle joystick setting not permitted
700008	Spindle jogging not permitted
700009	Spindle brake will not open
700010	Lubrication level too low
700011	Lubrication pressure not reached
700012	I/R module not ready
700013	I2t monitoring of I/R module
700014	Axis limit switch bypassed (tool recovery)
700015	Fault Q21 (230 V incoming supply)
700016	Fault Q4 (coolant pump)
700017	Fault Q2 (forced ventilation)
700018	Single block selection active (key switch)
700019	No valid spindle/feed type
700020	Runtime monitoring JOG selection program
700021	Spindle override not 100%
700022	Spindle not switched on
700023	Feed override = 0%
700024	With/without feed?
700025	Direction of rotation in thread cannot be changed

■