SIEMENS

SINUMERIK

SINUMERIK 802D sl
Manual Machine Plus Turning

Programming and Operating Manual

Foreword

Description 1

Software interface 2

Turning On, Reference Point Approach 3

Setting-up 4

Manual machining 5

Machining the machining step program manually 6

Messages 7

Appendix A

Valid for

Controller	Software version
SINUMERIK 802D sl T/M	1.4 SP5

06/2009
6FC5398-6CP10-1BA0

Legal information

Warning notice system

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

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

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

⚠CAUTION
with a safety alert symbol, indicates that minor personal injury can result if proper precautions are not taken.

CAUTION
without a safety alert symbol, indicates that property damage can result if proper precautions are not taken.

NOTICE
indicates that an unintended result or situation can occur if the corresponding information is not taken into account.

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

Qualified Personnel

The device/system may only be set up and used in conjunction with this documentation. Commissioning and operation of a device/system may only be performed by **qualified personnel**. Within the context of the safety notes in this documentation qualified persons are defined as persons who are authorized to commission, ground and label devices, systems and circuits in accordance with established safety practices and standards.

Proper use of Siemens products

Note the following:

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

Trademarks

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

Disclaimer of Liability

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

Siemens AG
Industry Sector
Postfach 48 48
90026 NÜRNBERG
GERMANY

Ordernumber: 6FC5398-6CP10-1BA0
℗ 06/2009

Copyright © Siemens AG 2009.
Technical data subject to change

Foreword

SINUMERIK documentation

The SINUMERIK documentation is organized in 3 parts:

- General documentation
- User documentation
- Manufacturer/Service documentation

An overview of publications, which is updated monthly and also provides information about the language versions available, can be found on the Internet at:

http://www.siemens.com/motioncontrol

Follow menu items "Support" → "Technical Documentation" → "Ordering Documentation" → "Printed Documentation".

The Internet version of DOConCD (DOConWEB) is available at:

http://www.automation.siemens.com/doconweb

Information about training courses and FAQs (Frequently Asked Questions) can be found at the following website:

http://www.siemens.com/motioncontrol under "Support".

Target group

This publication is intended for programmers, planning engineers, machine operators and system operators.

Benefits

With the Programming and Operating Manual, the target group can develop, write, test and debug programs and software user interfaces.

It also contains the information required for the target group to operate the hardware and software of a product/system (generally a machine).

Standard scope

This documentation only describes the functionality of the standard version. Additions or revisions made by the machine manufacturer are documented by the machine manufacturer.

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

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

Technical support

If you have any technical questions, please contact our hotline:

	Europe / Africa
Phone	+49 180 5050 222
Fax	+49 180 5050 223
Internet	http://www.siemens.com/automation/support-request

	America
Phone	+1 423 262 2522
Fax	+1 423 262 2200
E-Mail	mailto:techsupport.sea@siemens.com

	Asia/Pacific
Phone	+86 1064 719 990
Fax	+86 1064 747 474
E-Mail	mailto:adsupport.asia@siemens.com

Note

Country-specific telephone numbers for technical support are provided under the following Internet address:

http://www.siemens.com/automation/service&support

Calls are subject to charge, e.g. €0.14/min from a German landline. Tariffs of other phone providers may differ.

Questions about this documentation

If you have any queries (suggestions, corrections) regarding this documentation, please send a fax or email to the following address:

Fax	+49 9131- 98 2176
E-Mail	docu.motioncontrol@siemens.com

A fax form is available in the appendix of this document.

SINUMERIK Internet address

http://www.siemens.com/sinumerik

EC Declaration of Conformity (only for hardware descriptions)

The EC Declaration of Conformity for the EMC Directive can be found/obtained:

- on the internet:
  http://support.automation.siemens.com
  under the product/order No. 15263595
- at the relevant regional office of the I DT MC Business Unit of Siemens AG.

Table of contents

Foreword **3**

1 Description **9**

1.1 Control and display elements 9

1.2 Error and status displays 10

1.3 Key definition of the full CNC keyboard (vertical format) 11

1.4 Key definition of the machine control panel 13

2 Software interface **15**

3 Turning On, Reference Point Approach **17**

3.1 Entry to the "Manual Machine Plus" operating area 17

3.2 Reference point approach 19

4 Setting-up **23**

4.1 Measuring tools 23

4.2 Limit stops 26
4.2.1 Setting and activating/deactivating limit stops 26
4.2.2 Turning against a stop 29

4.3 Setting the workpiece zero 31

5 Manual machining **33**

5.1 Fundamentals of manual machining 33

5.2 Display and operator control optios in the main screen for "Manual Machine Plus" 34
5.2.1 Toggling the display 38
5.2.2 Machining with the handwheels 39
5.2.3 Setting the increment weighting for the handwheel 39
5.2.4 Machining with axis direction switch 39
5.2.5 Spindle advance/reverse 40
5.2.6 Tool change 41
5.2.7 Changing the feedrate/spindle value 43
5.2.8 Changing the feedrate/spindle type 44
5.2.9 Change the speed limitation for constant cutting rate 46

5.3 Manual machining with machining types 47
5.3.1 Axis-parallel traversal 47
5.3.2 Manual taper turning 48
5.3.3 Manual radius turning 49
5.3.3.1 Radius turning type A 52
5.3.3.2 Radius turning type B 52
5.3.3.3 Radius turning type C 53

5.4 Manual machining using cycles (functions) 54
5.4.1 Principle operating sequence 54
5.4.2 General parameters 58
5.4.3 Manual drilling centered 60
5.4.4 Manual thread tapping 62

5.4.5 Manual grooving/parting ... 65
5.4.5.1 Groove cycle - single ... 65
5.4.5.2 Groove cycle - multiple ... 68
5.4.5.3 Parting cycle - single ... 69
5.4.5.4 Multiple tapping ... 70
5.4.5.5 Extended grooving ... 71
5.4.5.6 Multiple extended grooving ... 75
5.4.6 Manual thread cutting ... 76
5.4.6.1 Thread cutting ... 77
5.4.6.2 Thread recutting ... 81
5.4.6.3 Thread shaving after thread cutting ... 83
5.4.7 Roughing cycles ... 84
5.4.7.1 Roughing cycle A ... 85
5.4.7.2 Roughing cycle B ... 88
5.4.7.3 Roughing cycle C ... 91
5.4.7.4 Roughing cycle D ... 94
5.4.7.5 Roughing cycle E ... 97
5.4.7.6 Roughing cycle F ... 99
5.4.7.7 Roughing cycle, free contour: ... 101
5.4.7.8 Execute a roughing cycle ... 105

6 Machining the machining step program manually ... 107

6.1 Tool change in the machining step program ... 111
6.2 Teach In ... 114
6.3 Simulate machining ... 117
6.4 Executing the machining step program ... 120

7 Messages ... 123

A Appendix ... 125

A.1 Feedback on the documentation ... 125
A.2 Overview of documentation ... 127

Index ... 129

1 Description

1.1 Control and display elements

Operator control elements

The defined functions are called up via the horizontal and vertical softkeys. For a description, please refer to this manual:

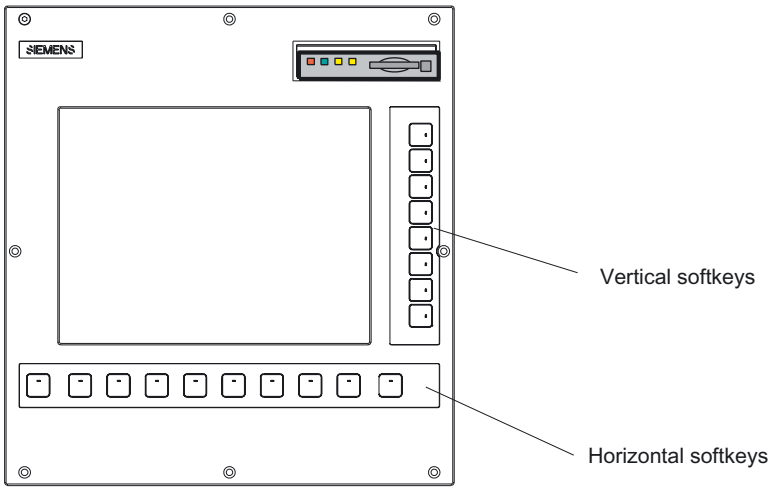
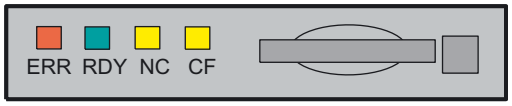


Figure 1-1 CNC operator panel

1.2 Error and status displays

LED displays on the CNC operator panel (PCU)

The following LEDs are installed on the CNC operator panel.



The individual LEDs and their functions are described in the table below.

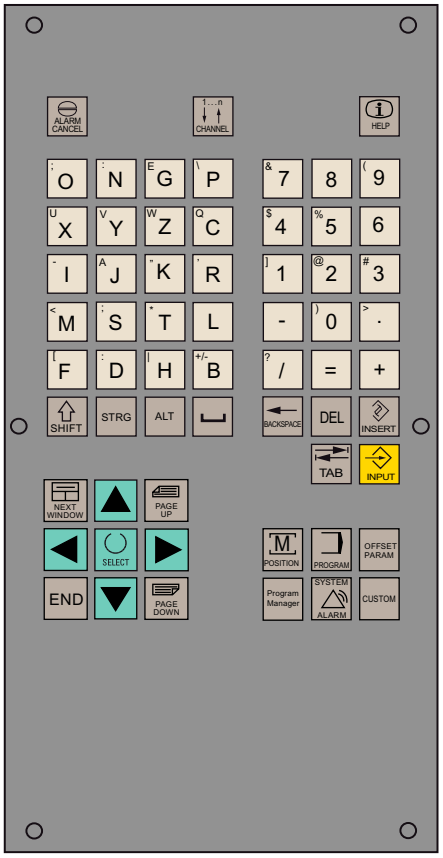
Table 1- 1 Status and error displays

LED	Meaning
ERR (red)	Serious error, remedy through power OFF/ON
RDY (green)	Ready for operation
NC (yellow)	Signoflife monitoring
CF (yellow)	Reading from/writing to CF card

Reference

You can find information on error description in SINUMERIK 802D sl, Diagnostics Manual

1.3 Key definition of the full CNC keyboard (vertical format)



DEL — Clear key

INSERT — Insert key

TAB — Tab stop

INPUT — ENTER / input key

POSITION — POSITION operating area key (position operating area)

PROGRAM — PROGRAM operating area key (program operating area)

OFFSET PARAM — OFFSET PARAM operating area key (parameter operating area)

PROGRAM MANAGER — PROGRAM MANAGER operating area key (program manager operating area)

SYSTEM ALARM — SYSTEM/ALARM operating area key (system/alarm operating area)

CUSTOM

ETC key

Recall key

NEXT WINDOW — Not assigned

ALARM CANCEL — Acknowledge alarm key

PAGE UP / PAGE DOWN — Scroll keys

1...n CHANNEL — Without function

HELP — Info key

END

SHIFT — Shift key

SELECT — Selection key / toggle key

Cursor keys

CTRL — Control key

Blank (SPACE)

A J / W Z — Alphanumeric keys
Double assignment on the shift level

ALT — ALT key

BACKSPACE — Delete key (backspace)

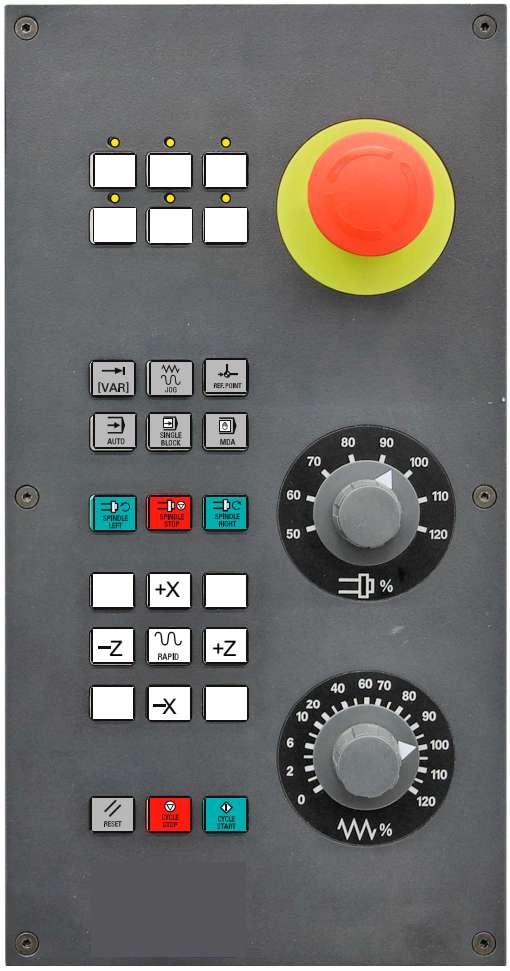
(9 /) 0 — Numeric keys
Double assignment on the shift level

Hot keys

In the part program editor and in the input fields of the HMI, the following functions can be carried out with certain key combinations on the full CNC keyboard:

Keystroke combination	Function
<CTRL> and <C>	Copy selected text
<CTRL> and <B>	Select text
<CTRL> and <X>	Cut selected text
<CTRL> and <V>	Paste copied text
<ALT> and <L>	Changeover to small letters
<ALT> and <H> or <HELP> key	Call help system

1.4 Key definition of the machine control panel



RESET — RESET

CYCLE STOP — CYCLE STOP (NC STOP)

CYCLE START — CYCLE START (NC START)

EMERGENCY STOP

Spindle Speed Override
Spindle override

User-defined key with LED

User-defined key without LED

[VAR] — INCREMENT
Increment

JOG — JOG

REF.POINT — REFERENCE POINT
Reference point

AUTO — AUTOMATIC

SINGLE BLOCK — SINGLE BLOCK
Single block

MDA — MANUAL DATA
Manual input

SPINDLE LEFT — SPINDLE START CCW
Counterclockwise

SPINDLE STOP — SPINDLE STOP

SPINDLE RIGHT — SPINDLE START CW
Clockwise

RAPID — RAPID TRAVERSE OVERLAY
Rapid traverse override

+X -X — X axis

+Z -Z — Z axis

Feedrate override
Feedrate control

Note

This documentation assumes an 802D standard machine control panel (MCP). Should you use a different MCP, the operation may be other than described herein.

Software interface

2

This Programming and Operating Manual focuses primarily on the software interface of the "Manual Machine Plus" system.

For descriptions of the software interface for the SINUMERIK 802D sl control system, please refer to the SINUMERIK 802D sl Turning Programming and Operating Manual.

See Sections:

- Screen layout
- Standard softkeys
- Operating areas
- Help system

Turning On, Reference Point Approach

3

3.1 Entry to the "Manual Machine Plus" operating area

Operating sequences

Note

The operating area "Manual Machine Plus" runs only in Siemens mode, and not in ISO mode.

Proceed as follows to open the "Manual Machine Plus" application:

Note

If the controller has already been preconfigured to "Manual Machine Plus" by the machine manufacturer, items 1 to 3 can be ignored in the following description. The operating area "Manual Machine Plus" is activated once the controller has been started up.

If you have not yet executed a reference point approach, you will be in the operating mode Reference point approach (see Fig. Reference point approach) after start-up.

1. The NC boots into the JOG REF operating mode of the NC basic machine.

2. Select the JOG operating mode.

   The interface can be seen in the screenshot below:

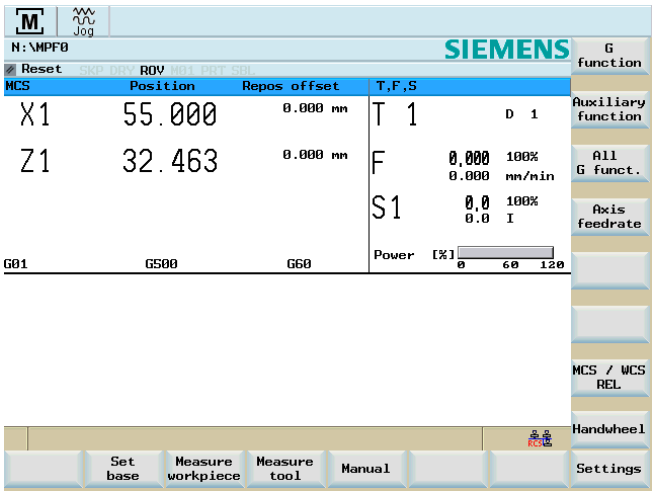


Figure 3-1 "Position" operating area

Manually

3. You can access the "Manual Machine Plus" area by clicking on softkey "Manual":

Note

If you have not yet executed a reference point approach, the JOG REF operating mode will be reselected automatically when you press the "Manual" softkey.

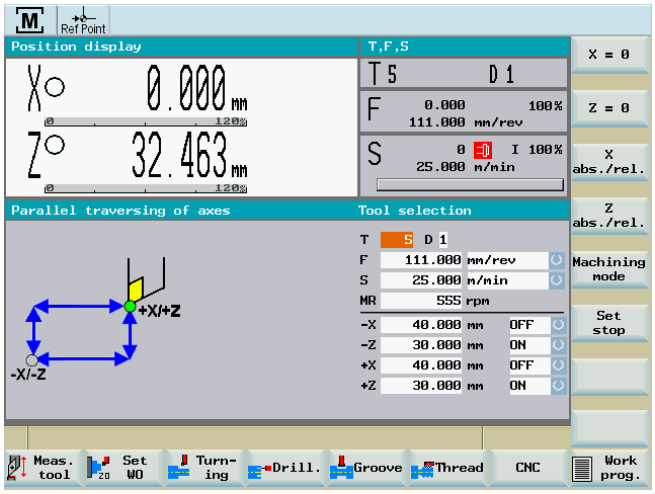


Figure 3-2 Reference point approach

CNC

4. The operating area "Manual Machine Plus" can be exited with the "CNC" softkey, after which the operating area of the NC basic machine is active again.

Note

The manual machine cannot be exited in the "Execute cycles" and "Execution window" screen forms of the stepping program.

General operating information

If a screen (e.g. for the "thread tapping" function) contains input fields, then you can edit them as follows:

- Select the input fields using the <cursor keys>.
- Enter data using the <numeric keys>.
- Accept the data using the <INPUT key>.
- Use the <Tab key> to switch between the input fields and the selection box during the opening and saving of files.
- If you do not want to accept an input value,then exist the corresponding field using <Cursor key> or change the screen form. Do not press the <INPUT> key.

Note

The pictures in the parameterization screenforms depend on the setting of the machine data by the machine manufacturer, i.e. display of the tool position before or behind the center of rotation with regards to the turret head.

3.2 Reference point approach

Functionality

The axes have not yet approached their reference points (see screenshot below).

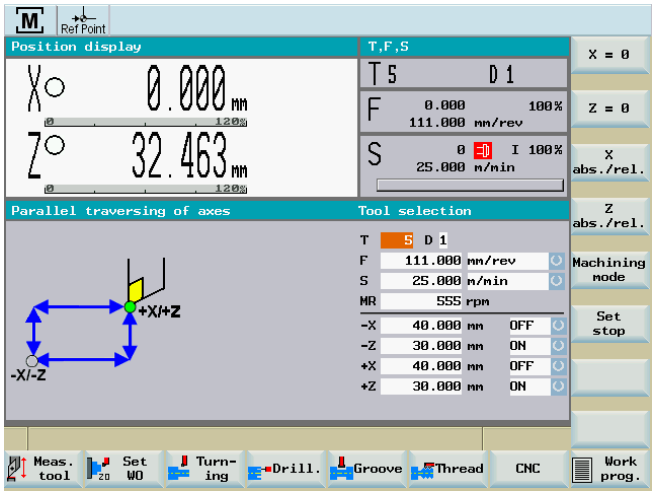


Figure 3-3 Reference point approach

Note

The meanings of the symbols in the axis display are as follows:

○ -> Axis still needs to be referenced.

◐ -> Axis is referenced.

Requirement

The following requirements must be fulfilled:

- There must be no NC alarms pending!

  -> Clear any that are pending by pressing the "Reset" softkey.

- 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 in front of the reference point.

⚠ DANGER

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

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 cam before the start of the reference point approach.

Operating sequence

1. Select the <JOG> operating mode.

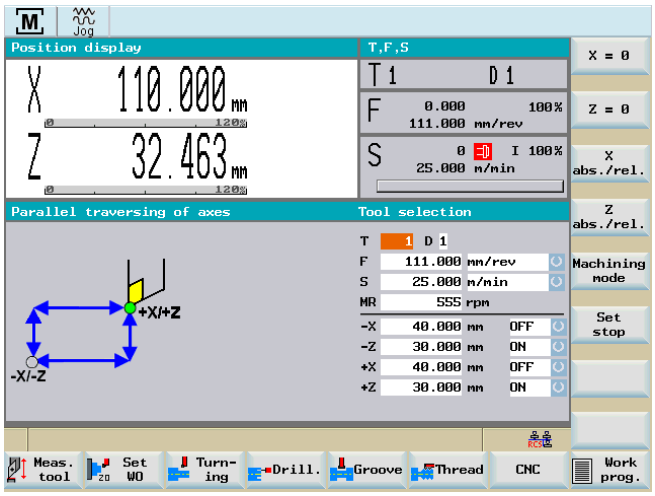


Figure 3-4 Main screen for "Manual Machine Plus"

2. Select handwheel increment weighting using the <INCREMENT> key.

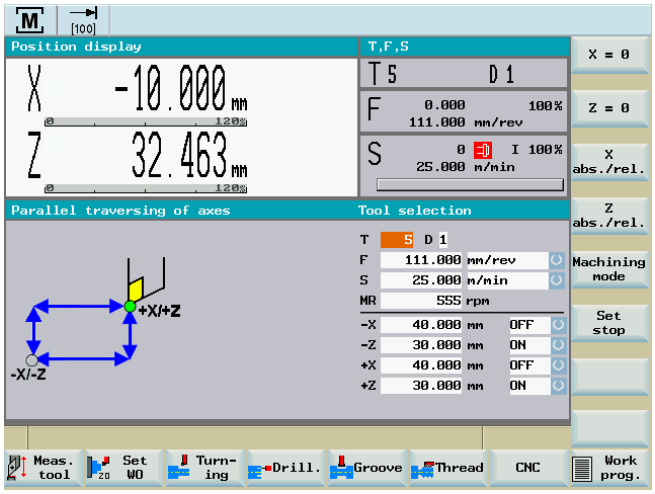


Figure 3-5 Handwheel increment weighting 100 INC

The current setting will appear on the top left of the screen (e.g.: 100 INC).

3. 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 state, the axes can be moved only by means of the handwheel. Traversing the axes using the axis traversing switch is inhibited. The spindle cannot be started in this operating state.

4. Select the <JOG REF> operating mode.

5. Using the axis direction key, start the X axis in the positive direction (X+).

   The reference point for the X axis is then approached automatically.

   The axis stops automatically when it has completed this process.

   The symbol appears in front of the axis name in the display.

6. Repeat step 5 for the Z axis.

> **NOTICE**
>
> The axes must be referenced in this sequence, i.e., the X axis first, followed by the Z axis. No other sequence will be accepted by the controller.

You have now referenced both axes.

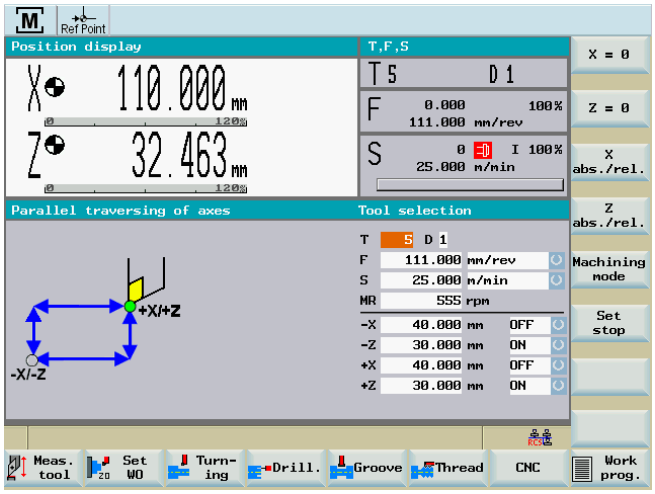


Figure 3-6 Reference points reached

7. Select the <JOG> operating mode.

   You are in the "Manual Machine Plus" operating area.

Setting-up

4

4.1 Measuring tools

Functionality

You can measure tools manually in the "Manual Machine Plus" operating area. In this case, the manual tool measurement function accesses the tool list data.

Note

You can access the tool list by pressing the operating area key <OFFSET/PARAM> and softkey "Tool list".

Literature

Further methods of handling tools and tool offsets are described in the "SINUMERIK 802D sl Turning Programming and Operating Manual".

⚠ DANGER

Notice: 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.

Requirement

Load the tool beforehand or enter the tool number in the "T" field. After confirming the input, a dialog will prompt you to press the <NC Start> key. If the <NC Start> key is pressed, the tool will be changed.

⚠ CAUTION

First approach a machine position where the tool change can be performed without danger.

Operating sequences

Proceed as follows to measure the tool for the X axis of the loaded turning tool.

Meas. tool

1. Press the "Meas. tool" softkey. The following screen appears:

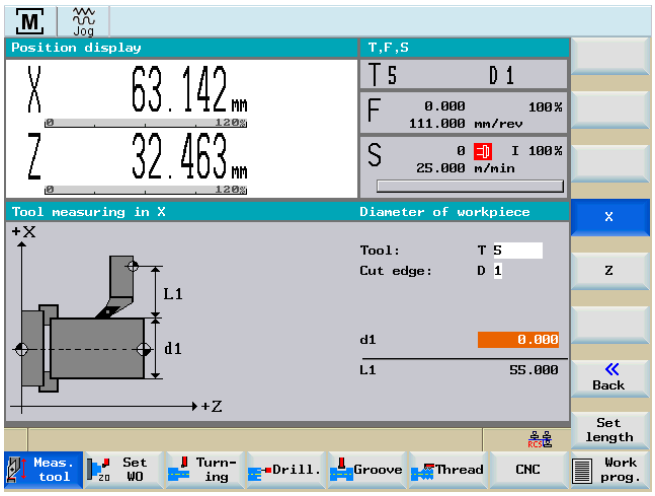


Figure 4-1 Measure a turning tool

X

2. Press the "X" softkey.

   The screen for measuring the X axis (L1) appears.

3. Check that the current tool number appears in the display field for the tool, since the calibration operation will relate to this tool.
4. Carefully "scratch" a workpiece by an X handwheel infeed when the spindle is turning.
5. Move the slide slightly (without changing the X position) along the Z axis (longitudinal turning) with the handwheel.
6. Switch the spindle off.
7. Enter the diameter measured on the workpiece in entry field "d1".

INPUT

8. Accept the value by pressing the <INPUT> key.

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

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

Set length

9. Press the "Set length" softkey.

   The modified tool offset for the selected tool is applied 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 position display of the tool measurement screen.

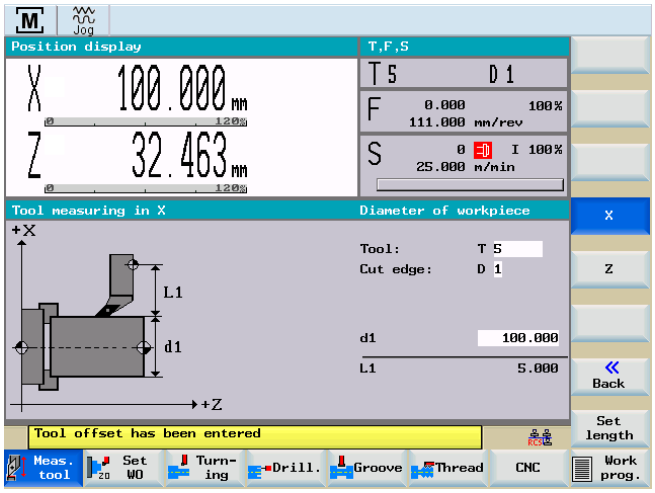


Figure 4-2 Measurement of turning tool in X axis completed

Z

10. Press the "Z" softkey.

    The screen for measuring the Z axis appears.

    The Z axis can be measured in the same way as the X axis.

    When measuring the tool in the Z axis, you may define a distance between the workpiece and the turning tool tip in input field "a1" to avoid surface damage on the workpiece.

X Abort

11. To return to the main screen for "Manual Machine Plus", select softkey "Abort".

4.2 Limit stops

Functionality

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.

By setting the limit stops, in the "Manual Machine Plus" operating area, it is possible to turn simple shoulders (including tapers) without the need for any further cycle parameterization.

Supplementary conditions

- The limit stop position is always an absolute dimension, which in turn always corresponds to the position in the absolute actual value display on the "Manual Machine Plus" interface. A relative limit stop position is not possible.
- A limit stop position can be entered/accepted only when the axes are stationary. Otherwise, an error message appears.

4.2.1 Setting and activating/deactivating limit stops

Functionality

In the main screen "Manual Machine Plus" limit stops can be entered into the input fields "-X/-Z/+X/+Z".

In the following screen, the cursor (the field has a dark background) is located in the input field "Endstop of axis X+".

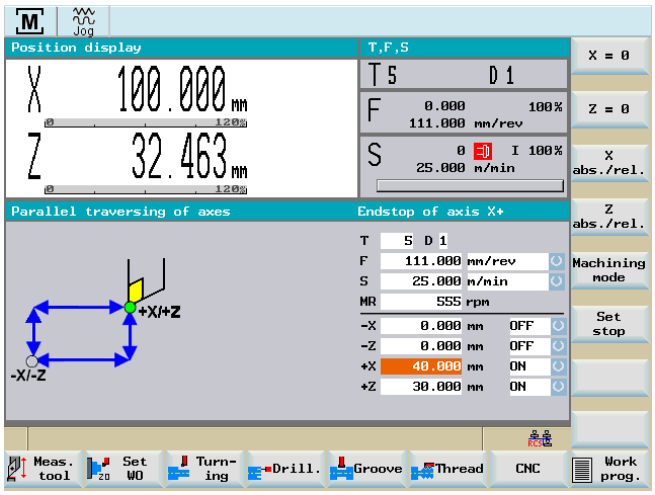


Figure 4-3 Endstop of axis X+

Parameter

Parameter	Description
ON	The limit stop is activated.
OFF	The limit stop is deactivated.
- X	Negative absolute position of the limit stop of the X axis. The axis stops automatically if: • The limit stop is active. • The specified axis traverses in the negative direction and reaches the absolute limit stop position.
+X	Positive absolute position of the limit stop of the X axis. The axis stops automatically if: • The limit stop is active. • The specified axis traverses in the positive direction and reaches the absolute limit stop position.
-Z	Negative absolute position of the limit stop of the Z axis. The axis stops automatically if: • The limit stop is active. • The specified axis traverses in the negative direction and reaches the absolute limit stop position.
+Z	Positive absolute position of the limit stop of the Z axis. The axis stops automatically if: • The limit stop is active. • The specified axis traverses in the positive direction and reaches the absolute limit stop position.

All of the limit stops are set in the following screen.

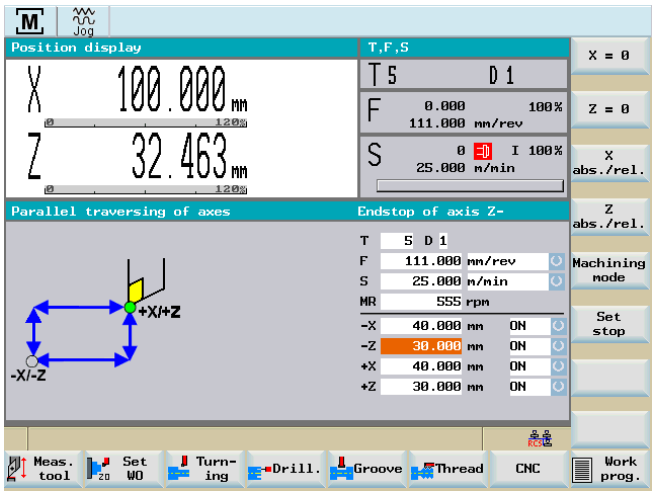


Figure 4-4 All limit stops set

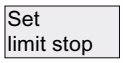
Operating sequences

You can use the following methods to enter a limit stop position:

- Direct position entry:
  - Select the input field of the relevant limit stop with the <Cursor keys>.
  - Now use the <Numeric keys> to enter the absolute position you require.
  - Press the <INPUT> key to accept the value.

- Accepting the current actual position:
  - Select the input field of the relevant limit stop with the <Cursor keys>.
  - Traverse to the required position using the axis direction switch (e.g. <-Z> or <+X/-X/+Z>).
  - Press the "Set limit stop" softkey.

    The current actual position of the relevant axis is transferred to the input field.



Activating/disabling limit stops

The limit stops are activated/de-activated individually using the <SELECT> key.

You can select between ON and OFF.

4.2.2 Turning against a stop

Example:

The following example explains the operating principle of limit stops using the axis direction keys.

You can also use the handwheel to perform the machining operation.

Task

The following shoulder with a finishing allowance of 0.2 mm must be turned:

- 100 mm in the Z direction
- 50 mm final diameter in the X direction

The end face starts at 0 mm int he Z direction. The blank diameter is 70 mm.

Operating sequences for infeeding to stop

1. Position the axes in front of the workpiece (e.g., X +75 mm/Z +5 mm).
2. Check the machining technology data.
3. Set the following limit stops:
   - -X at 50.4 mm
   - -Z at –99.8 mm (due to finishing allowance)
   - +Z at +5 mm
4. Delete the limit stop for +X; it is not required.
5. Start the spindle.
6. Using the handwheel, infeed to the 1st depth of cut in the X direction.
7. Start machining in the Z axis in the negative direction using the axis direction switch.

   When the limit stop position in Z –99.8 mm is reached, the Z axis stops automatically.

   The message "Limit stop –Z reached" is displayed.
8. Switch-out the axis direction switch.
9. Using the handwheel, retract the tool from the workpiece in the X direction.
10. Using the axis direction switch and rapid traverse override, move the tool in a positive Z direction towards the workpiece until the axis stops.

    The message "Limit stop +Z reached" is displayed.
11. Switch-out the axis direction switch.

12. Using the handwheel, infeed to the next depth of cut in the X direction.
13. Start machining in the Z axis in the negative direction using the axis direction switch.

    Repeat the procedure until the depth of rough cut is reached.

    The message "Limit stop -X reached" is displayed as the tool is fed in.

    Once this cut has been completed, adjust the limit stops to the finished dimension, provided that the axes are positioned in front of the workpiece.

Operating sequences for adjusting to finished dimension

1. Adjust the limit stops to the finished dimension: -X to 50.0 mm/-Z to –100.0 mm
2. Using the handwheel, infeed in the X direction until the "Limit stop -X reached" message appears.
3. Start machining in the Z axis in the negative direction using the axis direction switch.

   When the limit stop position in Z –100.0 mm is reached, the Z axis stops automatically.

   The message "Limit stop –Z reached" is displayed.
4. Switch-out the axis direction switch in the Z direction and start in the positive X direction (finishing the end face).
5. Switch-out the axis direction switch in the X direction as soon as the tool tip leaves the workpiece.

4.3 Setting the workpiece zero

Functionality

The “Set the workpiece zero" function can be used to specify the reference point for machining the workpiece.

Typical application/procedure:

1. Parameterize all the machining steps (cycles) for the workpiece in relation to a “virtual zero point” (e.g., an end face).
2. Clamping the blank
3. Scratch the relevant surface which corresponds to the "virtual zero point".
4. Use the "Set WO" function, adapt the workpiece coordinate system to the parameterized machining operation.

   Make sure that the axis does not exit from the approached position.

Additional information

The following operations are performed automatically when you select softkey "Set WO":

- The work offset is automatically calculated according to the current axis position in the longitudinal axis (Z), entered in the NC memory for the basis offset and activated.
- 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 is reset, the value “0.000” will automatically be entered in the NC memory for the basic offset. The workpiece coordinate system display will change to reflect this.

⚠DANGER
Notice: Setting the "workpiece zero" 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 without due care and attention can result in serious damage to the tool, workpiece or machine!!

Operating sequences

Set WO

Press the "Set WO" softkey in the main screen for "Manual Machine Plus".

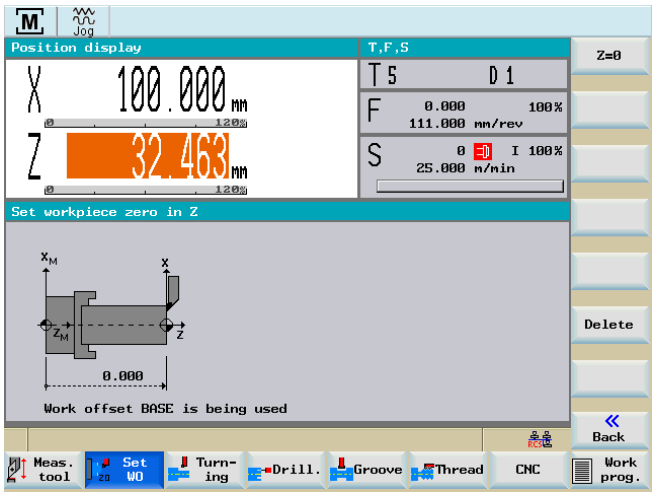


Figure 4-5 Set workpiece zero point

This screen displays the currently programmed Z value of the basic work offset.

The setting options in this screen are selected with softkeys. The softkey meanings are as follows:

Z = 0

This function is used to set the “workpiece zero”.
The workpiece coordinate system of the longitudinal axis (Z) displays the value "0.000".

The required work offset is computed automatically and stored in the appropriate place in the NC.

Delete

This function resets the work offset that is currently stored on the NC. -> The value “0.000” is entered in the basic offset memory location. However, all other offsets and the active tool offset remain unchanged.

Manual machining

5

5.1 Fundamentals of manual machining

Note

Please refer to the SINUMERIK 802D sl operating instructions for a description of the relevant commissioning requirements.

Functionality

You can perform the following machining operations manually:

- Axis-parallel traversal
- Taper turning
- Radius turning
- Drilling - centered
- Tapping
- Groove cycles/Parting
- Thread cutting
- Rough turning of contours

Fundamentals

The following operations must be performed before manual machining can proceed:

- Axes referenced
- Tools measured
- Limit stops set
- Set workpiece zero point

5.2 Display and operator control optios in the main screen for "Manual Machine Plus"

Functionality

Note

If the controller has already been preconfigured to "Manual Machine Plus" by the machine manufacturer, the operating area "Manual Machine Plus" is activated once the controller has been started up. If you have not yet executed a reference point approach, you will be in the operating mode Reference point approach after start-up.

You can reference the axes in the Siemens standard user interface as well as in the operating area "Manual Machine Plus".

You have referenced the axes and pressed the "Manual" softkey in the "Position" operating area. The following screen represents the main screen of the "Manual Machine Plus" operating area.

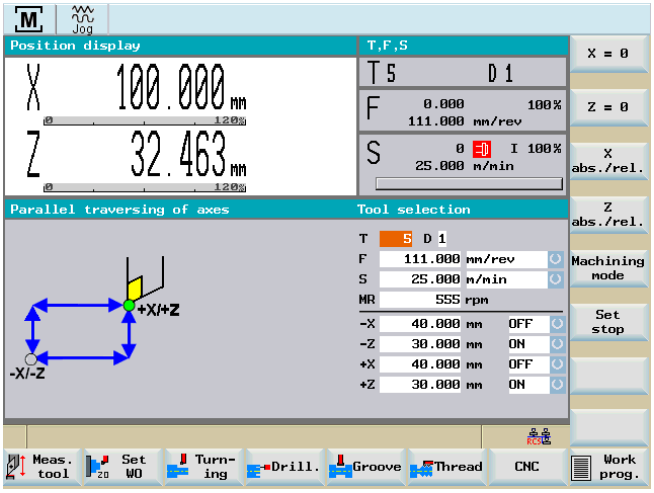


Figure 5-1 Main screen for "Manual Machine Plus", position display absolute

Note about the position display in the main screen for "Manual Machine Plus"

- Absolute position display active:

  The position value displayed in the large-size font is the absolute position. No additional value is shown.

- Relative position display active (see following diagram):

  The position value displayed in the large-size font is the relative position. The position value displayed next to it in the small-size font is the absolute position.

Controlling the axes and spindle

In manual machining mode, the axes and spindle can be controlled by the following methods:

- The compound slide rest is controlled by:
  - Handwheels (Page 39) for the X and Z axes, or
  - Axis direction switch (Page 39)
- The spindle is controlled by:
  - Spindle direction of rotation switch (Page 40)

Values displayed in the position display and T, F, S

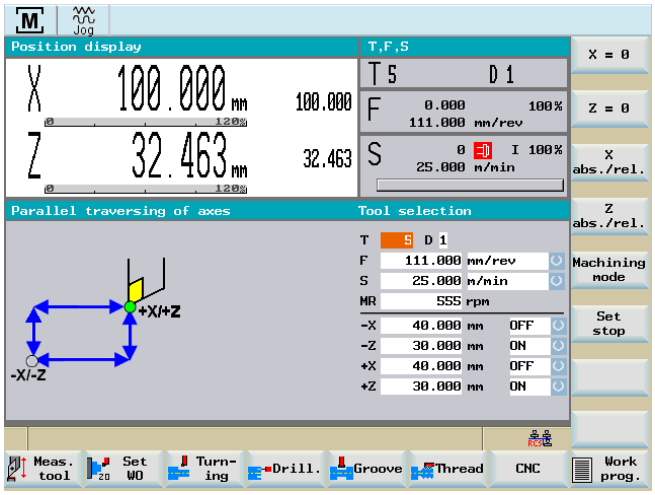


Figure 5-2 Main screen for "Manual Machine Plus", position display relative

Displayed values	Meaning
In front of axis letters +/-	• Current traversing direction of axes
S value/S type %	• The programmed value for either the spindle speed (rev/min) or the cutting rate (m/min) The display corresponds to the settings for the machining technology data. • Current position of the spindle override switch in %.
F value/F type %	• Programmed feed value in either "m/min" or "mm/Rev", depending on the settings for the machining technology data. • Current position of the feedrate override switch in %.
T value	• Tool number of the tool used
D value	• Tool offset applied
INC value	• Handwheel pulse weighting setting

Displayed values	Meaning
	• Feed stop as a result of: – Feedrate override at position 0%. – An alarm is active which prevents the axes from moving.
	• Spindle status – Spindle counter-clockwise – Spindle stop – Spindle clockwise

Machining and technology data

You must enter the machining technology data in the following input fields:

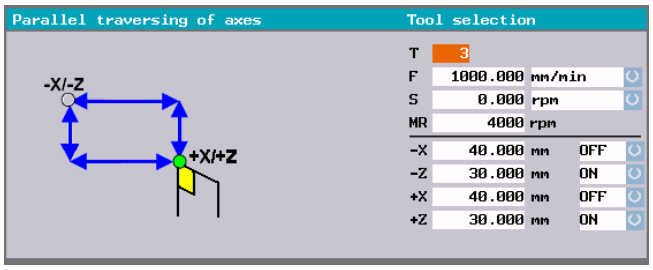


Figure 5-3 Display box from main screen "Manual Machine Plus", entry of machining technology data

The machining technology data are as follows:

Parameter	Description
T	Tool number of the used tool (only for use of a manual tool-changer system)
F	Feedrate with choice of units mm/min (time feed) and mm/rev (revolutional feed),
S	Spindle type with choice of units rev/min (constant spindle speed) and m/min (constant cutting rate),
MR	Speed limitation for constant cutting rate
-X -Z +X +Z	Positions of the limit stops, the limit stops can be activated using the toggle field "ON/OFF"

⚠ CAUTION

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

⚠ DANGER

Notice: When constant cutting rate (G96) is selected, the maximum permissible spindle speed, corresponding to the fitted tool chucking device must be entered in the input field MR (spindle speed limitation)!

Failure to pay sufficient attention to this point can lead to serious damage as a result of the chucking device speed being exceeded.

5.2.1 Toggling the display

Functionality

In the position display screen you can edit the displayed values using the vertical softkeys.

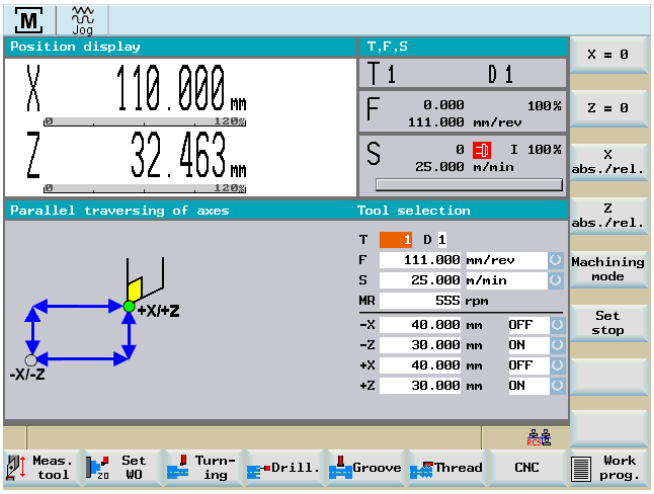


Figure 5-4 Main screen for "Manual Machine Plus"

Softkeys

X = 0	Change the display to "relative position display" and "reset" the display in the X axis.
Z = 0	Change the display to "relative position display" and "reset" the display in the Z axis.
X abs./rel.	Toggle the display between "absolute position display" and "relative position display" in the X axis.
Z abs./rel.	Toggle the display between "absolute position display" and "relative position display" in the Z axis.
Machined type	Switching between the following operating modes: • Traversing the axes parallel to the axis • Taper turning • Radius turning The parameters for the machining type are displayed in the main screen of the "Manual Machine Plus".
Set limit stop	The current actual position of the relevant axis is transferred to the selected input field (-X/-Z/+X/+Z).

5.2.2 Machining with the handwheels

Functionality

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 axis direction switch is in the zero position or the individual axis control keys are disabled.

The distance traversed per handwheel pulse depends on the increment weighting setting.

⚠ CAUTION
If the handwheel increment weighting is set to "0" or if the feedrate override weighting is in the "0" position, the handwheels are disabled.

5.2.3 Setting the increment weighting for the handwheel

Functionality

Set the increment weighting from the Increment weighting machine control panel.

If you are unable to adjust the increment weighing, it will be because the controller's internal mode is incompatible with this process.

Press the <JOG> key once to resolve the problem.

⚠ DANGER
Notice: An incorrect increment weighting setting can result in damage to the workpiece, tool and machine!

5.2.4 Machining with axis direction switch

Functionality

You can move the axes in the desired direction by changing over the axis direction switch.

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 weighting setting and, depending on the option selected in the machining technology screen (revolutional feed/cutting speed), by the spindle override weighting.

If the <Rapid traverse override> key is also pressed, the axis is moved at the maximum possible speed, unless the feedrate override weighting setting is used to specify a different value.

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

5.2.5 Spindle advance/reverse

Functionality

CAUTION
Start spindle The spindle value should be checked before starting the spindle (e.g. when changing the tool). The last value set is active (this depends on the machinery construction OEM).

You start the spindle in the appropriate direction (spindle advance / reverse) by changing over the spindle direction switch:

⚠ CAUTION
The spindle cannot be started, unless the chuck guard switch is enabled. -> Close the chuck guard!

⚠ DANGER
Notice: Do not alter or adjust the chuck guard/chuck guard switch.

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.

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

5.2.6 Tool change

Functionality

A basic differentiation must be made between a manual and an automatic tool-changer system.

For an automatic system, the tool change is controlled by the PLC user program. The currently loaded tool is displayed in the "Manual Machine Plus" main screen.

For a manual system, the required tool number is manually entered from an input screen form.

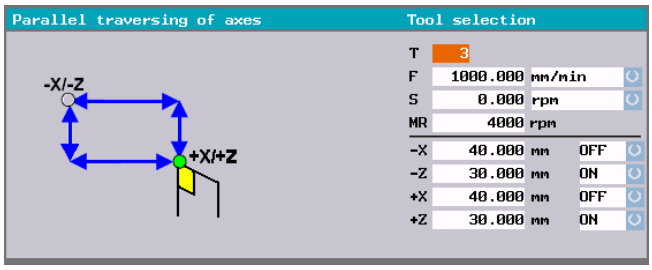


Figure 5-5 Extract from main screen "Manual Machine Plus", entry of tool number

Operating sequences

Follow the sequence of operations below to enter the required tool number:

1. Move the cursor onto the input field for the T value.
2. Enter the tool number using the numeric keys

   (the tool you wish to select must be set up in the tool list!).

3. Accept the tool number by pressing the <INPUT> key.

   The following information text with the corresponding tool number is displayed:

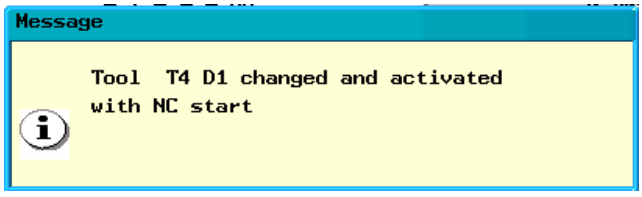


Figure 5-6 Tool change with NC Start

Acknowledge this information text using the "Cancel" softkey.

4. Press the <NC-Start> key.

The tool change is changed.

Please note the following for a manual tool change:

- The real tool change on the machine (tool relocation) is finished.
- The appropriate tool number (tool offset) must be communicated to the control by making a manual entry.

⚠ CAUTION
A new tool number may be selected only if all axes and the spindle are stationary.

⚠ DANGER
Notice: The tool number entered in the T value field must correspond to the tool loaded into the machine! Otherwise, the tool will have to be recalibrated (see also section "Measuring tools"). An uncalibrated or incorrectly calibrated tool can lead to dimensional errors or to incorrect cutting values.

5.2.7 Changing the feedrate/spindle value

Changing the operating sequence, feed rate "F"/ spindle value "S"

Follow the sequence of operations below to enter the required feedrate or spindle value:

1. Position the cursor on the input field for the value (see screenshot below) in the main screen for "Manual Machine Plus".

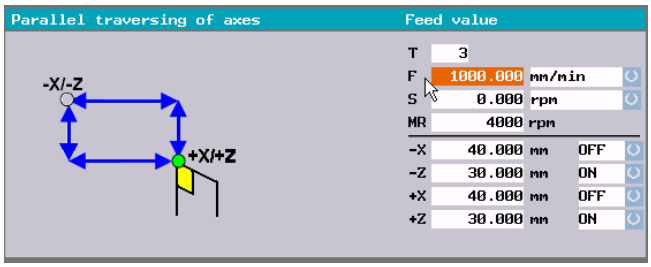


Figure 5-7 Extract from main screen "Manual Machine Plus", entry of feedrate value "F"

2. Edit the programmed value using the numeric keys.

3. Press the <INPUT> key.

   The value is activated.

⚠ CAUTION
The F value (feed rate) or the S value (spindle) can only be changed if all axes and the spindle are stationary.

5.2.8 Changing the feedrate/spindle type

Changing the operating sequences feedrate type "F"

By pressing the <Cursor keys>, you go to the display field which contains the currently programmed feedrate type (on dark background).

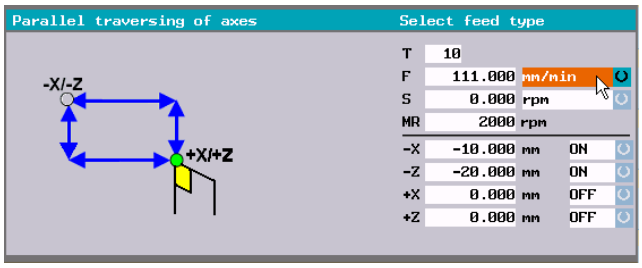


Figure 5-8 Type of feedrate

By pressing the toggle key <SELECT>, you can choose one of the following feedrate types:

- Time feed (mm/min)

  If time feed is selected, the axes are moved at the speed entered in this field (mm/min) (unless rapid traverse override is activated). It can be influenced by the feedrate override switch setting.

  The time feed is only possible for a constant spindle speed.

- Rotary feedrate (mm/rev)

  In "Spindle speed + revolutional feedrate" or "Const. cutting speed + revolutional feedrate" mode, the value entered in this field determines the axis speed (unless rapid traverse override is activated). It is influenced directly by the feedrate override weighting setting and indirectly by the spindle override weighting setting.

Changing the operating sequences spindle type "S"

By pressing the <Cursor keys>, you go to the display field which contains the currently programmed spindle type (on dark background).

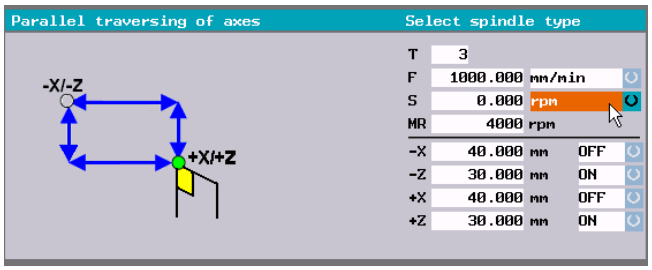


Figure 5-9 Spindle type

By pressing the toggle key <SELECT>, you can choose one of the following spindle types:

- Constant spindle speed (rpm)

  This value defines the programmed spindle speed for machining with "Spindle speed + Time feed" or "Spindle speed + Revolutional feedrate".

  The constant spindle speed is achieved only if no speed reduction is programmed by means of spindle override weighting or with spindle setting data.

- Constant cutting rate (m/min)

  Cutting speed input value for machining with "Cutting speed + revolutional feedrate". The spindle speed is adjusted to the machining diameter of the workpiece so that uniform cutting conditions are achieved.

  Since the spindle would (in simple mathematical terms) have to rotate at an "infinitely high" speed at the rotational center point in this mode, this speed is limited in the spindle setting data by the input value "MR".

  The constant cutting speed can also be influenced by means of the feedrate and spindle override weighting settings.

⚠ CAUTION
The feedrate or spindle type can only be changed if all axes and the spindle are stationary.

5.2.9 Change the speed limitation for constant cutting rate

Change the speed limitation operating sequences

When a constant cutting rate (G96) is programmed, the maximum permissible spindle speed, corresponding to the fitted tool chucking device must be entered in the input field "MR" (spindle speed limitation)!

⚠ DANGER

Spindle speed limitation

Failure to pay sufficient attention to this point can lead to serious damage as a result of the chucking device speed being exceeded.

1. Position the cursor on the input field for the value in the "Manual Machine Plus" main screen.

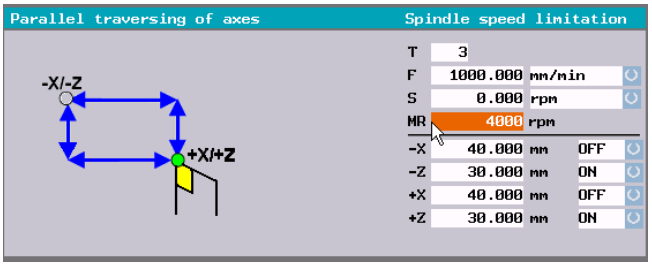


Figure 5-10 Speed limitation input

2. Edit the programmed value using the <Numeric keys>.
3. Press the <INPUT> key.

The value is activated.

⚠ CAUTION

The value may be changed only when all axes and the spindle are stationary.

5.3 Manual machining with machining types

5.3.1 Axis-parallel traversal

Functionality

The axis-parallel traversal is used for the simple cutting on the workpiece or for positioning the axes.

If you move the axis direction switch, the control then moves the X and Z axes accordingly.

Operating sequences

1. You can access the "Parallel traversing of axes" function via the main screen "Manual Machine Plus".

Machined type

2. If a different machining mode is active, press the "Machining mode" softkey until "Parallel traversing of axes" is displayed.

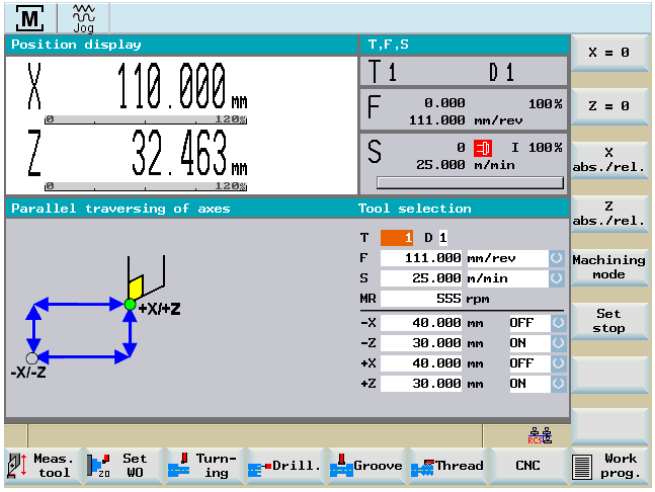


Figure 5-11 Main screen for "Manual Machine Plus"

5.3.2 Manual taper turning

Functionality

The "Manual taper turning" function is intended for the simple production of tapered workpieces.

For the machining type "Taper turning" you need to enter an angle (taper angle α). The angle input rotates the controller's internal coordinate system according to the angle value.

When you move the axis direction switch, the controller then uses the angle input to interpolate (and simultaneously traverses) the X and Z axes 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.

NOTICE
The desired taper is traversed only by means of an axis direction switch or axis direction keys of the machine control panel depending on the machine equipment. A traversal using the handwheels is not possible!

Operating sequences

1. You can access the "Manual taper turning" function in the main screen for "Manual Machine Plus".

Machined type

2. Press the "Machining mode" softkey until "Taper turning" is displayed.

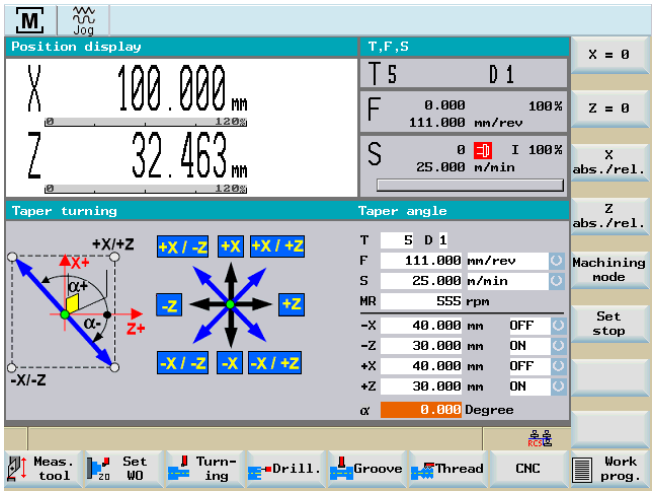


Figure 5-12 Taper turning

3. The input field for the taper angle "α" is immediately displayed on a dark background when the machining mode is selected. You must enter the angle using the <Numeric keys>.

   A positive angle value rotates the coordinate system in traverse direction X+.

   A negative angle value rotates the coordinate system in traverse direction X-.

4. The entered value is immediately accepted using the <INPUT> key.

The taper angle remains active until you exit the "Taper turning" by pressing the "Machining mode" softkey.

5.3.3 Manual radius turning

Functionality

The "Manual radius turning" function is designed to simplify the machining of inside and outside radii.

The positions of the axes at the time that machining is selected form the starting point for the radii to be traversed.

When you move the axis direction switch, the control then uses the input values to interpolate (and simultaneously traverse) the X and Z axes accordingly.

The programmed axis feed then applies to the path being traversed and not to the corresponding axis.

NOTICE
The desired radius is traversed only by means of an axis direction switch or axis direction keys of the machine control panel depending on the machine equipment. A traversal using the handwheels is not possible!

Operating sequences

1. You can access the "Manual radius turning" function in the main screen for "Manual Machine Plus".

Machined type

2. Press the "Machining mode" softkey until "Taper turning" is displayed.

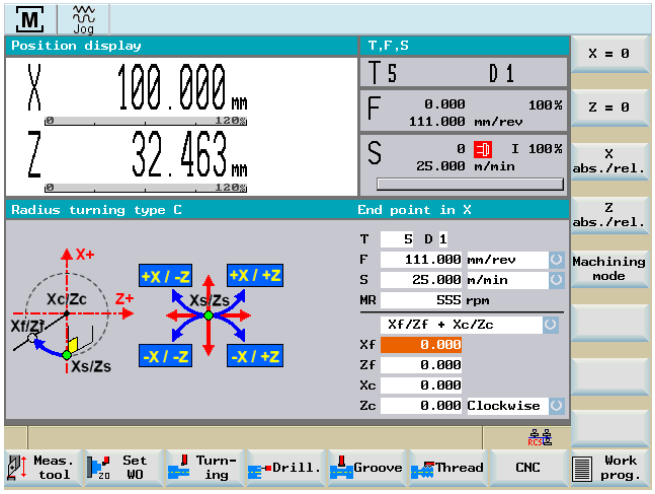


Figure 5-13 Radius turning

The "Radius turning" can be exited by pressing the "Machining mode" softkey.

The three types available for radius turning differ in the specification of the values for specifying the radius.

- Type A
- Type B
- Type C

3. By pressing the <Cursor keys> you can go to the display field which contains the active radius type (on dark background).

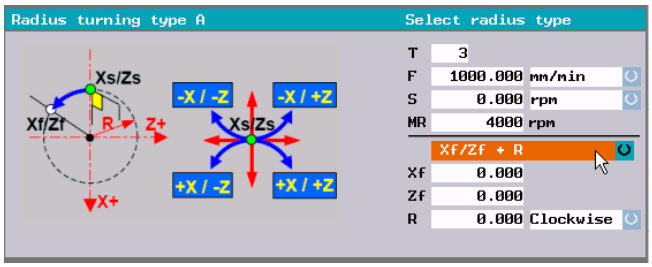


Figure 5-14 Radius turning type A

SELECT

4. By pressing the toggle key <SELECT>, you can select the radius type.

⚠DANGER

Notice: Omitting or using the wrong sign for the input values or entering the wrong arc direction can lead to a collision and may destroy the tool or the workpiece.

Note

Any limit stops that are activated should be disabled before starting radius turning or set to a value outside the traversing range needed for radius turning. Otherwise an error message is transmitted by the controller which prevents the "Manual radius turning" function being started.

5.3.3.1 Radius turning type A

For the radius turning type A, the radius to be machined is specified by the end point, the radius and the machining direction.

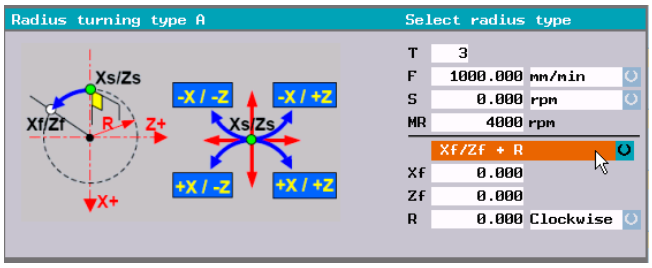


Figure 5-15 Radius turning type A

Parameter

Parameter	Description
Xf	This input value describes the position of the circle end point in the X axis. The input value is evaluated as absolute position (in the diameter).
Zf	This input value describes the position of the circle end point in the Z axis. The input value is evaluated as absolute position.
R	This input value describes the radius to be traversed.
Counterclockwise/Clockwise	This toggle field selects whether a circle must be traversed in the clockwise or counterclockwise direction.

5.3.3.2 Radius turning type B

For the radius turning type B, the radius to be machined is specified by the center, the radius, the opening angle and the machining direction.

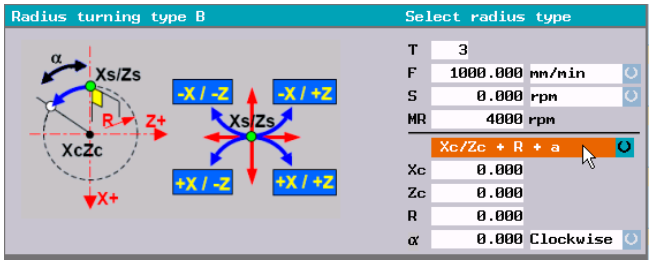


Figure 5-16 Radius turning type B

Parameter

Parameter	Description
Xc	This input value describes the position of the circle center in the X axis. The input value is evaluated as absolute position (in the diameter).
Zc	This input value describes the position of the circle center in the Z axis. The input value is evaluated as absolute position.
R	This input value describes the radius to be traversed.
α	This input value describes the opening angle of the radius to be traversed.
Counterclockwise/Clockwise	This toggle field selects whether a circle must be traversed in the clockwise or counterclockwise direction.

5.3.3.3 Radius turning type C

For the radius turning type C, the radius to be machined is specified by the center, the end point and the machining direction.

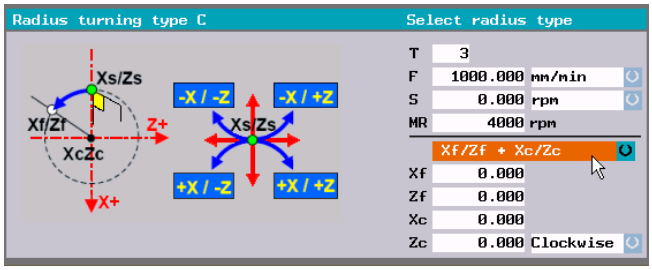


Figure 5-17 Radius turning type C

Parameter

Parameter	Description
Xf	This input value describes the position of the circle end point in the X axis. The input value is evaluated as absolute position (in the diameter).
Zf	This input value describes the position of the circle end point in the Z axis. The input value is evaluated as absolute position.
Xc	This input value describes the position of the circle center in the X axis. The input value is evaluated as absolute position (in the diameter).
Zc	This input value describes the position of the circle center in the Z axis. The input value is evaluated as absolute position.
Counterclockwise/Clockwise	This toggle field selects whether a circle must be traversed in the clockwise or counterclockwise direction.

5.4 Manual machining using cycles (functions)

5.4.1 Principle operating sequence

Functionality

You can perform the following functions manually:

- Drilling centric
- Tapping
- Groove cycles/Parting
- Thread cutting
- Rough turning of contours

When manually machining these functions, the operating sequence is essentially executed in the same way.

Requirement

The following requirements must be fulfilled before you can execute the functions:

- The spindle is rotating in the correct direction.

Note

Parts can be machined manually only by a spindle started in the correct rotational direction. An error message is otherwise displayed.

⚠ DANGER

Notice: 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!

- Any axis position from which the workpiece position to be machined can be approached without risk of collision.
- All parameters for the cycles are correctly assigned.

Operating sequences

Drilling

Tapping

1. Select the function (e.g. "Drilling " > "Tapping") in the main screen of the "Manual Machine Plus".
2. Parameterizing the function.

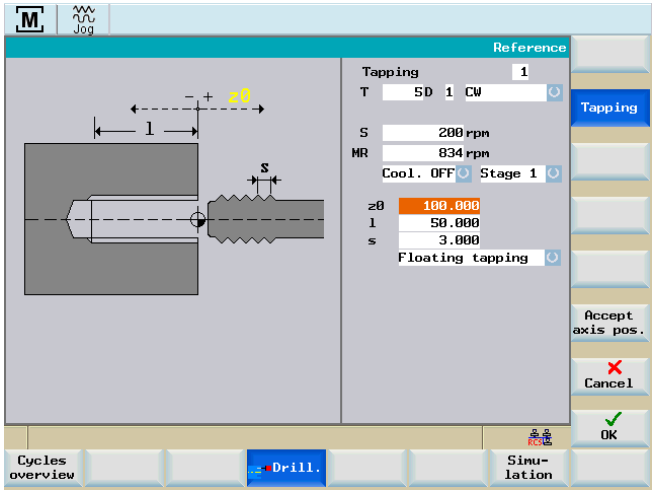


Figure 5-18 Example of input fields

Note

A detailed parameter description of each function can be found in the relevant sections.

The following softkeys will support you with the parameterization and execution of functions:

Accept axis pos.

The actual position value of the relevant axis is transferred to the parameter input fields when you press this softkey. The input field must be selected with the cursor keys, otherwise the error message "This value cannot be accepted" is displayed when accepting the axis position.

This softkey takes you back to the main screen.

If you have edited any values, the following prompt window appears:

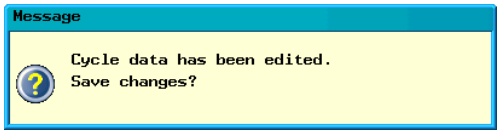


Figure 5-19 Cycles prompt text

Your inputs are accepted when you press the "OK" softkey.

Your inputs are discarded when you press the "Abort" softkey.

3. The function was parameterized (e.g. thread tapping).

   Activate the function using the "OK" softkey.

   The following execute screen appears:

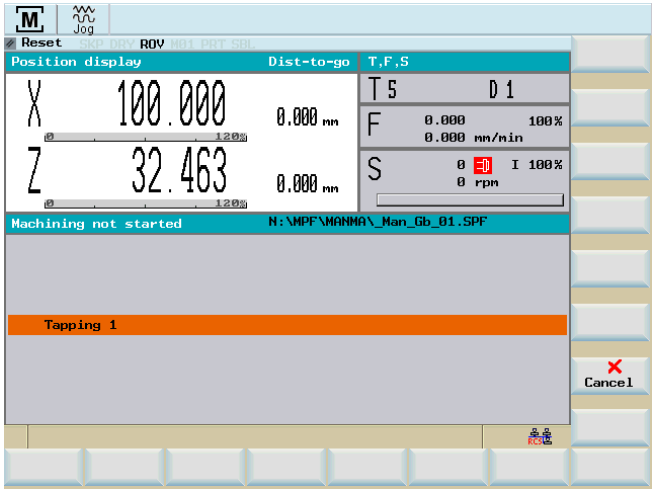


Figure 5-20 Example of executing a machining operation

The current machining status is displayed in the center of the execute screen. This status could be one of the following:

- Machining not started
- Machining active
- Machining aborted
- Machining interrupted
- Machining finished

In the example, the text "Machining not started" is displayed.

4. Start machining using the <NC-Start> key.

   The machining operation on the workpiece is executed.

Note

Press the <NC-Stop> key if you want to interrupt the machining operation. The selected direction of the spindle rotation continues to be activated.

By pressing the key <NC stop>, the operating mode JOG is automatically changed, i.e. you can traverse the axes manually. By continuing the execution with <NC start>, the interruption point is approached again and execution of the program is continued.

5. If machining was terminated (e.g. "Machining completed"), then the execution screen can be exited using the "Cancel" softkey.

See also

Messages (Page 123)

5.4.2 General parameters

General parameters

When parameterizing the particular functions, among others, the following general parameters are available:

Parameter		Description
Function name		Number of the selected function
Tool	T	Tool number
Compensation	D	Tool offset number
Direction of spindle rotation		Toggle field for spindle direction of rotation (clockwise/counterclockwise)
Feed value	F	Feed value
Type of feedrate		Toggle field for feed type (mm/min or mm/rev)
Spindle speed	S	Spindle speed value
Spindle type		Toggle field for spindle type (rev/min or m/min)
Spindle speed limitation	MR	Speed limitation for constant cutting rate
Coolant		Toogle field for coolant function (coolant OFF/coolant ON)
Machining		The type of machining operation can be selected in this toggle field. The following options are available: “Complete machining” -> “Roughing only” -> “Finishing only”
Position		The machining direction can be selected with the first toggle field. The following options are available: "Outside right" -> "Inside right" -> "Outside left". The diagram displayed on the screen changes to indicate which option is selected. The cutting direction can be selected with the second toggle field. Available selection: infeed in the longitudinal axis and roughing cut in the transverse axis (face), or roughing cut in the longitudinal axis and infeed in the transverse axis (long.). The current selection is shown in the form of a diagram on the screen.
Reference	z0	Contour start position in the longitudinal axis (absolute position of the Z axis).

Note

Additional parameter descriptions for the individual functions are provided in the corresponding chapters.

Optional parameter, gear stage preselection

In the input fields for the relevant manual machining, e.g. thread tapping, it is possible to pre-select the gear stage (see screenshot below).

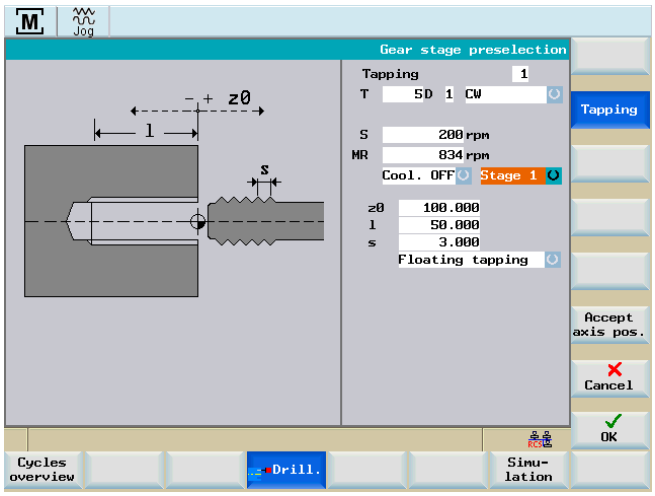


Figure 5-21 Gear stage preselection

If a gear unit is installed on the machine, you can select the gear stage using the <SELECT> key.

5.4.3 Manual drilling centered

Functionality

The "Manual drilling centered" function is designed to produce deep-hole drill holes in the turning center. Before you start the cycle, you must position the tool in such a way that it can approach the programmed Z initial position without risk of collision. The function itself will position the tool on the center of rotation.

Operating sequences

Drilling

You can access the drilling cycle overview by pressing the softkey "Drilling" in the basic screen for "Manual Machine Plus".

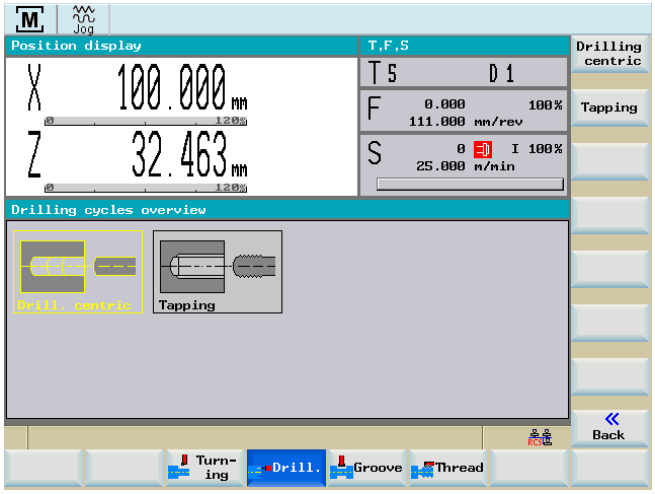


Figure 5-22 Drilling cycle overview - "Center drilling" selected

Drilling centric

You can access the "Manual center drilling" function by pressing the softkey "Center drilling" in the drilling cycle overview.
Alternatively, you can select "Center drilling" with <Cursor keys> and activate with the input key.

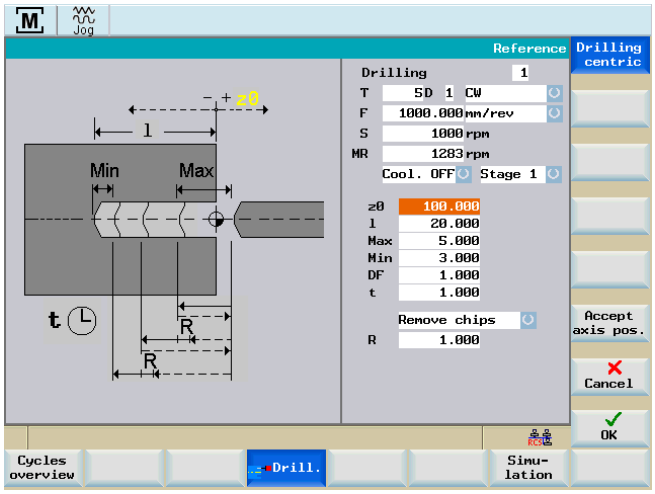


Figure 5-23 Drilling centric

Parameter

Parameter		Description
Reference	z0	Start position for the drill hole in the longitudinal axis (absolute position of Z axis)
Drilling depth	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.
Max. infeed	Max	Maximum infeed value for the first infeed in the longitudinal axis
Min. infeed	Min	Minimum infeed value in the longitudinal axis, which must be observed for the final infeed.
Degression factor f		Degression factor: the value by which the second and all subsequent infeeds in the longitudinal axis are multiplied. The following general rule applies: An input value greater than 1 increases the infeed depth with each infeed, an input value of less than 1 reduces it with each infeed. To switch off degression, enter 1 (or 0) here.
Dwell time	t	Dwell time on reaching the drilling depth
Return travel distance	R	Return travel distance in the longitudinal axis for chip breaking. This parameter is not displayed if "deswarfing" is selected.
Chip breaking / Deswarfing		With this toggle field you can select between machining with "chip breaking" or "deswarfing". With "chip breaking", on reaching the corresponding infeed depth the tool is retracted in the longitudinal axis by a defined value (chip 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 travel".

Drilling

The machining sequence is as follows:

1. 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 "Reference z0" parameter (taking into account the clearance distance).
2. The transverse axis is positioned to the center of rotation.
3. The first infeed in the axial axis (as defined in the "Infeed Max." parameter) is then performed.
4. The subsequent traversing movement in the axial axis depends on whether "chip breaking" or "deswarfing" has been selected. With "chip breaking" the tool is retracted in the longitudinal axis by the value set in the "return travel" parameter; with "deswarfing" the longitudinal axis is positioned at the cycle start point.
5. The subsequent infeeds in the longitudinal axis are always calculated in the same way: new infeed value = last infeed value x factor + return travel 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.
6. Infeed motion and "chip breaking/deswarfing" then alternate until the drilling depth specified in the "Length l" parameter is reached.
7. Once the required drilling depth is reached, the waiting time specified in the "Dwell t" parameter begins.
8. At the end of this waiting time, the tool is traversed to the cycle start point in the longitudinal axis.

See also

Principle operating sequence (Page 54)

General parameters (Page 58)

5.4.4 Manual thread tapping

Functionality

The "Manual thread tapping" function is designed to produce internal threads in the turning center, either with a compensating chuck or in a rigid tapping operation.

Before you start the cycle, you must position the tool in such a way that it can approach the programmed Z initial position without risk of collision. The function itself will position the tool on the center of rotation.

The machining feedrate is calculated from the programmed spindle speed and the thread pitch. This feedrate might not be the same as the programmed feedrate!

If you have selected "cutting rate" as the spindle type, the value set for the maximum spindle speed with G96 or the value for the maximum spindle speed is applied for thread tapping. (because the thread is tapped in the turning center, i.e. X=0)

⚠ DANGER
Notice: If "Time feed" is selected in the “Machining technology data” screen, in order for the pitch to be calculated correctly, the spindle override weighting must be set to "100%". Otherwise, the tapping tool or workpiece may be damaged!! **->Prior to NC start, check that the spindle override weighting is set to 100%!**

Operating sequences

Drilling

You can access the drilling cycle overview by pressing the softkey "Drilling" in the basic screen for "Manual Machine Plus".

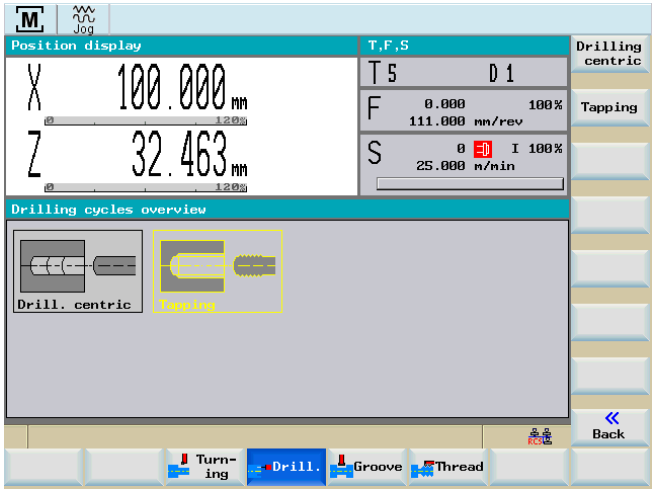


Figure 5-24 Drilling cycle overview - "Tapping" selected

Tapping

You can access the "Manual tapping" function by pressing the softkey "Tapping" in the drilling cycle overview.

Alternatively, you can select "Tapping" with <Cursor keys> and activate with the input key.

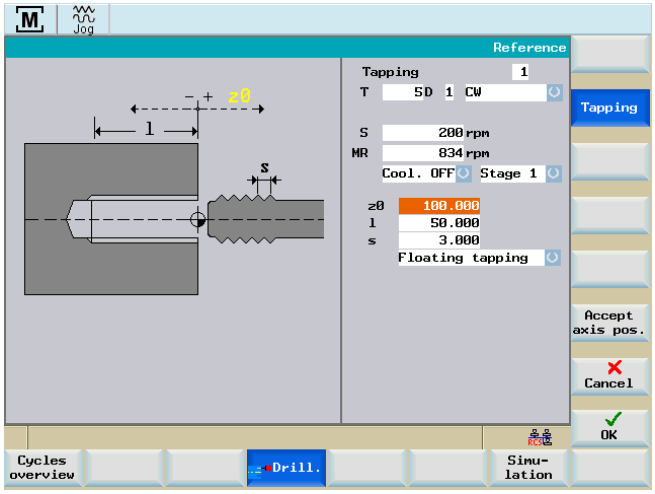


Figure 5-25 Tapping

Parameter

Parameter		Description
Reference	z0	Start position for the drill hole in the longitudinal axis (absolute position of Z axis)
Drilling depth	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.
Lead	s	Enter the thread pitch here.
With compensating chuck / Rigid tapping		Depending on the manufacturer, you can select as to whether the machining is carried-out with or without compensating chuck.

Tapping operation

The machining sequence is as follows:

1. 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 "Reference z0" parameter (taking into account the clearance distance).
2. The transverse axis is positioned to the center of rotation.
3. The controller then waits (at the cycle start point) for the next zero mark of the spindle encoder in order to start the axis movement in the longitudinal axis (defined thread start point).
4. 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.
5. 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.

See also

Principle operating sequence (Page 54)

General parameters (Page 58)

5.4.5 Manual grooving/parting

Functionality

The "Manual grooving" function is suitable for producing grooves on the peripheral surface and face end and for tapping turned parts. Groove cycles can be used to produce filleted corners or beveled edges on surfaces.

In addition, the "Multiple execution" function can be used to produce multiple tappings and multiple grooves with a uniform offset.

5.4.5.1 Groove cycle - single

Operating sequences

Groove

You can access the grooving cycle overview by pressing the softkey "Grooving" in the basic screen for "Manual Machine Plus".

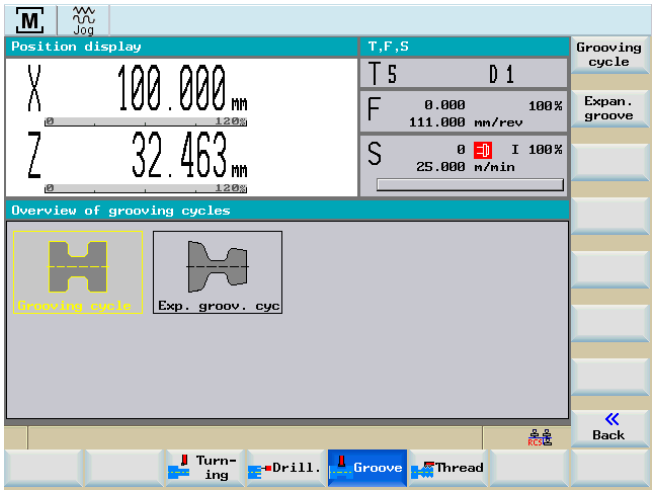


Figure 5-26 Grooving cycle overview - "Grooving cycle" selected

Recess cycle

You can access the "Manual grooving" function by pressing the softkey "Grooving cycle" in the grooving cycle overview.

Alternatively, you can select "Grooving cycle" with <Cursor keys> and activate with the input key.

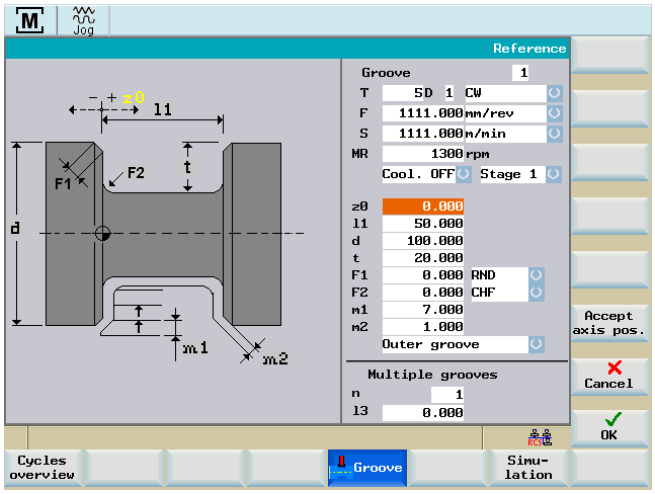


Figure 5-27 Outer groove

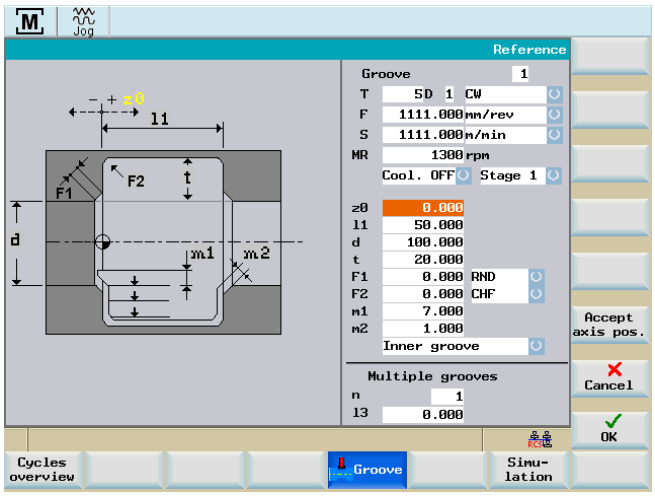


Figure 5-28 Inner groove

Parameter

Parameter		Description
Reference	z0	Starting position for the groove. The edge of the groove facing the chuck is always specified here. The value to be entered is the absolute position in the longitudinal axis (Z axis).
Groove width	l1	This value is the groove width, which together with the value for "Reference 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, and "0" is assigned to the parameters "Edge F1" and "Edge F2" (selection between "Chamfer CHF" and "Radius RND), the tapping function is activated.
Diameter	d	Starting diameter for the groove. The value to be entered is the absolute position in the transverse axis (X axis).

Parameter		Description
Groove depth	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.
Chamfer/radius	F1	Depending on the option selected, this value forms either an input radius (display "Radius RND") or an input chamfer (display "Chamfer CHF") of less than 45^0 on both sides of the groove. You can toggle between RND / CHF using the toggle key. An input value of 0.0 switches this function off.
Chamfer/radius	F2	Depending on the option selected, this value forms either a radius (display "Radius RND") or a chamfer (display "Chamfer CHF") of less than 45^0 on both sides of the groove as the transition to the base of the groove. You can toggle between RND / CHF using the toggle key. An input value of 0.0 switches this function off.
Max. infeed depth	m1	Enter the maximum infeed depth for roughing during grooving. The cycle's internal infeed calculation ensures that this input value is not exceeded during machining.
Finishing allowance	m2	Finishing allowance perpendicular to the contour.
Outer groove/inner groove		In this toggle field you can select whether an internal or an external groove is required. The selection is indicated by a diagram on the screen.

Groove cycle - single

The machining sequence is as follows:

1. Starting from the current axis position, the first calculated groove position is approached (diagonally) in both axes, taking into account the clearance distance and finishing allowance.
2. 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.
3. When the base of the groove is reached for the first time, the tool is withdrawn from the material at the programmed feedrate.
4. 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.
5. 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.
6. Finishing is started immediately after the roughing operation. The entire contour is traversed from both sides to the center of the base of the groove at the feedrate specified in the "Technology Data" screen before the start of the cycle.
7. Finally, the original position of the axes before the start of machining is approached diagonally.

See also

Principle operating sequence (Page 54)

General parameters (Page 58)

5.4.5.2 Groove cycle - multiple

Functionality

Note

The "Multiple groove" function supplements the "Single groove" option. This function can be used only if all the parameters for the "Groove cycle - single" function have been assigned!

As soon as you position the cursor in any of the input fields in the multiple grooves area of the screen, the display changes from single groove to multiple grooves:

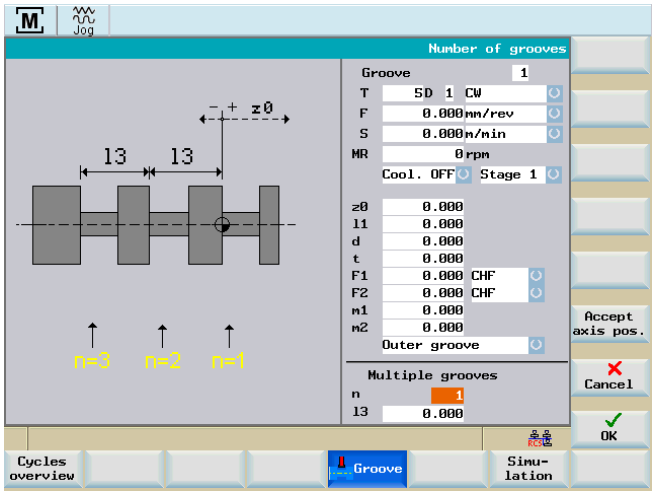


Figure 5-29 Groove cycle - multiple

Parameter

Parameter		Description
Distance	l3	Groove offset in the longitudinal axis (Z axis): This input value determines the offset between several identical grooves during production. The direction of the groove offset between the individual grooves is always towards the chuck.
Number	n	Number of grooves to be produced. Entering "0" or "1" here has the same effect: A single groove is produced. When you enter a value of ">1", the appropriate number of grooves is machined. The input value in parameter "Length l3" defines the necessary offset.

⚠ DANGER

Notice: When you are machining multiple grooves, make sure that there is sufficient clearance from the spindle as measured from starting position "Reference z0" to allow all parameterized grooves to be machined. There is otherwise a risk of collision between the tool and the chuck!

-> Check the plausibility of the input values before pressing NC start!

Multiple grooves

The machining sequence is as follows:

1. Starting from the current axis position, the first groove is produced as described under "Groove cycle - single".
2. 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).
3. Another complete groove is then machined (as described under "Groove cycle - single").
4. Groove machining and offset in the axial axis then alternate until the number of grooves specified in the "Number n" parameter has been produced.
5. On completion of the final groove, the original position of the axes before the start of machining is approached diagonally.

See also

General parameters (Page 58)

Principle operating sequence (Page 54)

5.4.5.3 Parting cycle - single

Functionality

The "Single parting" function is a subfunction of the "Groove cycle - single" function

Operating sequences

You activate the "Single parting" function by assigning the following parameters in the screen for "Groove cycle - single":

Parameter		Description
Groove width	I1	**Groove width "I1" must be identical to the tool width for the parting function.**
Chamfer/radius	F1	This value must be deactivated with a setting of 0.0 for parting.
Chamfer/radius	F2	This value must be deactivated with a setting of 0.0 for parting.

Note

The chamfer/radius parameters must be set to "0", otherwise the cycle cannot calculate the contour and displays an error message ("Groove form not defined correctly").

Parting

The machining sequence is as follows:

1. Starting from the current axis position, the first calculated parting position is approached (diagonally) in both axes, taking into account the clearance distance.
2. 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.
3. When the base of the groove is reached, the tool is withdrawn from the material at the programmed feedrate.
4. Finally, the original position of the axes before the start of machining is approached diagonally.

5.4.5.4 Multiple tapping

Functionality

The "Multiple parting" function is a subfunction of the "Groove cycle - multiple" function.

Operating sequences

You activate the "Multiple parting" function by assigning the following parameters in the screen for "Groove cycle - multiple":

Parameter		Description
Groove width	I1	**Groove width "I1" must be identical to the tool width for the "multiple parting" function.**
Chamfer/radius	F1	This value must be deactivated with a setting of 0.0 for "multiple parting".
Chamfer/radius	F2	This value must be deactivated with a setting of 0.0 for "multiple parting".

Note

The chamfer/radius (CHF/RND) parameters must be set to "0", otherwise the cycle cannot calculate the contour and displays an error message ("Groove form not defined correctly").

Multiple tapping

⚠ DANGER
Notice: When multiple parting is selected, make sure that there is sufficient clearance from the spindle as measured from starting position "Reference z0" to allow all parameterized parting operations to be performed. There is otherwise a risk of collision between the tool and the chuck! -> Check the plausibility of the input values before pressing NC start!

The machining sequence is as follows:

1. Machining takes place in the same way as for machining of "multiple grooves". The only difference lies in the fact that this is a "parting" rather than a "grooving" operation.
2. Finally, the original position of the axes before the start of machining is approached diagonally.

5.4.5.5 Extended grooving

Operating sequences

Groove

You can access the grooving cycle overview by pressing the softkey "Grooving" in the basic screen for "Manual Machine Plus".

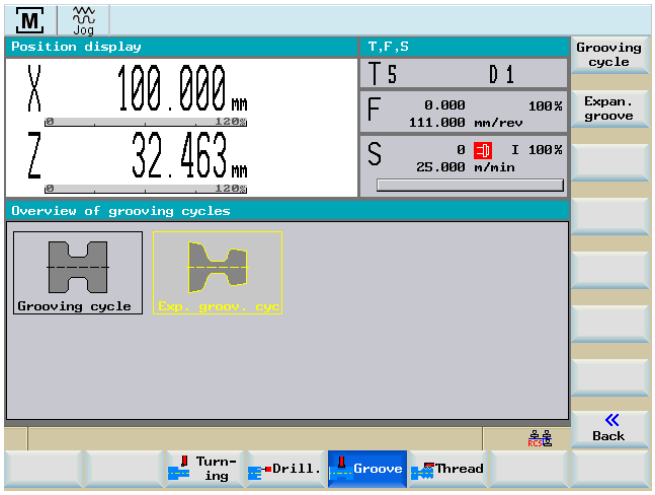


Figure 5-30 Grooving cycle overview - "Extended groove" selected

Extended Recess

You can access the "Extended grooving" function by pressing the softkey "Extended groove" in the grooving cycle overview.

Alternatively, you can select "Extended groove" with <Cursor keys> and activate with the input key.

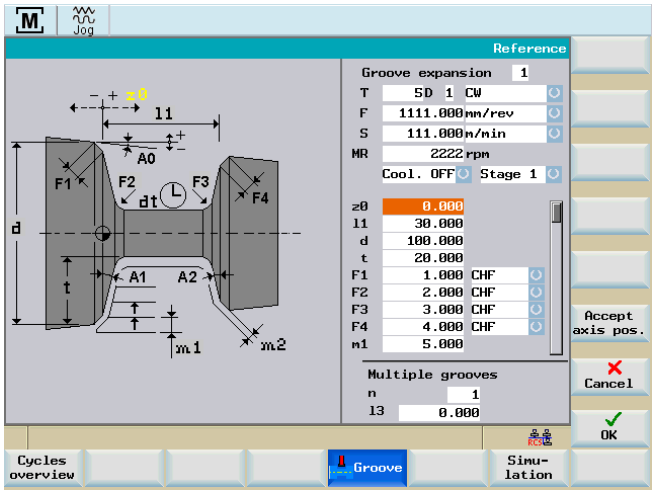


Figure 5-31 Extended external groove

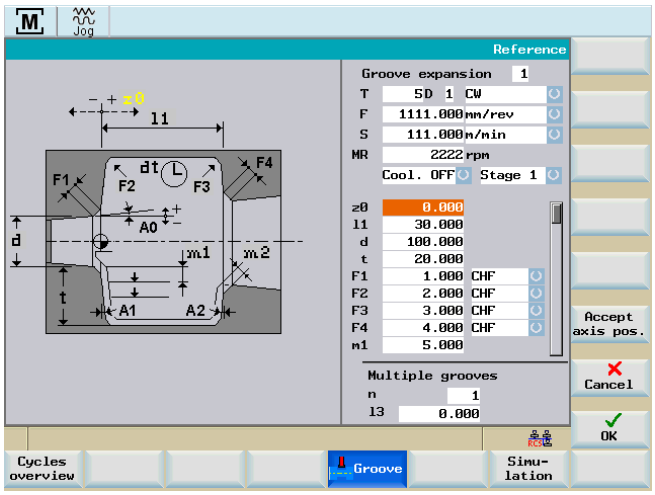


Figure 5-32 Extended internal groove

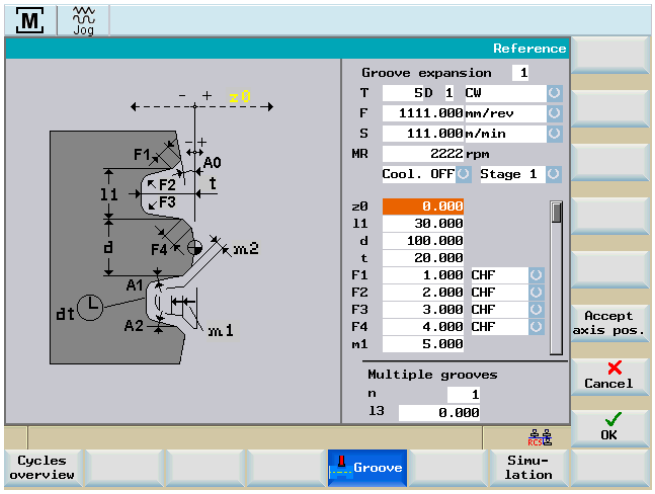


Figure 5-33 Face groove

Parameter

Parameter		Description
Reference	z0	Starting position for the groove. The edge of the groove facing the chuck is always specified here. The value to be entered is the absolute position in the longitudinal axis (Z axis).
Groove width	l1	This value is the groove width, which together with the value for "Reference 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, and "0" is assigned to the parameters "Edge F1" and "Edge F2" (selection between "Chamfer CHF" and "Radius RND), the tapping function is activated.
Diameter	d	Starting diameter for the groove. The value to be entered is the absolute position in the transverse axis (X axis).
Groove depth	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.
Chamfer/radius	F1	Depending on the option selected, this value forms either an input radius (display "Radius RND") or an input chamfer (display "Chamfer CHF") on the first side of the groove. You can toggle between RND / CHF using the toggle key. An input value of 0.0 switches this function off.
Chamfer/radius	F2	Depending on the option selected, this value forms either a radius (display "Radius RND") or a chamfer (display "Chamfer CHF") on the first side of the groove as the transition to the base of the groove. You can toggle between RND / CHF using the toggle key. An input value of 0.0 switches this function off.
Chamfer/radius	F3	Depending on the option selected, this value forms either a radius (display "Radius RND") or a chamfer (display "Chamfer CHF") on the second side of the groove as the transition to the base of the groove. You can toggle between RND / CHF using the toggle key. An input value of 0.0 switches this function off.
Chamfer/radius	F4	Depending on the option selected, this value forms either an input radius (display "Radius RND") or an input chamfer (display "Chamfer CHF") on the second side of the groove. You can toggle between RND / CHF using the toggle key. An input value of 0.0 switches this function off.
Max. infeed depth	m1	Enter the maximum infeed depth for roughing during grooving. The cycle's internal infeed calculation ensures that this input value is not exceeded during machining.

Parameter		Description
Finishing allowance	m2	Finishing allowance perpendicular to the contour.
External groove/internal groove/planar to chuck/planar from chuck		In this toggle field you can select the type of groove machining required whereby the respective selection is displayed in a diagram on the screen.
Contour angle	A0	This input value specifies the angle of the incline where the groove is to be executed.
Flank angle 1	A1	This input value determines the inclination of the first groove flank.
Flank angle 2	A2	This input value determines the inclination of the second groove flank.
Dwell time at recess base	dt	Here you can enter the dwell time of the tool on the groove base.

Extended grooving

The machining sequence is as follows:

1. Starting from the current axis position, the first calculated groove position is approached (diagonally) in both axes, taking into account the clearance distance and finishing allowance.
2. Executing the depth infeeds in the form of a roughing movement: 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.
3. When the base of the groove is reached for the first time, the tool is withdrawn from the material at the programmed feedrate.
4. Now the width infeed is executed: the width offset is calculated cycle-internally, taking into account the tool width and the groove width (length "l1") so that the machining is as uniform as possible.
5. 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.
6. Finishing is started immediately after the roughing operation. The entire contour is traversed from both sides to the center of the base of the groove at the feedrate specified in the “Technology Data” screen before the start of the cycle.
7. Finally, the original position of the axes before the start of machining is approached diagonally.

5.4.5.6 Multiple extended grooving

Functionality

Note

The "Multiple extended grooving" function supplements the "Extended grooving" option. This function can be used only if all the parameters for the "Extended grooving" function have been assigned!

As soon as you position the cursor in any of the input fields in the multiple grooves area of the screen, the display changes from single groove to multiple grooves:

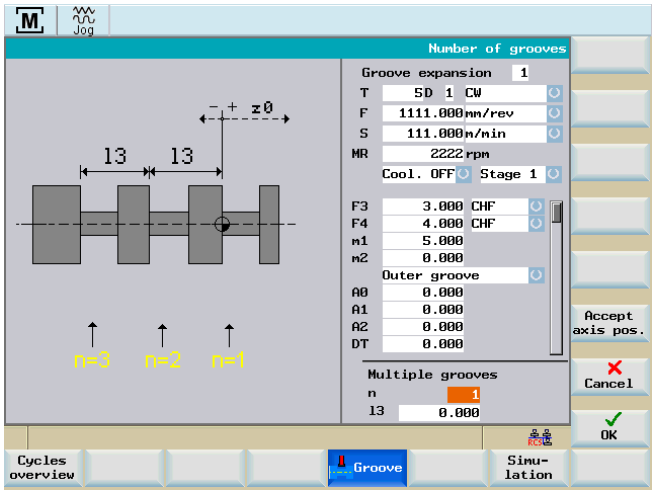


Figure 5-34 Multiple extended grooving

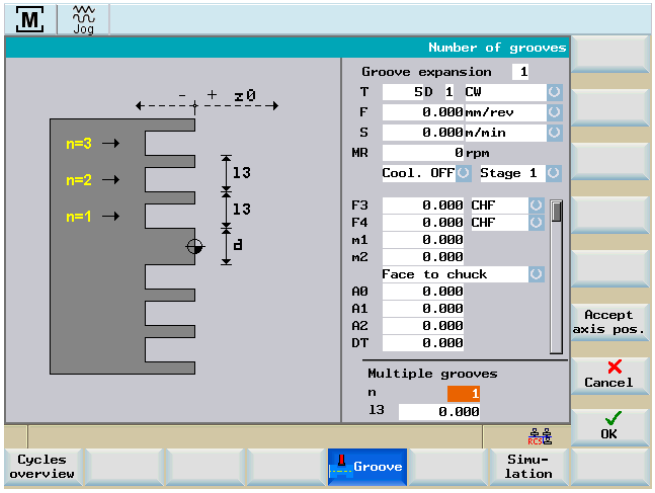


Figure 5-35 Face groove

Parameter

Parameter		Description
Distance	I3	Groove offset: This input value determines the offset between several identical grooves during production.
Number	n	Number of grooves to be produced. Entering "0" or "1" here has the same effect: A single groove is produced. When you enter a value of ">1", the appropriate number of grooves is machined. The input value in parameter "Length I3" defines the necessary offset.

Multiple grooves

The machining sequence is as follows:

1. Starting from the current axis position, the first groove is produced as described under "Extended grooving".
2. The starting point for the next groove is then approached taking into account the clearance distance.
3. Another complete grooving cycle is then executed (as described under "Extended grooving").
4. Groove machining and offset then alternate until the number of grooves specified in the "Number n" parameter has been executed.
5. On completion of the final groove, the original position of the axes before the start of machining is approached diagonally.

5.4.6 Manual thread cutting

Functionality

The "Manual thread cutting" function provides a wide range of options for producing, remachining and recutting longitudinal, tapered and face threads.

They can be single-start or multiple-start threads.

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.

5.4.6.1 Thread cutting

Operating sequences

Thread

You can access the "Manual thread cutting" function by pressing the softkey "Thread" in the main screen for "Manual Machine Plus".

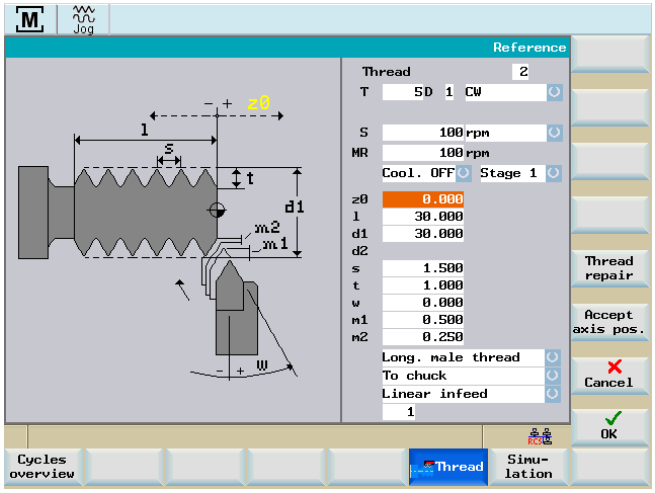


Figure 5-36 Longitudinal external thread

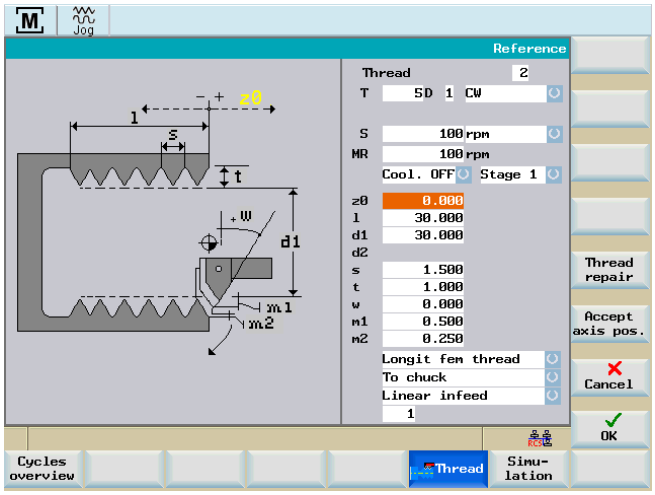


Figure 5-37 Longitudinal internal thread

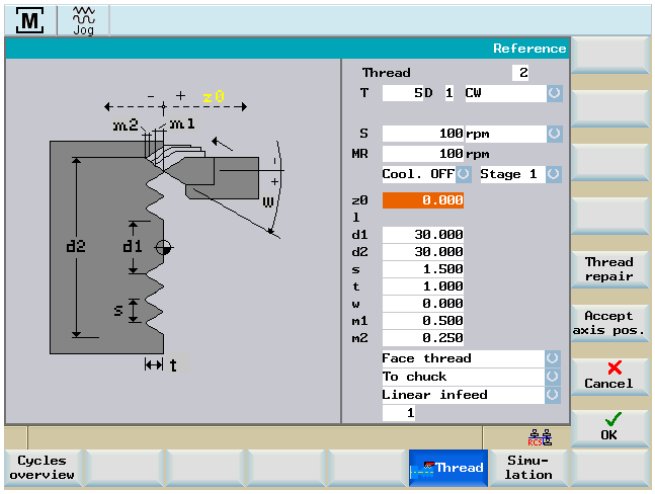


Figure 5-38 Face thread

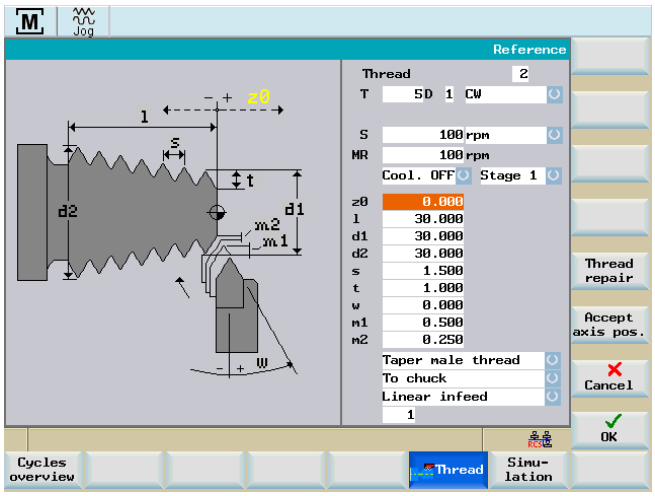


Figure 5-39 Taper external thread

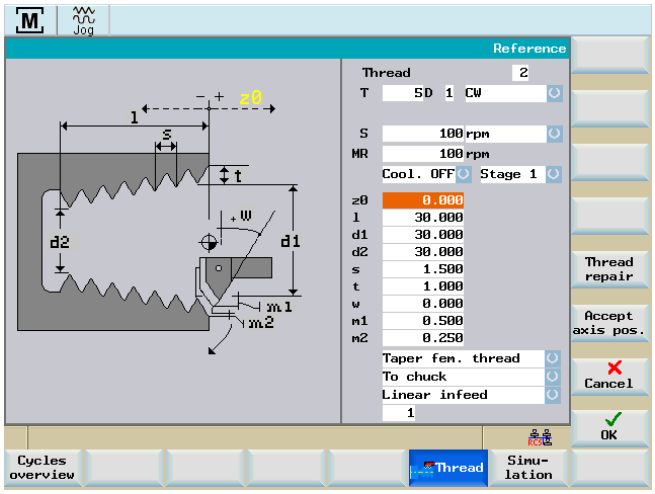


Figure 5-40 Taper internal thread

Parameter

Parameter		Description
Reference	z0	Start position for the thread in the longitudinal axis (absolute position of the Z axis).
Thread length	l	Enter the length of the thread to be created, taking the start position for the thread ("Reference z0") as the starting point. The thread cutting direction is selected by pressing the "Feed direction" softkey 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.
Diameter Start	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.
Diameter End	d2	End position for the thread in the transverse axis (absolute position of the X axis in the diameter).
Lead	s	Enter the required pitch in mm/rev.
Depth	t	This parameter is used to set the thread depth. The infeed direction and infeed depth depend on other parameters ("Intern Extern." and "Linear Degres." softkeys; "Angle" parameter). **Note:** If the value of the displayed machine date 1108 equals 1 (requirement for automatically calculating the thread depth) and the input field "t" equals 0, then the thread depth "t" will be automatically calculated and entered when you enter the pitch value. The following applies to all male threads: • a thread pitch of 1 mm results in a thread depth of 0.613 mm The following applies to all female threads: • a thread pitch of 1 mm results in a thread depth of 0.541mm The thread depth is adjusted according to an increase or reduction in the pitch value.
Infeed angle	w	Infeed angle; this specifies the angle of infeed during machining. A negative value causes an alternating infeed.
Max. infeed depth Min. infeed depth	m1	Enter here the maximum or minimum infeed depth for roughing. The cycle-internal infeed calculation ensures that this entry value is not exceeded or falls short during thread cutting. Input of a minimum infeed depth is possible only with degressive infeed. **Note:** The input field for the minimum infeed depth is displayed only if you have selected "Degressive infeed".
Finishing allowance	m2	Finishing allowance
Longitudinal internal thread/ longitudinal external thread/ face thread/ taper external thread/ taper internal thread		In this toggle field you can select whether an internal or an external thread is required. The selection is indicated by a diagram on the screen.
To chuck/ From chuck		You can select the machining direction of the thread using this toggle field.

Parameter	Description
Linear infeed/ Degressive infeed	This function key is used to switch between "linear infeed" and "degressive infeed". "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.
Number of thread starts	The number of thread starts is defined here.

Softkey

Thread recutting	This softkey is used to select remachining or thread recutting (thread repair).

Thread cutting

The machining sequence is as follows:

1. Starting from the current axis position, the start position for the thread (d1/z0) is approached in rapid traverse.
2. This is followed by infeed by the first depth of cut.
3. The controller then waits for the next zero mark from the spindle encoder in order to start the axis movements (longitudinal axis and/or transverse axis) (depending on the thread geometry).
4. Once the end position for the thread has been reached in both axes, the tool is withdrawn from the workpiece in rapid traverse.
5. The start position for the thread is then approached in the longitudinal and transverse axis in rapid traverse, observing a clearance distance.
6. Infeed to the next depth of cut.
7. 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 to smooth the thread is then performed and the start position for the thread is approached in the longitudinal and transverse axes.
8. There is now a choice of 2 options:
   - Machining is now complete and the "Execute" screen can be closed with the "Abort" softkey.
   - If you wish to continue machining the thread, e.g. if thread finishing is required, press NC Start again.

See also

Principle operating sequence (Page 54)

General parameters (Page 58)

5.4.6.2 Thread recutting

Functionality

The "Thread recutting" function is a subfunction of "Manual 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.

Operating sequences

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.

Thread recutting

You can call the thread recutting function with softkey "Thread repair".

The following screen form appears:

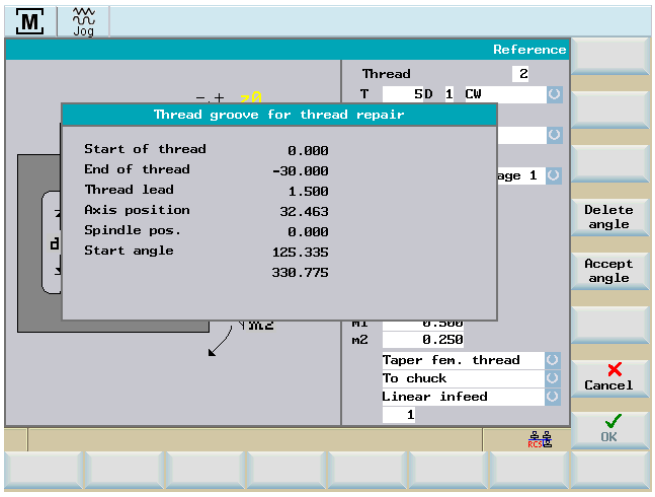


Figure 5-41 Thread recutting

The values displayed for start of thread, end of thread and axis position relate to the type of thread selected.

Longitudinal and taper thread ≤ 45° (Z axis)

Face and taper thread ≤ 45° (X axis)

All values displayed on this screen form are for information only; it is not, therefore, possible to alter them directly.

Execute thread recut

The following requirements must be fulfilled before you can recut a thread:

- Appropriate values must already have been entered in the "Thread Cutting" screen form at this point.
- The screen above is displayed.
- 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".

Accept angle

- Press the softkey "Accept angle". 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.

OK

- Press the softkey "OK" and the following screen appears:

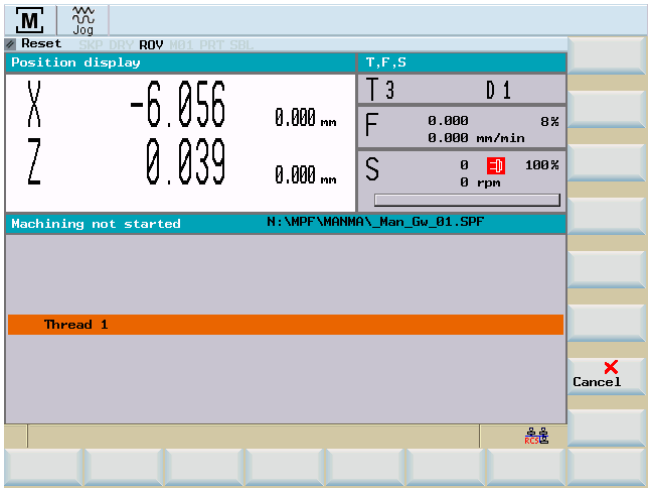


Figure 5-42 Execute thread recut

The rest of the thread cutting process is exactly the same as that described for "Manual 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 for "thread repair" (thread recutting) as the "start angle offset".

5.4.6.3 Thread shaving after thread cutting

Functionality

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".

Operating sequences

The following query is displayed at the thread end:

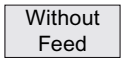
"Should the threads be recut?"

Softkeys

If no thread recut is necessary, confirm the prompt with "Cancel".

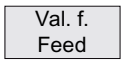
To perform a thread recut, confirm with "OK".

You can then select the infeed type using the following softkeys:



Pressing this softkey means that an additional cut is executed at the same infeed depth as the previous cut when you press <NC-Start> (additional "thread smoothing").

When this softkey is selected, the tool will be fed in by the amount entered in the "Val. f. Feed" field next time the NC Start key is pressed. The thread is machined accordingly in the next cut.



This softkey is used to enter the infeed value for subsequent machining. On pressing the key, the corresponding entry field (already on dark background) is displayed in the screen form shown below to enable you to enter the value.

Press the <INPUT> key as usual to accept the value.

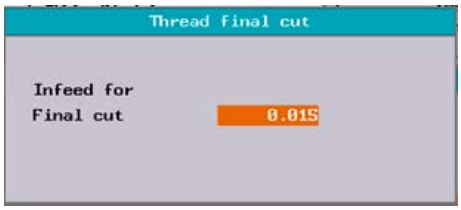


Figure 5-43 Enter infeed

After pressing the "With Feed" or "Without Feed" softkey, press NC Start to start machining an additional cut.

If you do not wish to do this, exit the "Execute Thread" screen by selecting softkey "Abort" without executing an NC start.

5.4.7 Roughing cycles

Functionality

The roughing cycles (integrated in the control) are the easiest way of producing common paraxial cutting contours. They are defined by setting particular input parameters in the appropriate screen forms.

The contour can be machined using the following position of the contour:

- "Outside right"
- "Inside right"
- "Outside left"

Roughing either be "Longitudinal" or "Face".

Operating sequences

Turning

Starting from the main menu for "Manual Machine Plus" you can reach the roughing cycle functions via softkey menu "Turning".

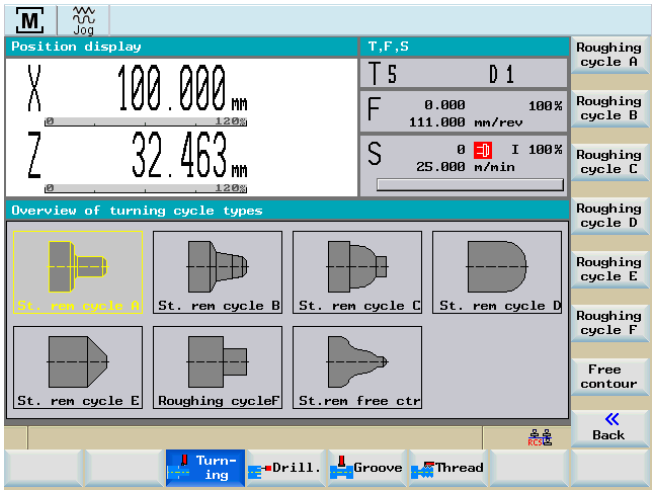


Figure 5-44 Turning cycle overview - Roughing cycle A selected

The following roughing cycles can be used via the vertical softkey bar or by selecting via <Cursor keys>:

- Roughing cycle A - simple stepped contour
- Roughing cycle B - expanded stepped contour with beveled edges
- Roughing cycle C - expanded stepped contour with rounding
- Roughing cycle D - single radius
- Roughing cycle E - single taper
- Roughing cycle F - Face and longitudinal turning
- Roughing cycle - free contour

5.4.7.1 Roughing cycle A

Functionality

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

Operating sequences

Turning

Roughing cycle A

You can access the turning cycle overview by pressing the softkey "Turning" in the basic screen for "Manual Machine Plus".

In the softkey menu "Turning", press the softkey "Roughing cycle A".

Alternatively, you can select "Roughing cycle A" from the turning cycle overview with <Cursor keys> and activate with the input key.

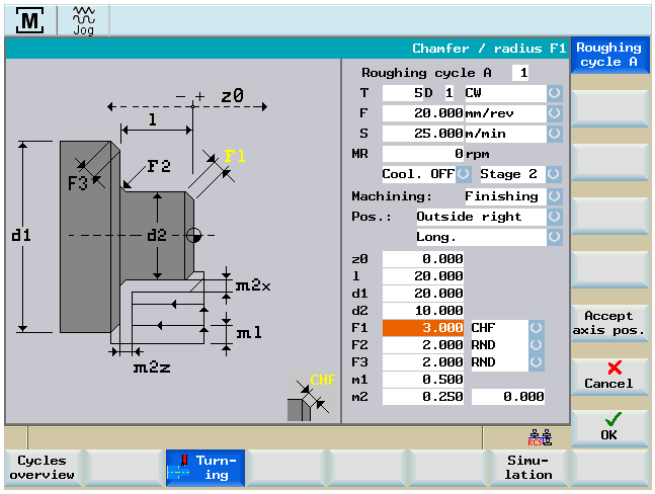


Figure 5-45 Roughing cycle A, position "Outside right"

Input fields

The input fields in the "Roughing A" screen form have the following meanings:

Parameter		Description
Length	l	Enter the length of the "step" to be produced, taking the contour start position ("Reference z0") in the axial axis (Z axis) as the starting point.
Diameter	d1	Outside diameter of the "step" to be machined in the radial axis (absolute position of the X axis in the diameter).
Diameter	d2	Inside diameter of the "step" to be machined in the radial axis (absolute position of the X axis in the diameter).

Parameter		Description
Chamfer/radius	F1	Depending on the option selected, this value forms either a transition radius (display "RND") or a transition chamfer (display "Chamfer CHR" or "Chamfer CHF") of less than 45^0 between the end face and the inside diameter of the "step". You can toggle between RND / CHR / CHF using the toggle key. An input value of 0.0 switches this function off. Two types of dimensioning are possible with the chamfers: • in the case of chamfer CHR, the value specifies the width of the chamfer in the direction of the movement, • in the case of chamfer CHF, the value corresponds with the length of the chamfer.
Chamfer/radius	F2	Depending on the option selected, this value forms either a transition radius (display "RND") or a transition chamfer (display "Chamfer CHR" or "Chamfer CHF") of less than 45^0 between the end face and the inside diameter of the "step". You can toggle between RND / CHR / CHF using the toggle key. An input value of 0.0 switches this function off. Two types of dimensioning are possible with the chamfers: • in the case of chamfer CHR, the value specifies the width of the chamfer in the direction of the movement, • in the case of chamfer CHF, the value corresponds with the length of the chamfer.
Chamfer/radius	F3	Depending on the option selected, this value forms either a transition radius (display "RND") or a transition chamfer (display "Chamfer CHR" or "Chamfer CHF") of less than 45^0 between the end face and the inside diameter of the "step". You can toggle between RND / CHR / CHF using the toggle key. An input value of 0.0 switches this function off. Two types of dimensioning are possible with the chamfers: • in the case of chamfer CHR, the value specifies the width of the chamfer in the direction of the movement, • in the case of chamfer CHF, the value corresponds with the length of the chamfer.
Max. infeed depth	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.
Finishing allowance	m2	Finishing allowance in the X axis (m2x) Finishing allowance in the Z axis (m2z)

The following possibilities exist for the position of the geometry:

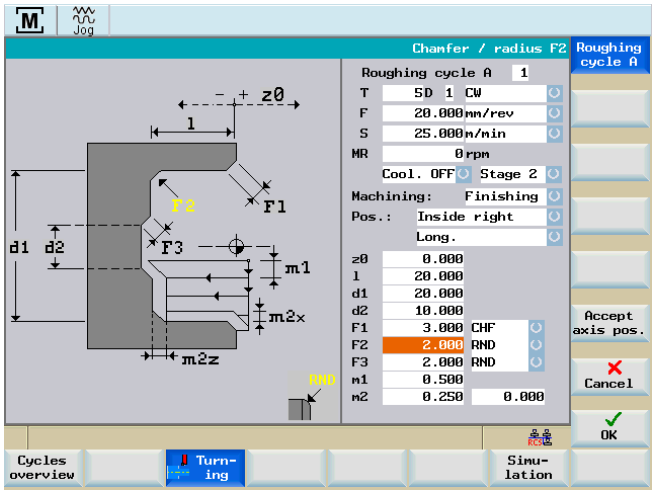


Figure 5-46 Roughing cycle A, position "Inside right"

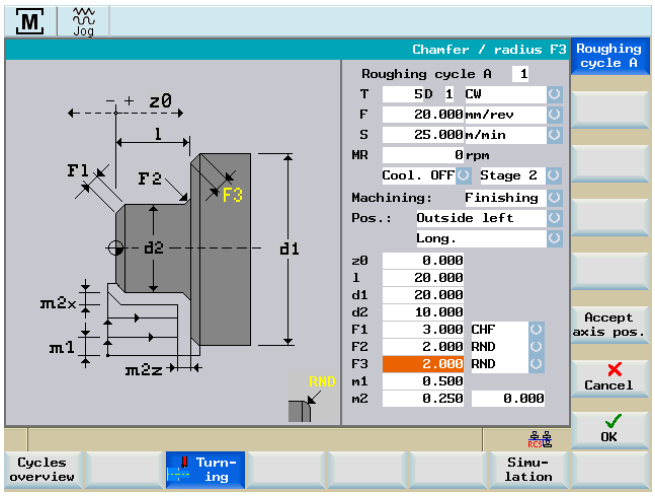


Figure 5-47 Roughing cycle A, position "Outside left"

See also

General parameters (Page 58)

Principle operating sequence (Page 54)

5.4.7.2 Roughing cycle B

Functionality

The function "Roughing B" 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.

Operating sequences

Turning

You can access the turning cycle overview by pressing the softkey "Turning" in the basic screen for "Manual Machine Plus".

Roughing cycle B

In the softkey menu "Turning", press the softkey "Roughing cycle B".

Alternatively, you can select "Roughing cycle B" from the turning cycle overview with <Cursor keys> and activate with the input key.

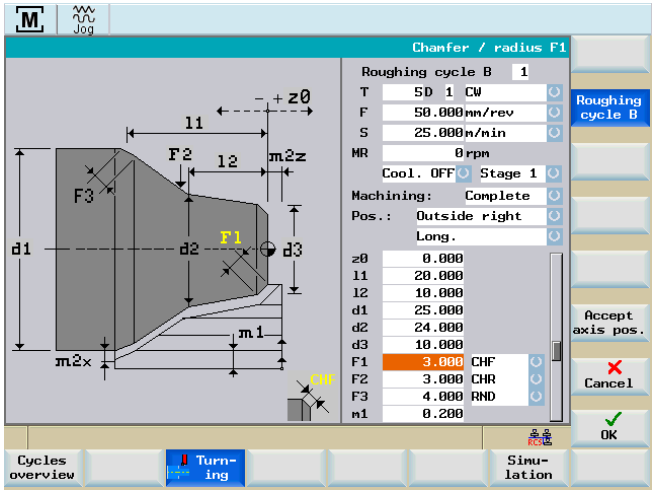


Figure 5-48 Roughing cycle B, position "Outside right"

Input fields

The input fields in the "Roughing B" screen form have the following meanings:

Parameter		Description
Length	l1	Enter the length of the "step" to be produced, taking the contour start position ("Reference z0") in the axial axis (Z axis) as the starting point.
Length	l2	Interpolation point position, which defines the position of the additional contour interpolation point in the longitudinal axis (Z axis).
Diameter	d1	Outside diameter of the "step" to be machined in the radial axis (absolute position of the X axis in the diameter).
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".
Diameter	d3	Inside diameter of the "step" to be machined in the radial axis (absolute position of the X axis in the diameter).
Chamfer/radius	F1	Depending on the option selected, this value forms either a transition radius (display "RND") or a transition chamfer (display "Chamfer CHR" or "Chamfer CHF") of less than 45^0 between the end face and the inside diameter of the "step". You can toggle between RND / CHR / CHF using the toggle key. An input value of 0.0 switches this function off. Two types of dimensioning are possible with the chamfers: • in the case of chamfer CHR, the value specifies the width of the chamfer in the direction of the movement, • in the case of chamfer CHF, the value corresponds with the length of the chamfer.
Chamfer/radius	F2	Depending on the option selected, this value forms either a transition radius (display "RND") or a transition chamfer (display "Chamfer CHR" or "Chamfer CHF") of less than 45^0 between the end face and the inside diameter of the "step". You can toggle between RND / CHR / CHF using the toggle key. An input value of 0.0 switches this function off. Two types of dimensioning are possible with the chamfers: • in the case of chamfer CHR, the value specifies the width of the chamfer in the direction of the movement, • in the case of chamfer CHF, the value corresponds with the length of the chamfer.
Chamfer/radius	F3	Depending on the option selected, this value forms either a transition radius (display "RND") or a transition chamfer (display "Chamfer CHR" or "Chamfer CHF") of less than 45^0 between the end face and the inside diameter of the "step". You can toggle between RND / CHR / CHF using the toggle key. An input value of 0.0 switches this function off. Two types of dimensioning are possible with the chamfers: • in the case of chamfer CHR, the value specifies the width of the chamfer in the direction of the movement, • in the case of chamfer CHF, the value corresponds with the length of the chamfer.
Max. infeed depth	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.
Finishing allowance	m2	Finishing allowance in the X axis (m2x) Finishing allowance in the Z axis (m2z)

The following possibilities exist for the position of the geometry:

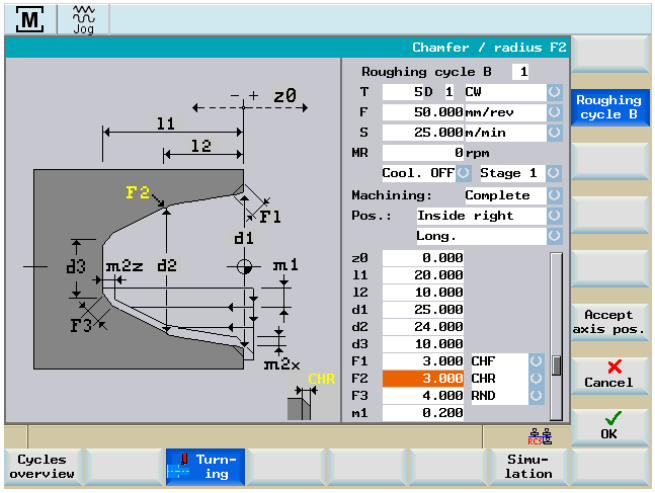


Figure 5-49 Roughing cycle B, position "Inside right"

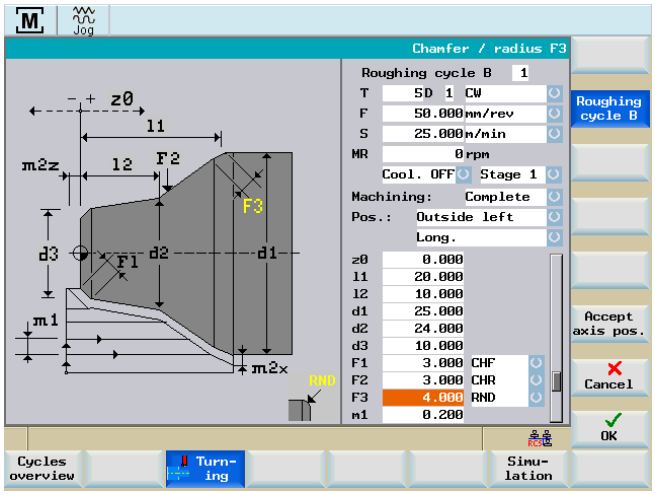


Figure 5-50 Roughing cycle B, position "Outside left"

See also

General parameters (Page 58)

Principle operating sequence (Page 54)

5.4.7.3 Roughing cycle C

Functionality

The function "Roughing C" 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.

Operating sequences

Turning

You can access the turning cycle overview by pressing the softkey "Turning" in the basic screen for "Manual Machine Plus".

Roughing cycle C

In the softkey menu "Turning", press the softkey "Roughing cycle C".

Alternatively, you can select "Roughing cycle C" from the turning cycle overview with <Cursor keys> and activate with the input key.

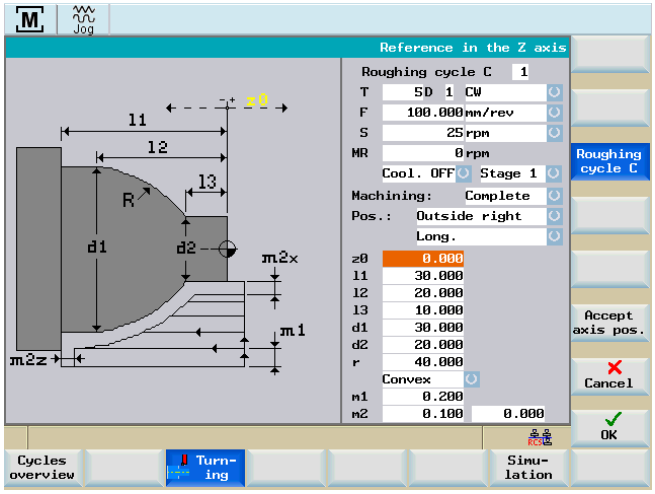


Figure 5-51 Roughing cycle C, position "Outside right"

Input fields

The input fields in the "Roughing C" screen form have the following meanings:

Parameter		Description
Length	l1	Enter the end point of the contour in the axial axis here, taking the contour start position ("Reference z0") in the axial axis (Z axis) as the starting point.
Length	l2	End point of filleting in the longitudinal axis (Z axis).
Length	l3	Start point of filleting in the longitudinal axis (Z axis).
Diameter	d1	Outside diameter of the "step" to be machined in the radial axis (absolute position of the X axis in the diameter).
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 "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.
Convex/concave		This toggle key is used to specify on which side of the contour the circle center point should be located. The circular machining direction, and hence the appearance of the finished contour, is adjusted accordingly.
Max. infeed depth	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.
Finishing allowance	m2	Finishing allowance in the X axis (m2x) Finishing allowance in the Z axis (m2z)

The following possibilities exist for the position of the geometry:

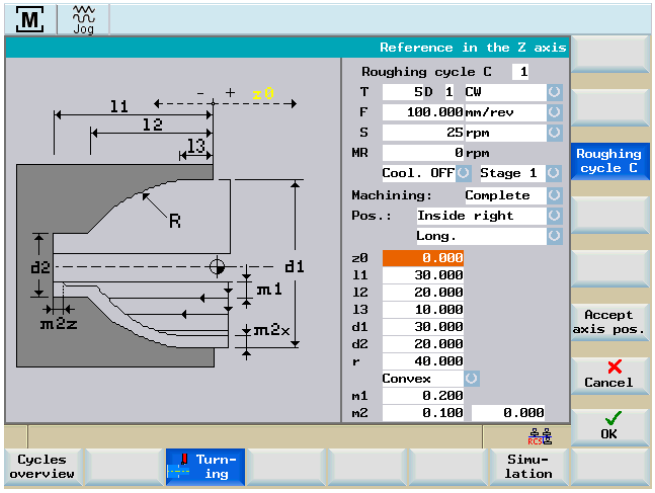


Figure 5-52 Roughing cycle C, position "Inside right"

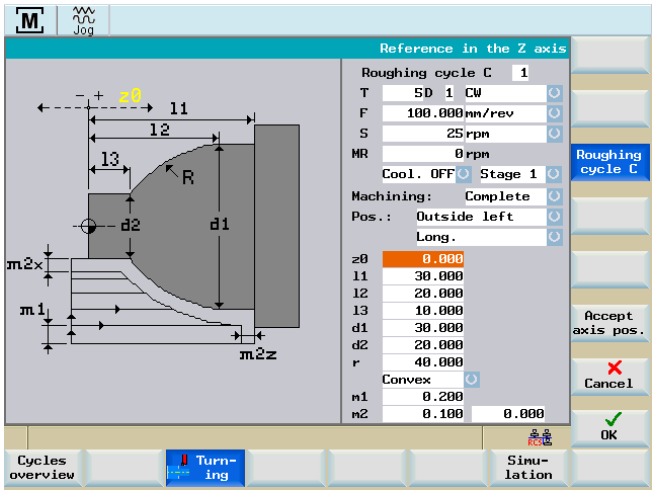


Figure 5-53 Roughing cycle C, position "Outside left"

See also

General parameters (Page 58)

Principle operating sequence (Page 54)

5.4.7.4 Roughing cycle D

Functionality

The function "Roughing D" allows a single radius contour to be machined, supported by cycles.

Operating sequences

Turning

You can access the turning cycle overview by pressing the softkey "Turning" in the basic screen for "Manual Machine Plus".

Roughing cycle D

In the softkey menu "Turning", press the softkey "Roughing cycle D".

Alternatively, you can select "Roughing cycle D" from the turning cycle overview with <Cursor keys> and activate with the input key.

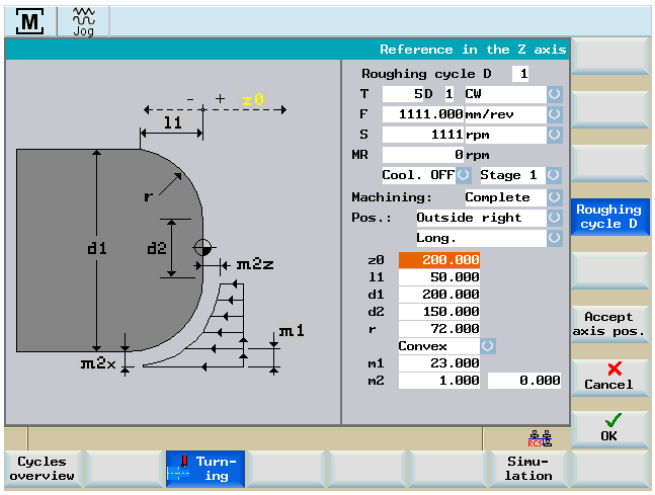


Figure 5-54 Roughing cycle D, position "Outside right"

Input fields

The input fields in the "Roughing D" screen form have the following meanings:

Parameter		Description
Length	l1	Enter the end point of the contour in the axial axis here, taking the contour start position ("Reference z0") in the axial axis (Z axis) as the starting point.
Diameter	d1	Outside diameter of the contour to be machined in the transverse axis (absolute position of the X axis in the diameter).
Diameter	d2	Inside diameter of the "radius" to be machined in the radial axis (absolute position of the X axis in the diameter).
Radius	R	This input value determines the size of the radius, the center of the circle being calculated internally. It is located on the imaginary line that is central and "normal" (90^0) to the imaginary connecting line between points "(Z0-l1)/d1" and "Z0/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 "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.
Convex/concave		This toggle key is used to specify on which side of the contour the circle center point should be located. The circular machining direction, and hence the appearance of the finished contour, is adjusted accordingly.
Max. infeed depth	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.
Finishing allowance	m2	Finishing allowance in the X axis (m2x) Finishing allowance in the Z axis (m2z)

The following possibilities exist for the position of the geometry:

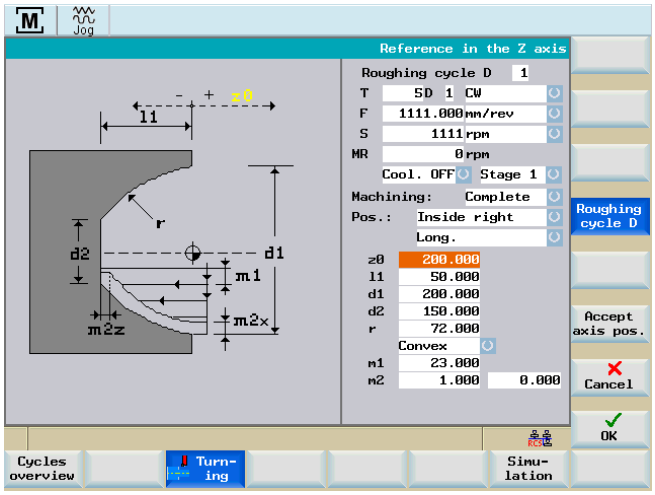


Figure 5-55 Roughing cycle D, position "Inside right"

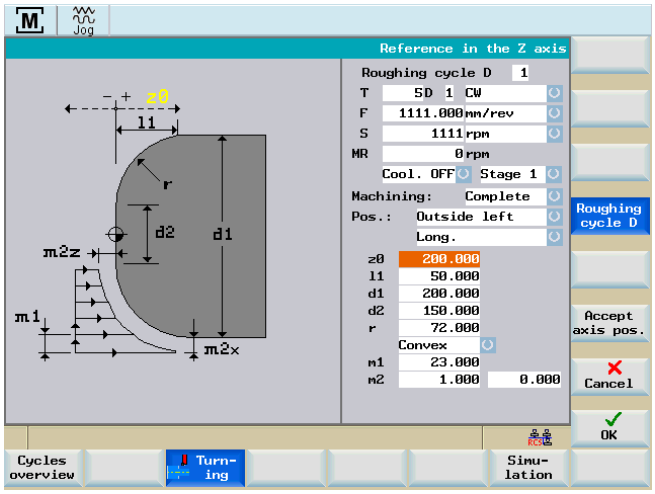


Figure 5-56 Roughing cycle D, position "Outside left"

See also

General parameters (Page 58)

Principle operating sequence (Page 54)

5.4.7.5 Roughing cycle E

Functionality

The function "Roughing E" allows a single taper contour to be machined, supported by cycles.

Operating sequences

Turning

You can access the turning cycle overview by pressing the softkey "Turning" in the basic screen for "Manual Machine Plus".

Roughing cycle E

In the softkey menu "Turning", press the softkey "Roughing cycle E".

Alternatively, you can select "Roughing cycle E" from the turning cycle overview with <Cursor keys> and activate with the input key.

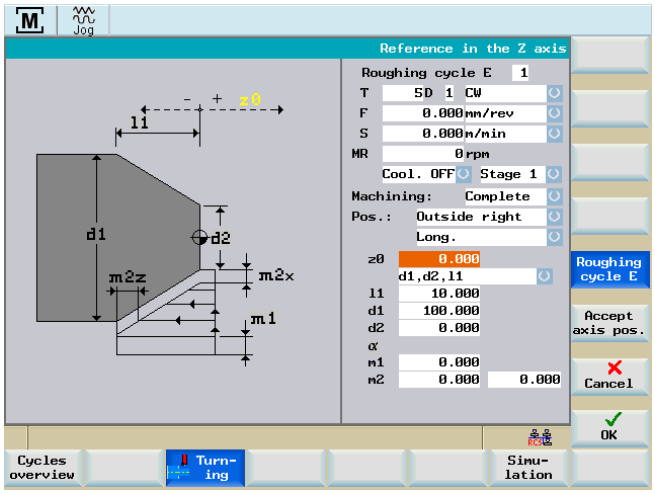


Figure 5-57 Roughing cycle E, position "Outside right"

Input fields

The input fields in the "Roughing E" screen form have the following meanings:

Parameter		Description
d1, d2,...		The dimensioning type can be selected in this toggle field. The following options are available: "d1,d2,l1" -> "d1,l1,angle" -> "d2,l1,angle" -> "d1,d2,angle(d1)" -> "d1,d2,angle(d2)" The selection is indicated in the diagram displayed on the screen.
Length	l1	Enter the length of the taper to be produced, taking the contour start position ("Reference z0") in the axial axis (Z axis) as the starting point.
Diameter	d1	Outside diameter of the taper to be machined in the transverse axis (absolute position of the X axis in the diameter).
Diameter	d2	Inside diameter of the taper to be machined in the transverse axis (absolute position of the X axis in the diameter).

Parameter		Description
Angle	α	Angle of the taper to be machined. The reference point is either d1 or d2 depending on which dimensioning type is selected.
Max. infeed depth	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.
Finishing allowance	m2	Finishing allowance in the X axis (m2x) Finishing allowance in the Z axis (m2z)

The following possibilities exist for the position of the geometry:

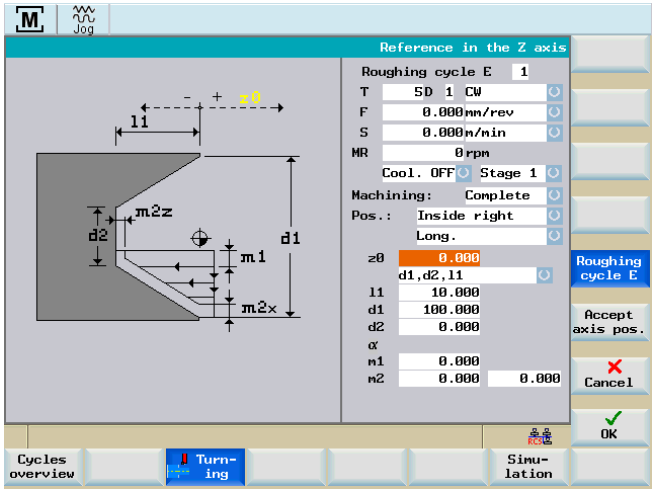


Figure 5-58 Roughing cycle E, position "Inside right"

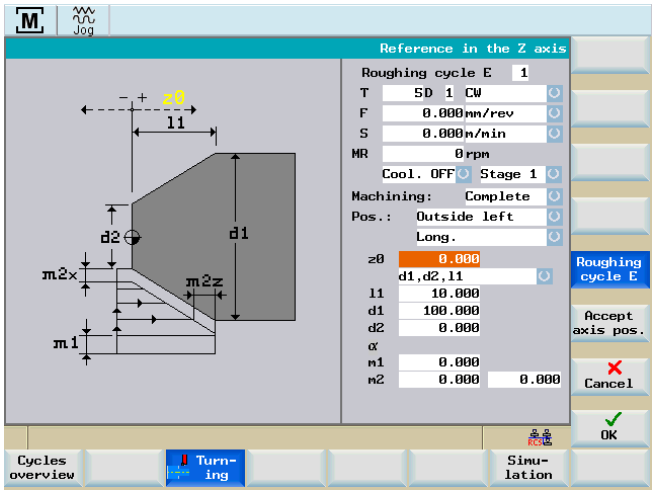


Figure 5-59 Roughing cycle E, position "Outside left"

See also

General parameters (Page 58)

Principle operating sequence (Page 54)

5.4.7.6 Roughing cycle F

Functionality

The function "Roughing F" allows cycle-supported production of an end face (cutting direction "Planar") or of a peripheral surface (cutting direction "Longitudinal").

Operating sequences

Turning

You can access the turning cycle overview by pressing the softkey "Turning" in the basic screen for "Manual Machine Plus".

Cutting cycle F

In the softkey menu "Turning", press the softkey "Roughing cycle F".

Alternatively, you can select "Roughing cycle F" from the turning cycle overview with <Cursor keys> and activate with the input key.

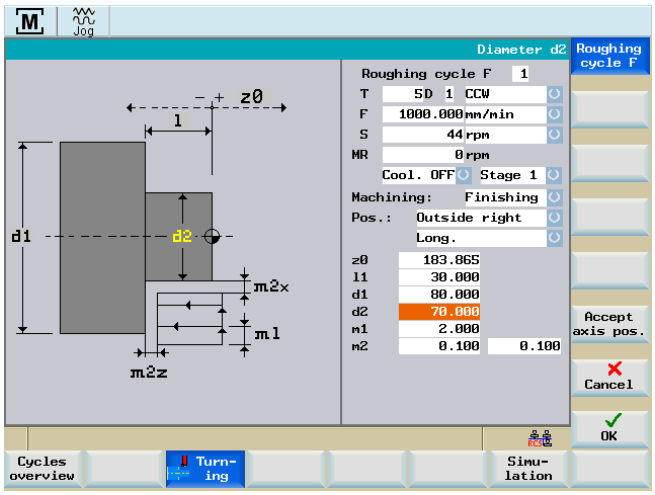


Figure 5-60 Roughing cycle F, position "Outside right"

Input fields

The input fields in the "Roughing F" screen form have the following meanings:

Parameter		Description
Length	l	Enter here the length of the end face to be cut, taking the contour start position ("Reference z0") in the longitudinal axis (Z axis) as the starting point.
Diameter	d1	External diameter of the end face to be cut (absolute position of the X axis in the diameter).
Diameter	d2	Internal diameter of the end face to be cut (absolute position of the X axis in the diameter).
Max. infeed depth	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.
Finishing allowance	m2	Finishing allowance in the X axis (m2x) Finishing allowance in the Z axis (m2z)

The following option is available for the position of the geometry:

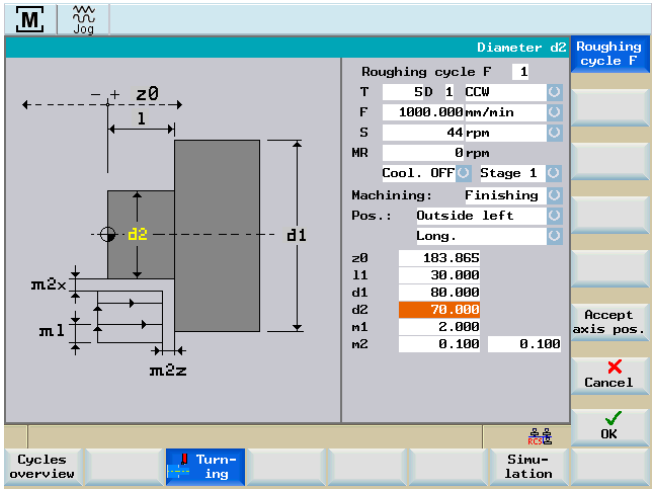


Figure 5-61 Roughing cycle F, position "Outside left"

5.4.7.7 Roughing cycle, free contour:

Functionality

The cycle "Free contour" is used for the input and for the processing of an arbitrary contour path.

Operating sequences

Turning

You can access the turning cycle overview by pressing the softkey "Turning" in the basic screen for "Manual Machine Plus".

Free contour

In the softkey menu "Turning", press the softkey "Free contour".

Alternatively, you can select "Roughing free contour" from the turning cycle overview with <Cursor keys> and activate with the input key.

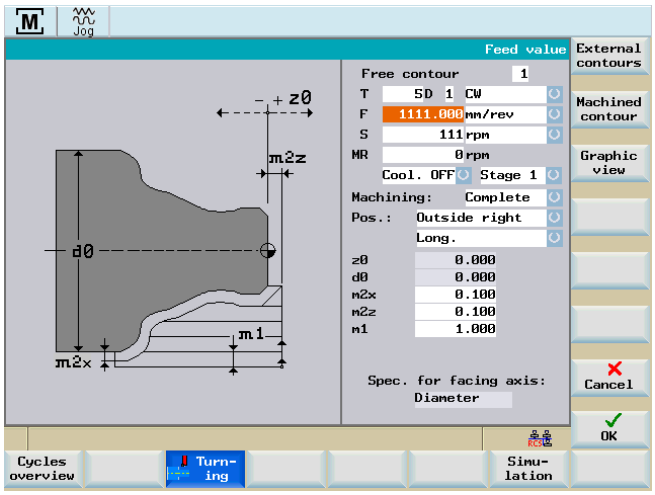


Figure 5-62 Roughing contour input, position "Outside right"

Parameter	Description
d0	Outside diameter of the shaft to be machined in the transverse axis (absolute position of the X axis in the diameter).
m2x	Finishing allowance horizontal to the contour.
m2z	Finishing allowance perpendicular to the contour.
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.
NAME	If an external contour has been selected, the path to the contour program is shown here.
Specification for transverse axis	The selected specification is displayed.

Softkeys

Cycle overview

The "Cycle overview" function lists all free contours contained in the machining step program.

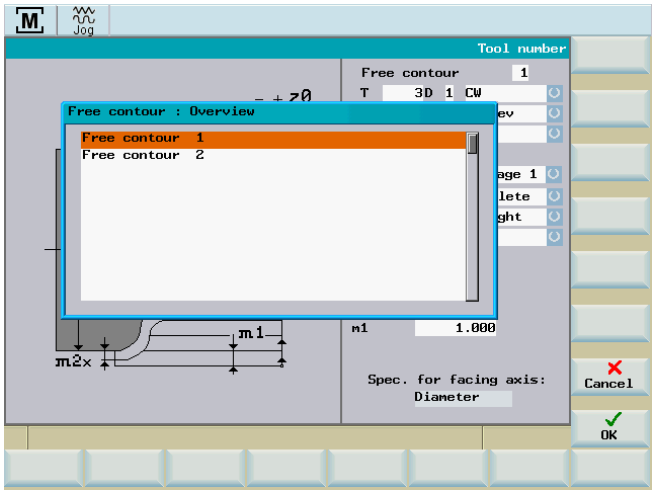


Figure 5-63 Free contour - overview

To select, place the cursor on the appropriate line and press the "OK" softkey.

External contours

It is also possible to assign a contour of an external contour subroutine to the cycle.

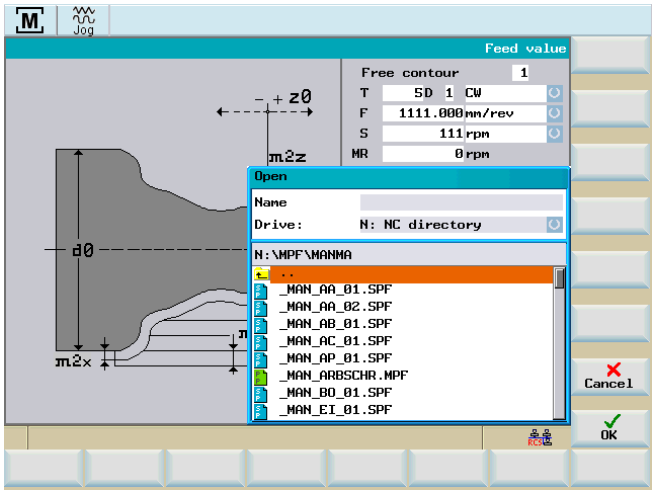


Figure 5-64 Add external contours

The "External contours" function opens a dialog box with which the contour subroutine can be selected. The "OK" softkey function links the selected program with the cycle.

Machine contour

The function branches to the contour input.

Note

Only contours listed in the cycle overview can be machined. External contours cannot be machined.

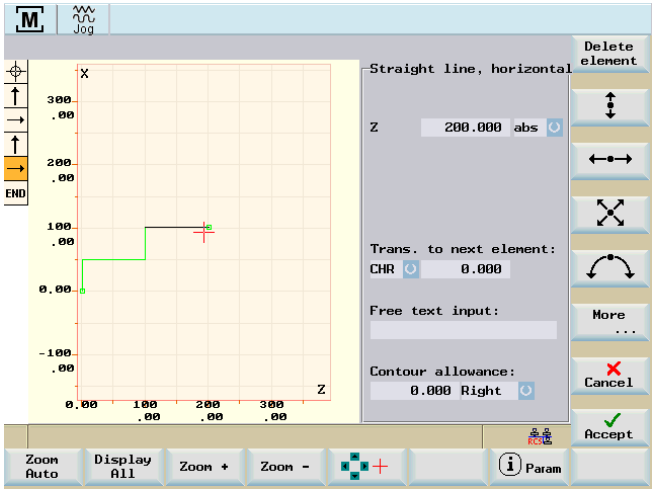


Figure 5-65 Machining window for free contours

First, define the contour starting point.

References

The function "Machine contour" is described in detail in the SINUMERIK 802D sl Turning Programming and Operating Manual in the chapter "Part programming; Free contour programming, ... Define a start point".

Graphic view

Rather than the auxiliary chart, the function displays the entered contour section.

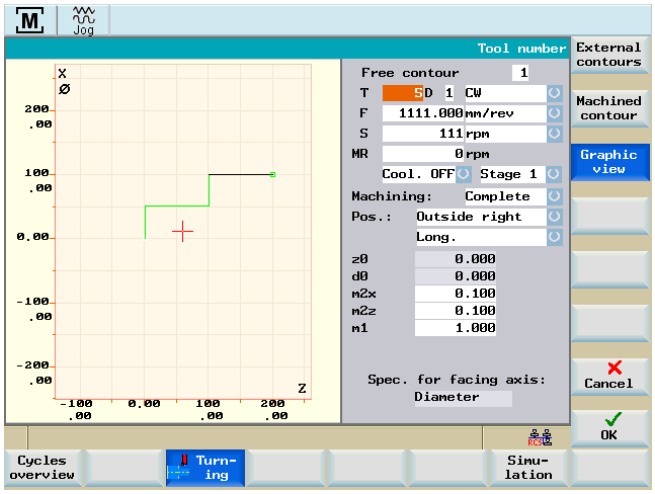


Figure 5-66 Graphic view of a contour section

5.4.7.8 Execute a roughing cycle

Rough cutting

Starting from the current axis position, the rough cutting sequence is as follows:

1. Diagonal approach to the start position calculated in the cycle in both axes.

   The safety clearance and finishing allowance are taken into account.

2. Infeed in the infeed axis (transverse axis or longitudinal axis, depending on whether "Face" or "Longitudinal" was selected).

   The infeed is calculated within the cycle as follows:

   – The input value "m1" is not exceeded.

   – The infeed quantity is guaranteed to remain constant until the unmachined contour is reached (taking finishing allowance into account).

3. Execution of the paraxial roughing motion in the cutting axis until the unmachined contour is reached.

   The finishing allowance is taken into account.

4. Withdrawal from the material in the infeed axis by the infeed distance.
5. Retraction by the clearance distance below 45^0 in both axes.
6. Return in the cutting axis to the start position calculated in the cycle.
7. 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.

Finish cutting

When the final rough cut is completed, the contour is finish cut with the following motions:

1. Traverse in the infeed axis to the finished dimension for the contour, corrected by the clearance distance.
2. Infeed in both axes (below 45^0) to the contour start point.
3. Execution of the finishing motion along the parameterized contour.
4. Retraction by the clearance distance below 45^0 in both axes.
5. Return in the cutting axis to the start position calculated in the cycle.
6. Finally, the original position of the axes before the start of machining is approached diagonally.

See also

Principle operating sequence (Page 54)

Machining the machining step program manually

6

Functionality

The "machining step program" function can be used to define a list containing an optional sequence of machining cycles.

This list can then be automatically machined step by step.

The controller can store a maximum of 390 steps.

Operating sequences

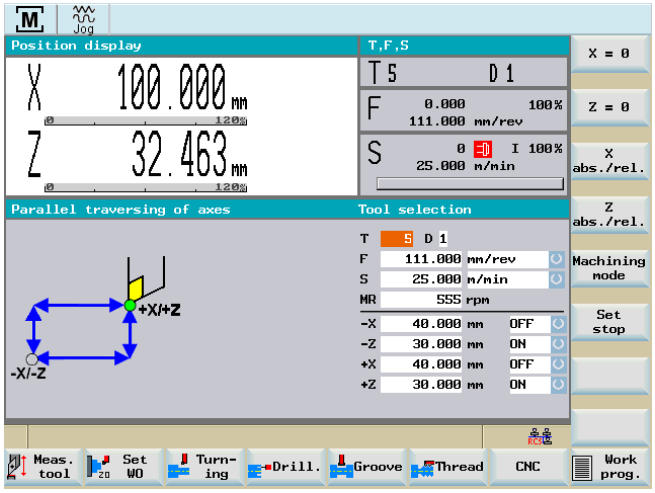


Figure 6-1 Entry into the machining step program

You can access the screen for input into the list by pressing softkey "Work prog." in the main screen for "Manual Machine Plus".

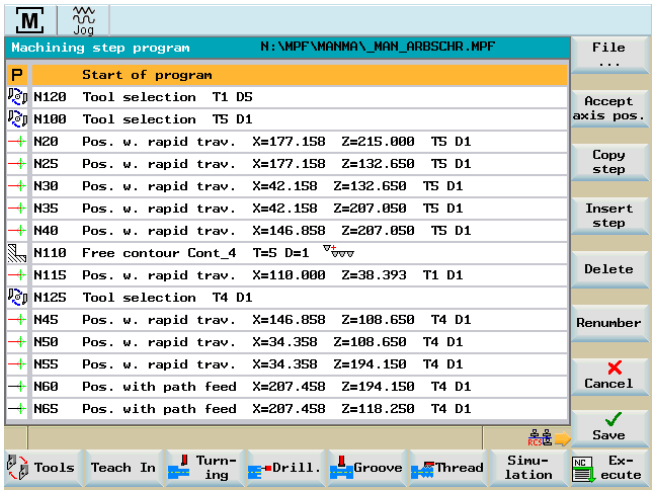


Figure 6-2 Machining step program

Screen handling functions

"Cursor up / down"	With the cursor up/cursor down keys, you can move selected machining steps up and down within the list. The selected step is displayed on an orange background.
"Cursor right"	If you have selected a machining cycle , the input screen for this cycle or the taught block opens automatically when you press the cursor key on the right.

Softkeys

Other inputs in the screen are made with softkeys:

File service

Opens the following dialog box:

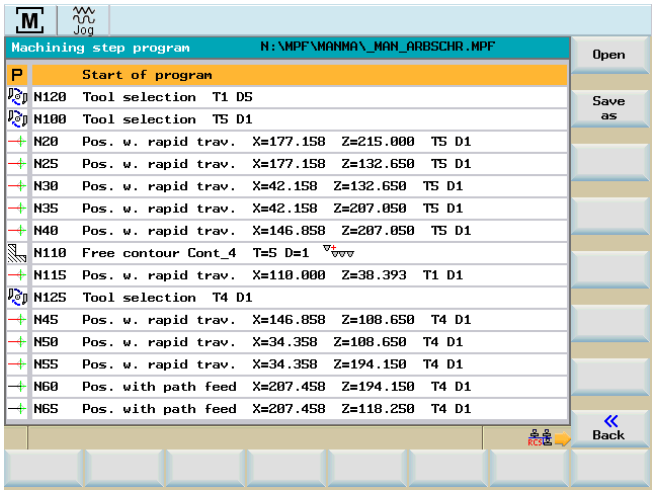


Figure 6-3 "File..." menu

Open	Displays a dialog used to open an existing machining step program or create a new machining step program. If the file is not located in drive N (NC storage), ensure that the external medium is not removed during the machining.
Save under	A save dialog appears.
<< Back	Returns to the menu for the machining step program

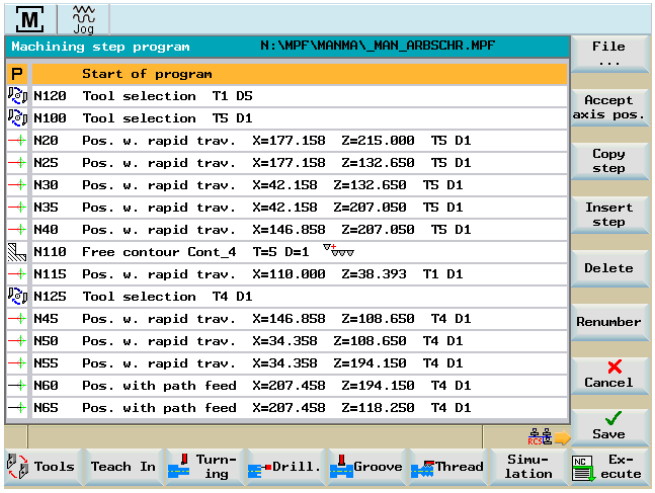


Figure 6-4 Machining step program

Accept axis pos.	This function inserts a positioning block at the current machine axis position in the selected machining step.
Delete	This function deletes the currently selected machining step.
X Abort	To interrupt the function "Machining step program" press "Cancel". This softkey returns you to the main screen for "Manual Machine Plus". If you have changed any values, a prompt window informs you accordingly. Press "OK" to save the values Press "Cancel" to discard the settings.
Save	The machining step program is saved. This softkey returns you to the main screen for "Manual Machine Plus".
Tool	Inserting a tool change into the program. Change into the dialog box for tool change (Page 111).

Teach In cycle	Inserting a traversing block in the program.
Turning	Inserting a roughing cycle in the program. Change into the dialog box for roughing cycles (Page 84).
Drilling	Inserting a drilling cycle in the program. Change into the dialog box (Page 60) for the drilling cycles (Page 62).
Groove	Inserting a groove/cutting cycle into the program. Change into the dialog box for the groove/cutting cycles (Page 65).
Thread	Inserting a thread cycle Change into the dialog box for the thread cycles (Page 76).
Simu-lation	Change into the dialog box for machining simulation (Page 117).
	The selected step program is saved by pressing the "Execute" softkey. This softkey opens the screen form in which the machining (Page 120) step program is actually executed:
	The softkey function changes from "Execute" to "Exec. here" by pressing the <ETC> key.
	By pressing the softkey "Exec. here" the function changes into the screen form in whichMachining (Page 120) should be realized. In this case, the machining step program is executed from the current cursor position in the machining step program (marked step in the program).

6.1 Tool change in the machining step program

Functionality

You add a tool change step to the machining step program.

If the value of the display machine data 361 (USER_MEAS_TOOL_CHANGE) is 1, the tool number can be specified manually. Otherwise the controller saves the active tool as machining step in the step program.

Operating sequences

You have opened a machining step program.

1. Move the cursor to the machining step after which the tool should be changed.

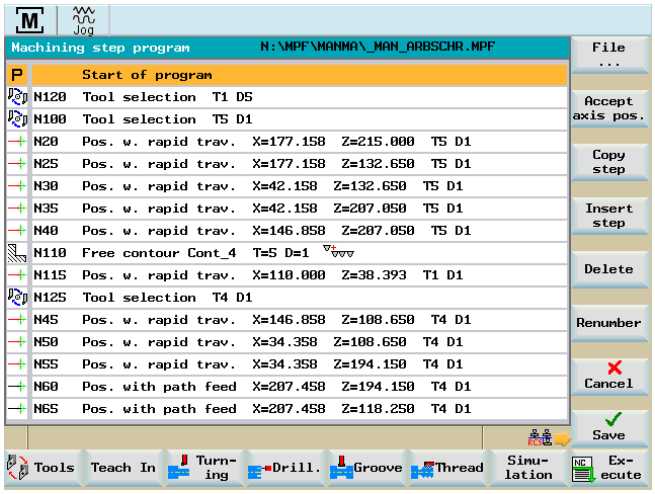


Figure 6-5 Machining step programmed opened

Tool

2. Press the "Tools" softkey.

   A list with all tools created in the NC will be displayed.

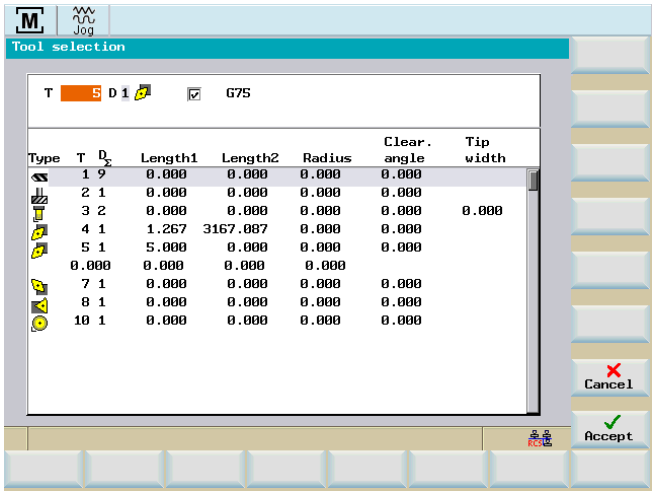


Figure 6-6 Tool selection for tool change

The fields "T" and "D" contain the active tool and the active cutting edge number, respectively.

3. To select the tool, enter the tool number and the cutting edge number in the input fields "T" and "D", respectively.

   - OR -

   Use the <Tab key> to change to the list and position the cursor on the appropriate tool and press the <INPUT> key to confirm the selection.

   The selected tool is copied into the "T" input field.

4. If the tick is set in the field "G75" (approach fixed point), the controller travels to a stored fixed point before the tool change.

5. Press "OK".

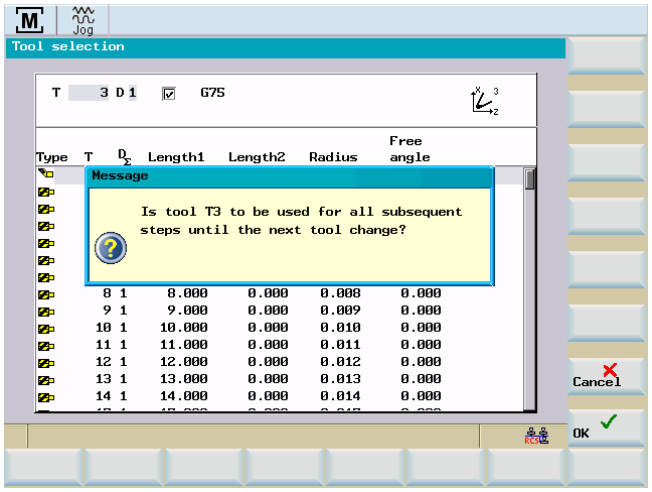


Figure 6-7 Tool change in the machining step program, first prompt

With this prompt, you decide whether the tool should be used in all subsequent steps up to the next tool change.

6. The following options are available:

OK

7. If you press "Cancel", the tool change only takes place in this step, or

   if you press "OK", the tool is also changed in all subsequent program steps.

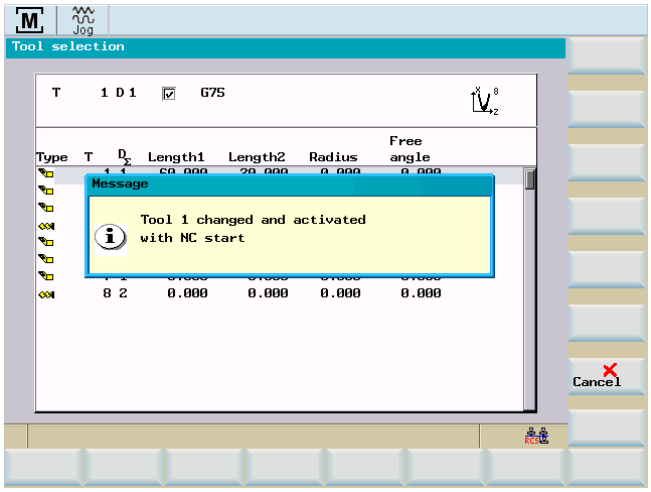


Figure 6-8 Tool change in the machining step program, confirmation

Abort

8. Press "Cancel" to confirm the message.

   The active tool is inserted into the step program as machining step.

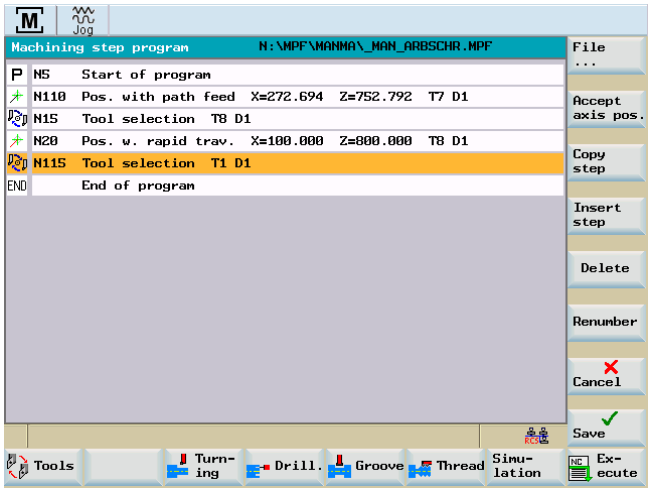


Figure 6-9 Tool change in the machining step program

Note

If the tool change is to be performed without travelling to a fixed point, travel to a safe machine position beforehand and save this point as work step in the step program.

6.2 Teach In

Functionality

Using this function, an approached axis position can be directly entered into a specific traversing block.

Operating sequences

Teach In cycle

1. You can reach the "Teach In" function in the machining step program by pressing the "Teach In" softkey.

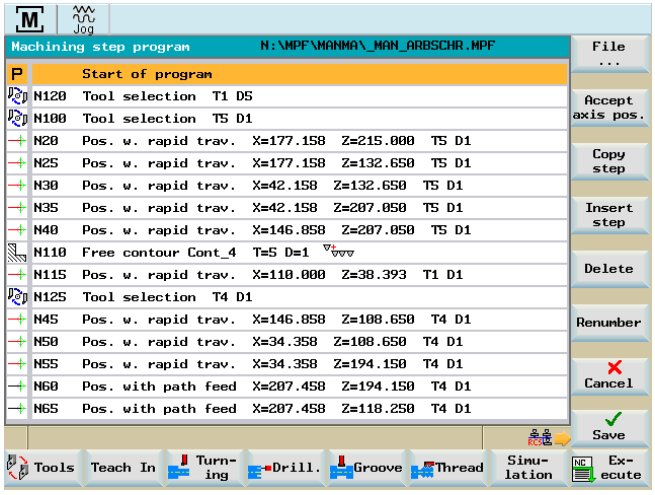


Figure 6-10 Selecting the "Teach In" function

The controller switches to the manual machining screen forms of axis-parallel turning, taper turning and radius turning.

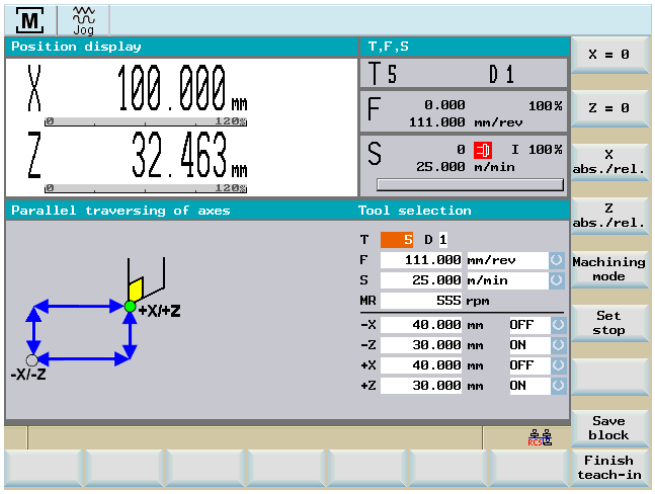


Figure 6-11 Executing "Teach In" - axis-parallel roughing machining mode

Set
Save

2. Traverse to a position that is to be taught-in and press "Save block".

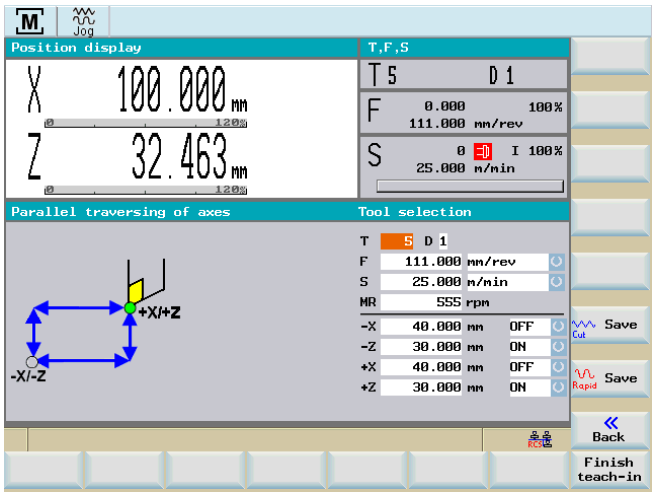


Figure 6-12 "Save block" menu

Save

3. You can save the position with path feed.

Save
Rapid

4. You can save the position with rapid traverse.

After the control acknowledged the action with a screen message (e.g.: "The block was inserted as N20"), a new position could be traversed to and this in turn taught-in using "Save block".

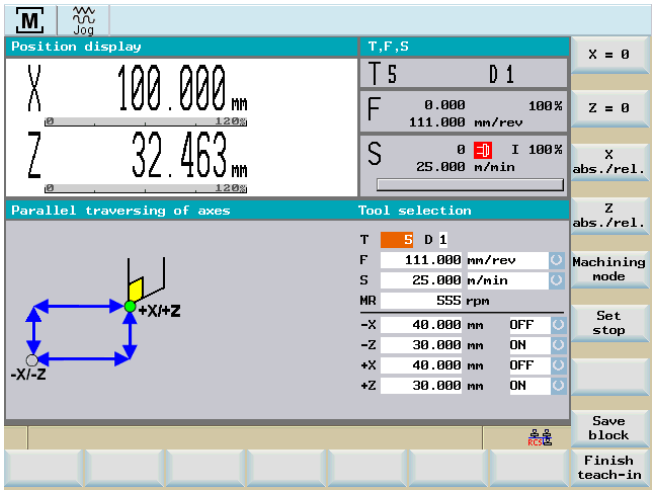


Figure 6-13 Closing "Teach In"

Teach In
Exit

5. Exit the "Teach In" mode using the function "Finish Teach In".

   The menu returns to the machining step program.

   The cursor is at the last block that was entered (refer to the following screen shot).

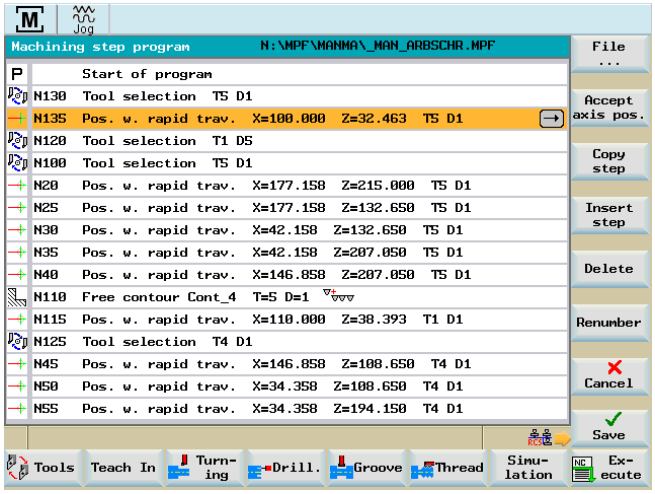


Figure 6-14 Machining step program "Teach In" ended

6.3 Simulate machining

Function

You can use this function to graphically display the execution of the program on the screen, in order to easily control the programming result without moving the machine axes.

Operating sequences

Simu-lation

The start screen is opened.

The simulation of the part program can be reproduced on the HMI using the following two functions:

- Standard simulation

  The execution of the part program is simulated on the HMI by considering the axis feedrates. In the case of more extensive NC programs, the simulation may, therefore, take more time.

- Contour simulation

  Execution of the part program is simulated on the HMI. The simulation is based on pure calculations and is, therefore, faster in the case of more extensive NC programs.

Standard simulation

Standard simulation

The execution of the part program is simulated with this function on the HMI by considering the axis feedrates.

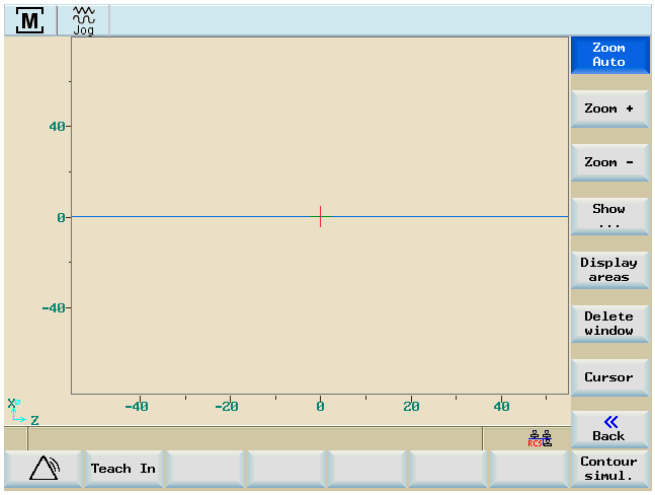


Figure 6-15 Standard simulation

CYCLE START

Press <NC START> to start the standard simulation for the selected part program.

Contour simulation

Contour simulation

Execution of the part program is simulated with this function on the HMI.

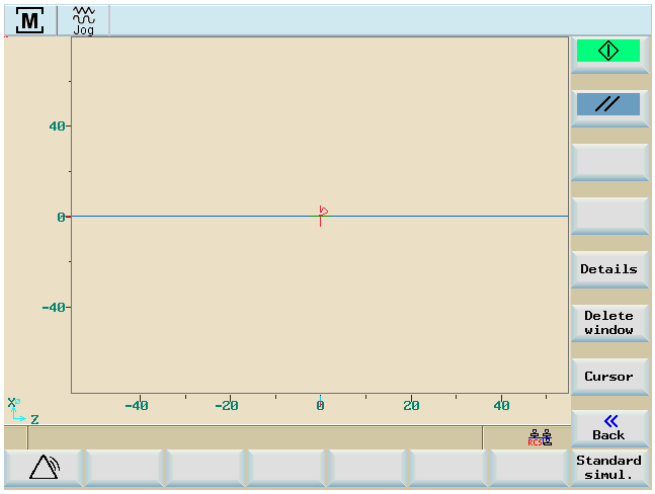


Figure 6-16 Contour simulation

The selected part program is started for the contour simulation.

Simulation of individual cycles

Note

If the simulation is used to test a single cycle, the display area is divided into the traversing movements and technology data columns. The technology data cannot be changed in simulation mode.

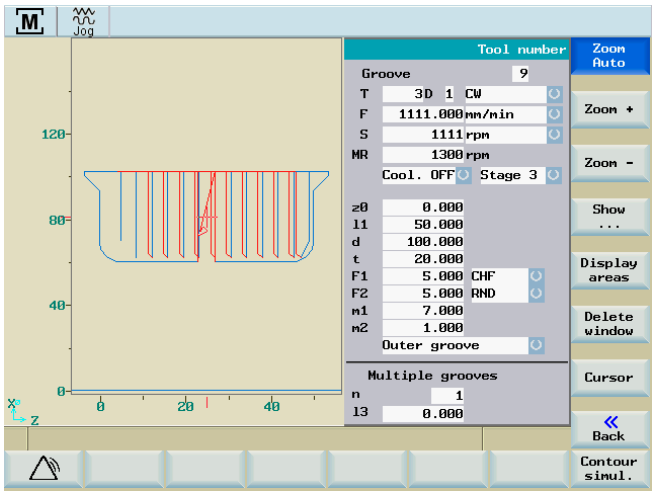


Figure 6-17 Simulation of a single cycle - standard simulation

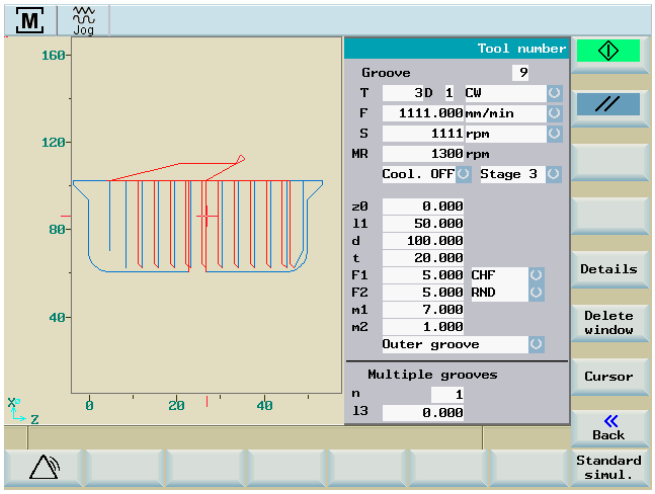


Figure 6-18 Simulation of a single cycle - contour simulation

References

A description of further operating options for a simulation can be found in the "Programming and Operating Manual SINUMERIK 802D sl Turning".

6.4 Executing the machining step program

Functionality

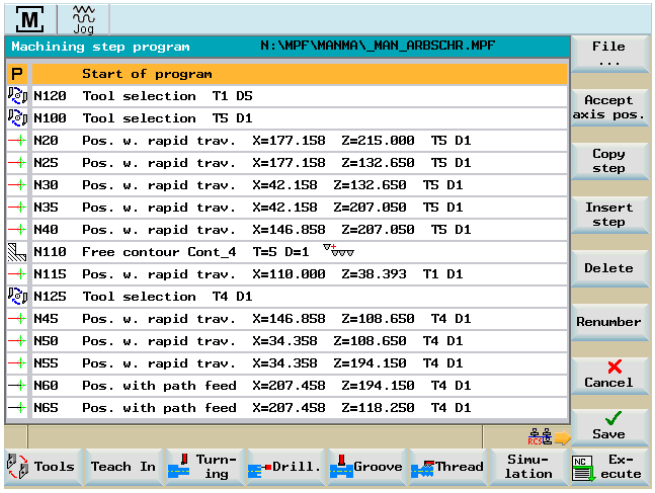


Figure 6-19 Machining step program

In the "Machining step program" function, you can toggle between the horizontal softkey functions "Execute" and "Exec. here" using the <ETC> key.

The two functions change from the "Machining step program" into that screen form in which the actual machining step program is to be executed:

The complete machining step program is executed with "Execute".

With "Exec. here", the program is executed from the actual cursor position in the machining step program (step that is marked in the program).

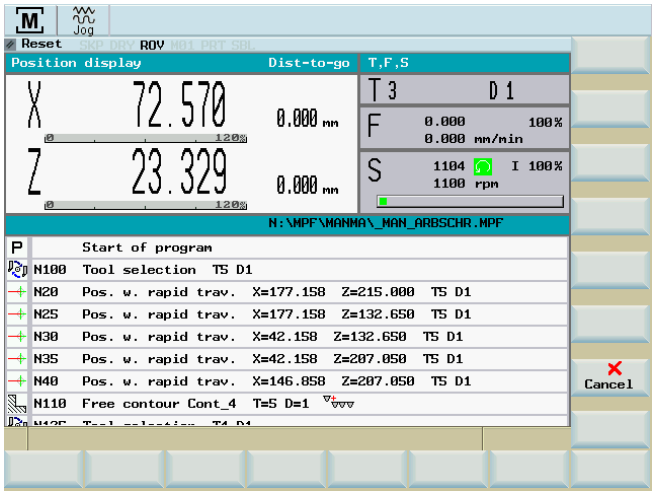


Figure 6-20 Execute the machining step program

Operating sequences, executing the machining step program

The current machining status is displayed in the center of the execute screen (see the screen "Executing the machining step program"). This status could be one of the following:

- Machining not started
- Machining active
- Machining aborted
- Machining interrupted
- Machining finished

In the example, the text "Machining not started" is displayed.

1. Start machining using the <NC-Start> key.

   The machining operation on the workpiece is executed.

   The individual machining steps are executed in the order in which you program them.

Note

Press the <NC-Stop> key if you want to interrupt the machining operation.

2. If machining was terminated (e.g. "machining completed"), then the execution screen can be exited using the "Cancel" softkey.

Note

The individual machining steps are performed with the currently programmed direction of spindle rotation! This need not coincide with the position of the spindle direction switch or with the spindle direction selected with the spindle keys!

Messages

7

7.1 Messages

Functionality

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

PLC alarm messages are specially configured for the "Manual Machine Plus" system in the shipped version of the "Manual Machine Plus" software.

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

700000	
700001	
700002	REFERENCE POINT ON X AXIS NOT REACHED
700003	REFERENCE POINT ON Z AXIS NOT REACHED
700004	
700005	
700006	
700007	POSITION OF SPINDLE DIRECTION SWITCH IS ILLEGAL
700008	INCORRECT SPINDLE DIRECTION ACTIVATED
700009	NO VALID SPINDLE/FEED TYPE
700010	RUNTIME MONITORING JOG SELECTION PROGRAM
700011	SPINDLE OVERRIDE NOT 100%
700012	SPINDLE NOT SWITCHED ON
700013	FEED OVERRIDE = 0%
700014	WITH/WITHOUT FEED?
700015	DIRECTION CANNOT BE REVERSED IN THREAD
700016	DRIVE NOT READY
700017	I2/T ALARM FROM I/RF MODULE
700018	MOTOR OVERHEATING
700019	COOLANT TOO LOW

700020	LUBRICANT MOTOR OVERHEATING
700021	LUBRICANT TOO LOW
700022	TURRET HEAD MOTOR OVERHEATING
700023	PROGR.TOOL NO. > MAX.TOOL NO. ON TURRET HEAD
700024	802D MACHINE CONTROL PANEL DEFECT
700025	BRAKE IS RELEASED FOR MOTOR OPTIMIZATION
700026	NO CHECKBACK FROM TURRET HEAD
700027	
700028	
700029	
700030	

Appendix

A

A.1 Feedback on the documentation

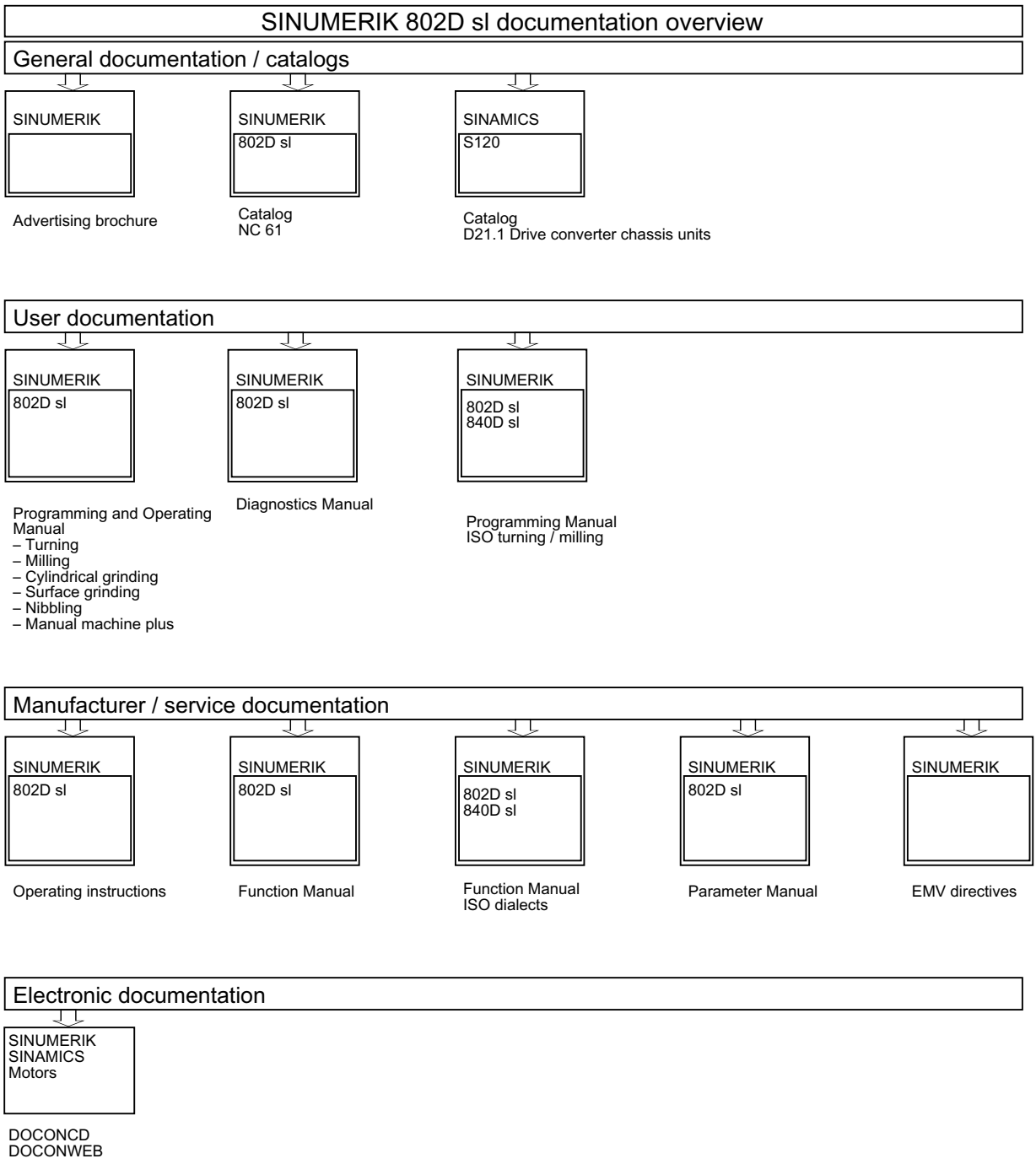
This document will be continuously improved with regard to its quality and ease of use. Please help us with this task by sending your comments and suggestions for improvement via e-mail or fax to:

E-mail:	mailto:docu.motioncontrol@siemens.com
Fax:	+49 9131 - 98 2176 Please use the fax form on the back of this page.

To	From
SIEMENS AG I DT MC MS1 P.O. Box 3180 D-91050 Erlangen / Germany	Name:
	Address of your company/department
	Street:
	Zip code: City:
	Phone: /
Fax: +49 9131 - 98 2176 (Documentation)	Fax: /

Suggestions and/or corrections

A.2 Overview of documentation



Index

A

Axis direction switch, 39
Axis-parallel traversal, 47

C

Contour list, 102
End face, 99
Peripheral surface, 99
cutting rate, 45

D

D value, 35
Drilling centric, 60

E

Cutting F, 99
Error displays, 10

F

F value/F type, 35
Feedrate type, 44

G

Gear stage preselection, 59
Groove cycle - multiple, 68, 75
Grooving, 65

H

Handwheels, 39
Hot keys, 12

I

INC value, 35

L

LED displays on the CNC operator panel (PCU), 10
Limit stop position, 26
Limit stops, 26, 29

M

Machine contour, 104
Machining step program, 107
  Simulation, 117
  Teach In, 114
  Tool change, 111
Manual Machine Plus, operating area, 34
Manual softkey, 18
Multiple tapping, 70

O

Operator control and display elements, 9

P

Parting, 69
Cutting F, 99
Position display, 34

R

Radius turning, 49
Radius turning type A, 52
Radius turning type B, 52
Radius turning type C, 53
Reference point, 21
Reference point approach, 19
Revolutional feedrate, 44
Roughing cycle A, 85
Roughing cycle B, 88
Roughing cycle C, 91
Roughing cycle D, 94
Roughing cycle E, 97

S

S value/S type, 35
Spindle, 40
Spindle speed, 40, 45
Spindle speed limitation, 46
Spindle type, 44
Spindle value, 43
Status displays, 10

T

T value, 35
Taper turning, 48
Tapping, 62
Thread cutting, 76
Thread recutting, 81
Time feed, 44
Tool change, 42
Tool measurement, 23
Travel direction, 35
Traverse axes, 20