SIEMENS

SINUMERIK
SINUMERIK 808D ADVANCED
Programming and Operating Manual (Manual Machine Plus (MM+), Turning)
User Manual

Legal information

Warning notice system

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

⚠ DANGER

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

⚠ WARNING

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

⚠ CAUTION

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

NOTICE

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

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

Qualified Personnel

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

Proper use of Siemens products

Note the following:

⚠ WARNING

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

Preface

Applicable products

This manual is valid for the following control systems:

Control system	Software version
SINUMERIK 808D ADVANCED T (Turning)	V4.90: PPU16x.3/PPU15x.3 with spindle/feed servo system

Documentation components and target audience

End-user documentation	Target audience
Programming and Operating Manual (Turning)	Programmers and operators of turning machines
Programming and Operating Manual (Milling)	Programmers and operators of milling machines
Programming and Operating Manual (ISO Turning/Milling)	Programmers and operators of turning/milling machines
Programming and Operating Manual (Manual Machine Plus (MM+), Turning)	Programmers and operators of turning machines
Diagnostics Manual	Mechanical and electrical designers, commissioning engineers, machine operators, and service and maintenance personnel
Manufacturer/service documentation	**Target audience**
Commissioning Manual	Installation personnel, commissioning engineers, and service and maintenance personnel
Function Manual	Mechanical and electrical designers, technical professionals
Parameter Manual	Mechanical and electrical designers, technical professionals
Service Manual	Mechanical and electrical designers, technical professionals, commissioning engineers, and service and maintenance personnel
Readme file	
Third-party software - Licensing terms and copyright information	

My Documentation Manager (MDM)

Under the following link you will find information to individually compile your documentation based on the Siemens content:

www.siemens.com/mdm

Standard scope

This manual only describes the functionality of the standard version. Extensions or changes made by the machine tool manufacturer are documented by the machine tool manufacturer.

Technical support

Country	Hotline [1]	Further service contact information:
Germany	+49 911 895 7222	• Worldwide Web site: https://support.industry.siemens.com/cs/ww/en/
China	+86 400 810 4288	• Chinese Web site: http://www.siemens.com.cn/808D

[1] You can find more hotline information at the worldwide Web site given above.

EC Declaration of Conformity

The EC Declaration of Conformity for the EMC Directive can be found on the Internet at https://support.industry.siemens.com/cs/ww/en/.

Here, enter the number "**59843164**" as the search term or contact your local Siemens office.

Table of contents

1 Fundamental safety instructions

1.1 General safety instructions

WARNING

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

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

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

WARNING

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

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

- Protect the parameterization (parameter assignments) against unauthorized access.
- Handle possible malfunctions by taking suitable measures, e.g. emergency stop or emergency off.

1.2 Warranty and liability for application examples

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

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

1.3 Industrial security

Note

Industrial security

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

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

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

For additional information on industrial security measures that may be implemented, please visit:

Industrial security (http://www.siemens.com/industrialsecurity)

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

To stay informed about product updates, subscribe to the Siemens Industrial Security RSS Feed at:

Industrial security (http://www.siemens.com/industrialsecurity)

Further information is provided on the Internet:

Industrial Security Configuration Manual (https://support.industry.siemens.com/cs/ww/en/view/108862708)

WARNING

Unsafe operating states resulting from software manipulation

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

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

2 Turning on, Reference Point Approach

Note

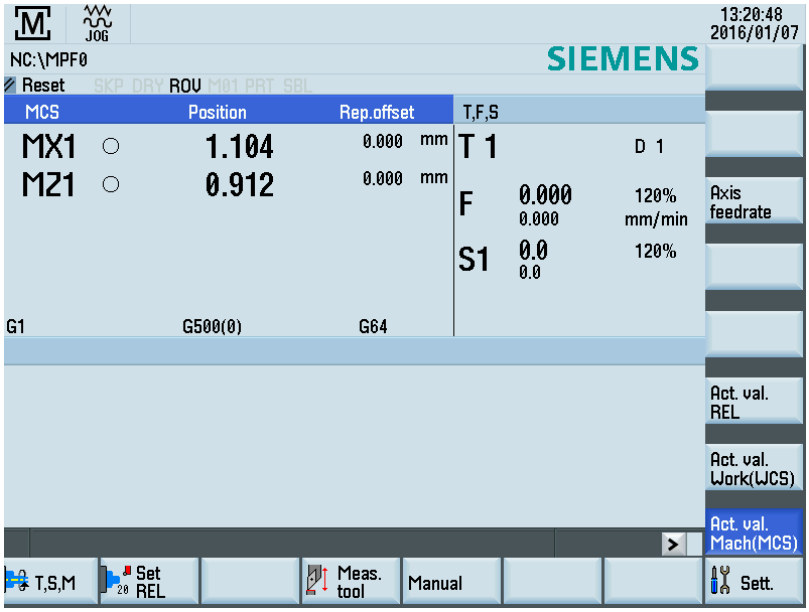
If the controller has already been preconfigured to Manual Machine Plus (MM+) by the machine manufacturer, the main screen for MM+ is activated once the controller has been started up.

This operating area runs only in SIEMENS mode instead of ISO mode.

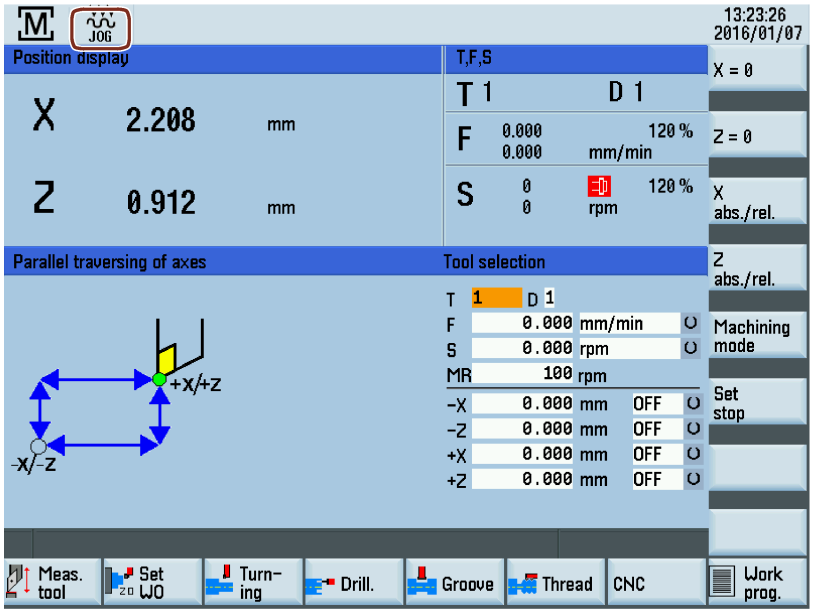
Operating sequence

Note that if MD1105 = **1**, after power on, the control system automatically opens the main screen for MM+ only when you set the startup operating area to the machining operating area.

1. Press this key on the machine control panel to open the "JOG" window.



2. Press this softkey to open the MM+ operating area.



3. If the axes have not approached the reference point, press this key to change to the "REF.POINT" window for reference point approaching operation; otherwise, skip this step.

   For more information on reference point approaching, see Chapter "Switching on and referencing" of the SINUMERIK 808D ADVANCED Programming and Operating Manual (Turning).

4. Press this softkey to exit the operating area.

Note

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

3 Setting-up

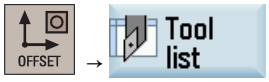
3.1 Measuring tools

Functionality

You can measure tools manually in the MM+ operating area. In this case, the manual tool measurement function accesses the tool list data.

Note

To access the tool list, proceed as follows:



NOTICE
Tool breakage or workpiece damage 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 following hardkey. If this key is pressed, the tool will be changed.

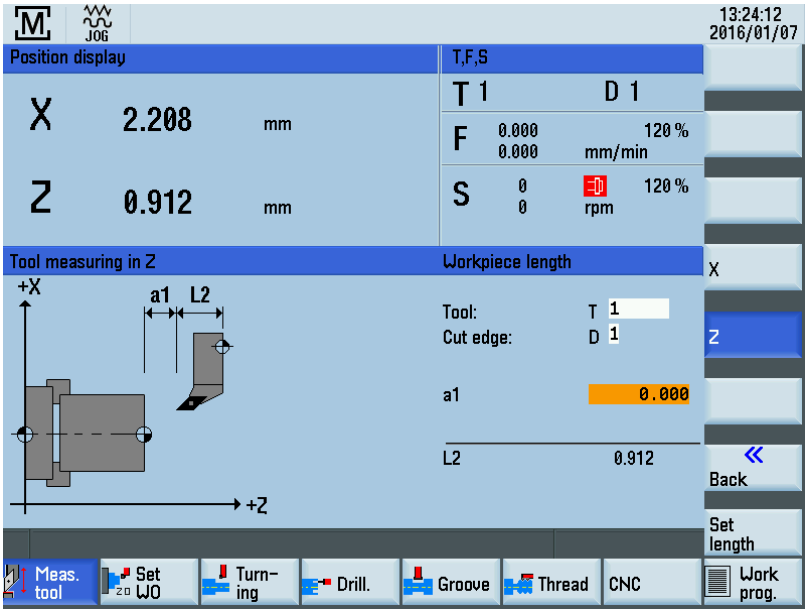
Note

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

Operating sequence

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

1. Press this softkey. The following screen appears.



2. Press this softkey.

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

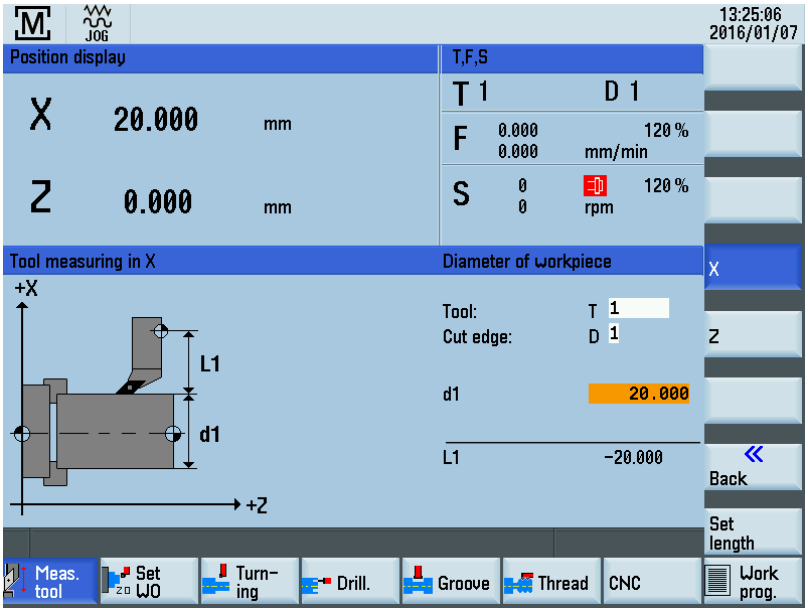
3. Check that the current tool number appears in the display field for the tool, since the calibration operation will relate to this tool.
4. Carefully "scratch" a workpiece by an X handwheel infeed when the spindle is turning.
5. Move the slide slightly (without changing the X position) along the Z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

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

Set length

9. Press this 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.



Z

10. Press this 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.

« Back

11. Press this softkey to return to the main screen for MM+.

Note

New offset settings lost

If you exit the screen form in step 8, the new offset will not take effect.

3.2 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 MM+ operating area, it is possible to turn simple shoulders (including tapers) without the need for any further cycle parameterization.

Supplementary conditions

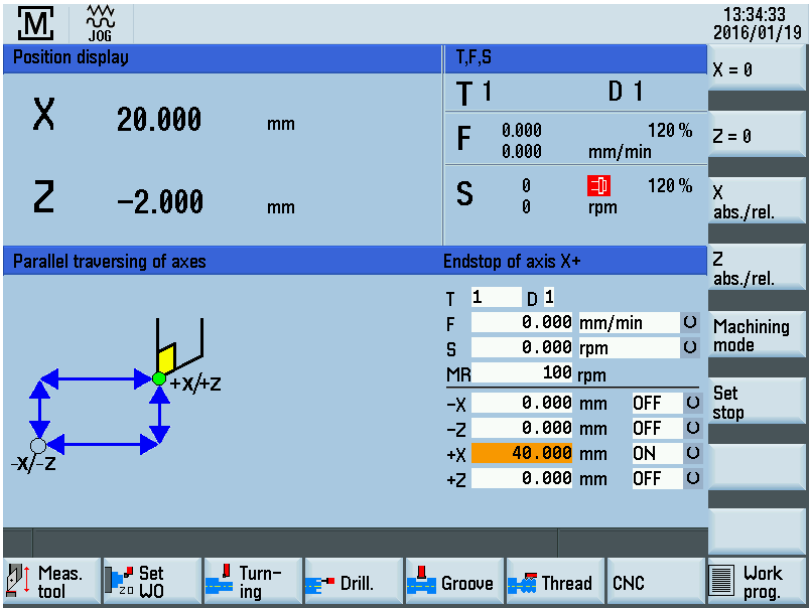
- The limit stop position is always an absolute dimension, which in turn always corresponds to the position in the absolute actual value display on the MM+ interface. A relative limit stop position is not possible.
- A limit stop position can be entered/accepted only when the axes are stationary; otherwise,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

3.2.1 Setting and activating/deactivating limit stops

Functionality

In the main screen of MM+, limit stops can be entered into the input fields "-X/-Z/+X/+Z".

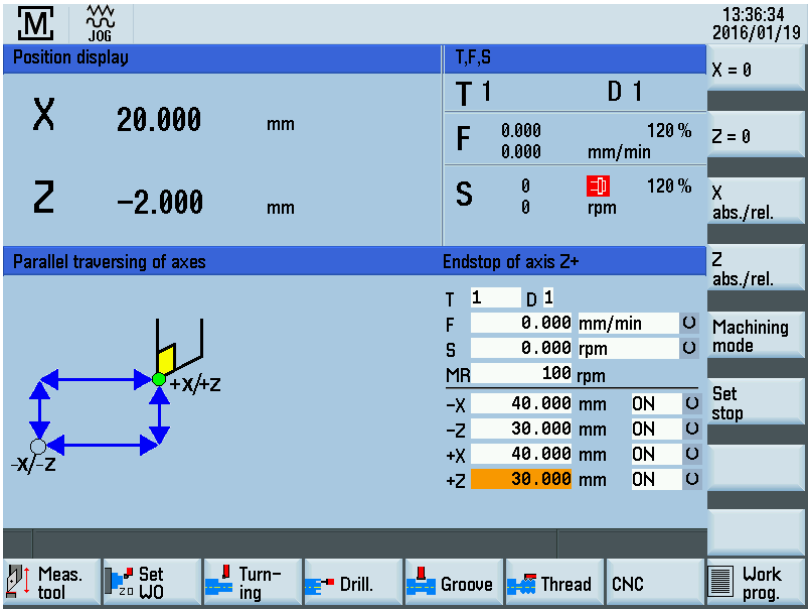
In the following screen, the cursor is located in the desired input field (with an orange background).



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 when both of the following are met: • 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 when both of the following are met: • 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 when both of the following are met: • 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 when both of the following are met: • 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.



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 above hardkey 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 (for example, <-Z> or <+X/-X/+Z>).

  - Press the above softkey.
    The current actual position of the relevant axis is transferred to the input field.

Activating/disabling limit stops

The limit stops are activated/deactivated individually using the above hardkey.

Here, you can select ON or OFF.

Note

After activating a limit stop, set MD20152[59] = 1 to retain the current setting even after you press the following key.

3.2.2 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 in the Z direction. The blank diameter is 70 mm.

Operating sequences for infeeding to stop

1. Position the axes in front of the workpiece (for example, 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.
   A message is displayed, saying that limit stop -Z is reached.
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.
    A message is displayed, saying that limit stop +Z is reached.
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.
    A message is displayed as the tool is fed in, saying that limit stop -X is reached.
    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 a message saying that limit stop -X is reached 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.
   A message is displayed, saying that limit stop -Z is reached.
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.

3.3 Setting the workpiece zero

Functionality

The function of setting the workpiece zero 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" (for example, an end face).
2. Clamp the blank.
3. Scratch the relevant surface which corresponds to the "virtual zero point".
4. Use the function of setting the workpiece zero to adapt the workpiece coordinate system to the parameterized machining operation.

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

Additional information

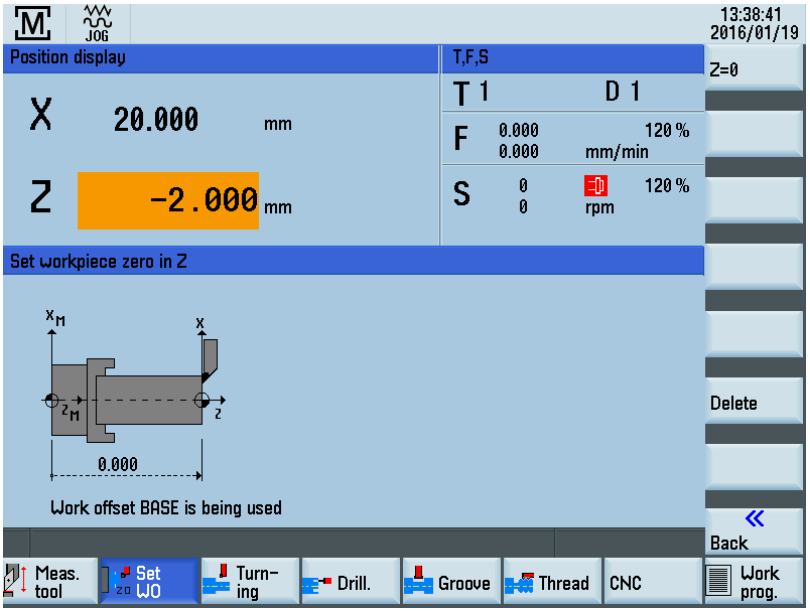
The following operations are performed automatically when you select this softkey:

- The work offset is automatically calculated according to the current axis position in the longitudinal axis (Z), entered in the control system memory as the basic 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 be automatically entered in the control system memory as the basic offset. The workpiece coordinate system display will change to reflect this.

NOTICE
Setting the workpiece zero carefully 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

1. Press this softkey in the main screen for MM+.



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

2. Press this softkey 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 control system.

3. Press this softkey to reset the work offset that is currently stored on the control system. The value "0.000" is entered in the basic offset memory location. However, all other offsets and the active tool offset remain unchanged.

4 Manual machining

4.1 Fundamentals of manual machining

Functionality

You can perform the following machining operations manually:

- Axis-parallel traversal
- Taper turning
- Radius turning
- Drilling - centered
- Tapping
- Groove cycles/Expan. groove
- 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

4.2 Display and operator control options in the main screen

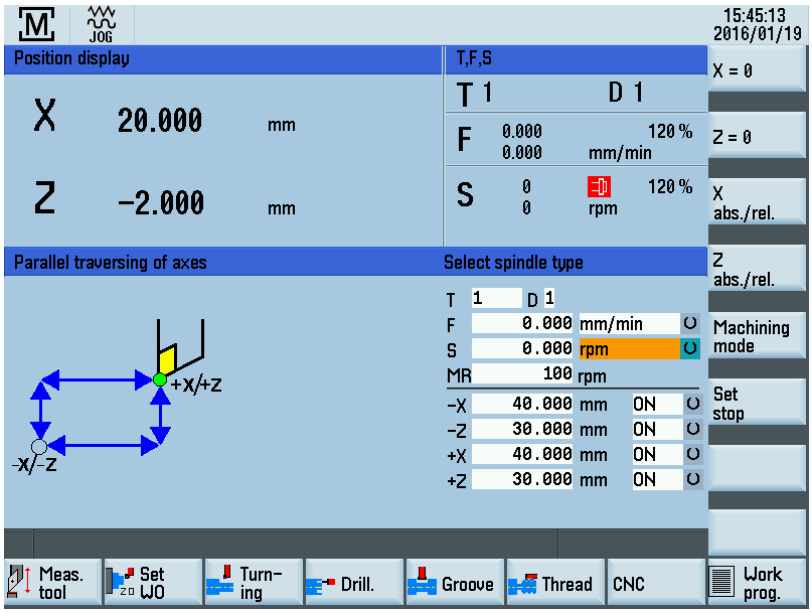
Functionality

Note

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

You can reference the axes in the Siemens standard user interface as well as in the operating area MM+.

You have referenced the axes and accessed the MM+ operating area. The following screen represents the main screen of this operating area.



Note about the position display in the main screen for MM+

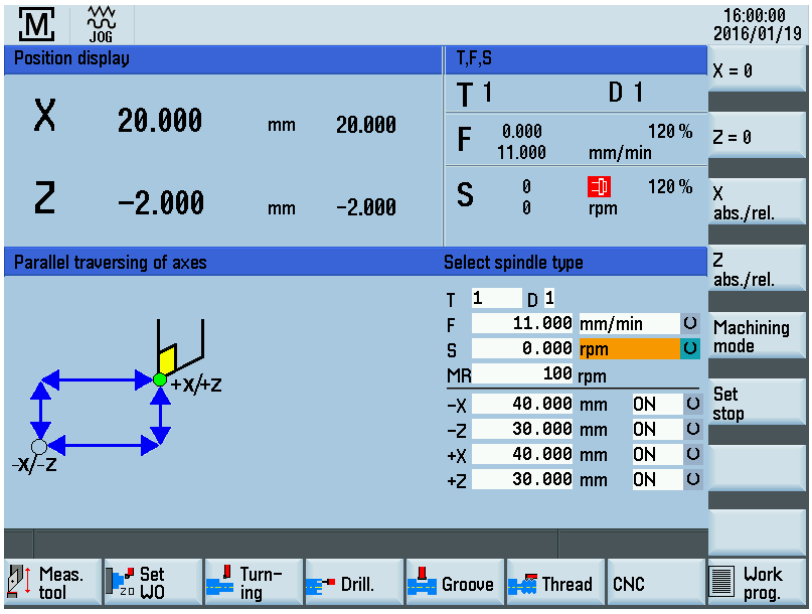
- Absolute position display active:
  The position value displayed in the large-size font is the absolute position. No additional value is shown.
- Relative position display active (see following diagram):
  The position value displayed in the large-size font is the relative position. The position value 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 17) for the X and Z axes, or
  - Axis direction switch (Page 17)
- The spindle is controlled by:
  - Spindle direction of rotation switch (Page 18)

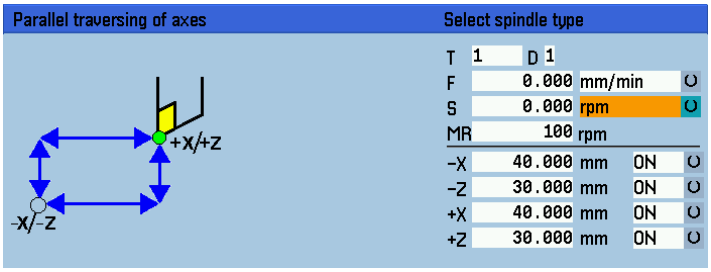
Values displayed in the position display and T, F, S



Displayed values	Meaning
In front of axis letters +/-	• Current traversing direction of axes
S value/S type %	• Programmed value for the spindle speed (rev/min) in either "rev/min" or "m/min", depending on 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
	• 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:



The machining technology data is 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 the choice of units rpm (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"

Note

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

NOTICE

Device damage caused by excessive chucking device speed

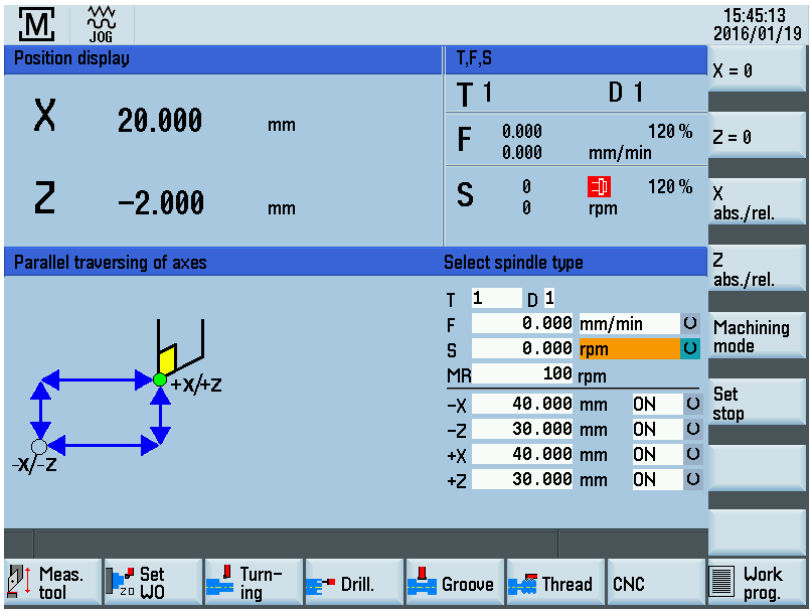
When constant cutting rate (G96) is selected, the maximum permissible spindle speed, corresponding to the fitted tool chucking device must be entered in the input field MR (spindle speed limitation).

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

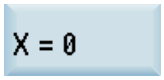
4.2.1 Toggling the display

Functionality

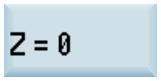
In the position display screen, you can edit the displayed values using the vertical softkeys.



Softkeys



Change the display to the relative position display mode and reset the display in the X axis.



Change the display to the relative position display mode and reset the display in the Z axis.

Toggle the display between the absolute position display mode and the relative position display mode in the X axis.

Toggle the display between the absolute position display mode and the relative position display mode in the Z axis.

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 MM+.

The current actual position of the relevant axis is transferred to the selected input field (-X/-Z/+X/+Z).

4.2.2 Machining with the handwheels

Functionality

The handwheels for the X and Z axes 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.

Note

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

4.2.3 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 above hardkey once to resolve the problem.

NOTICE
Damage to equipment due to an incorrect setting of increment weighting
An incorrect increment weighting setting can result in damage to the workpiece, tool and machine.
• Set increment weighing carefully.

4.2.4 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 an axis is traversed depends on the settings in the input fields for machining technology data.

The axis feedrate is also influenced by the feedrate override weighting setting and, depending on the option selected in the input fields for machining technology data (revolutional feed/cutting speed), by the spindle override weighting.

If the above hardkey 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.

Note

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.

4.2.5 Spindle advance/reverse

Functionality

NOTICE
Damage to equipment due to improper spindle value settings The last value settings are active (this depends on the machinery construction of a machine manufacturer). Improper spindle value settings may cause damage to equipment. • Check the spindle value before starting the spindle (for example, when changing the tool).

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

Note

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

NOTICE
Damage to equipment due to improper operation of the chuck guard Improperly altering or adjusting the chuck guard/chuck guard switch may cause damage to equipment. • 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 (for example, 50 %).

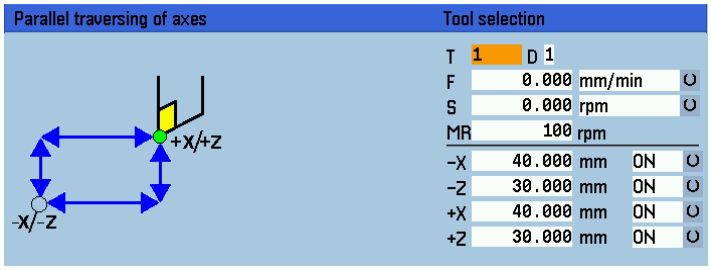
4.2.6 Tool change

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 MM+ main screen.

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



Note

The following display machine data defines the display:

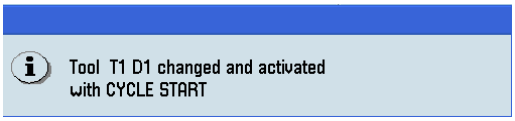
- MD290 CTM_POS_COORDINATE_SYSTEM
  - = 0 -> Position of the tool after the turning center (refer to the figure above)
  - = 2 -> Position of the tool before the turning center
- MD1104 TOOL_CHG_MANUALMODE_MA
  - = 0 -> Editing of the "T" and "D" fields is not possible, the fields are grayed out
  - = 1 -> Editing of the "T" and "D" fields is possible

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



   Acknowledge this information text using this softkey.

4. Press this hardkey.
   The tool number 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.

Note

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

Note

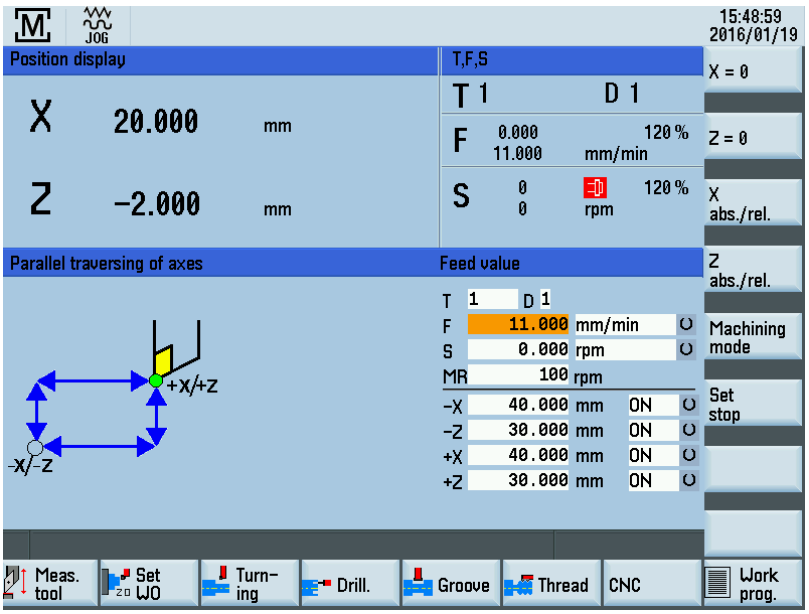
The tool number entered in the T value field must correspond to the tool loaded into the machine; otherwise, the tool will have to be recalibrated (see also section "Measuring tools (Page 6)"). An uncalibrated or incorrectly calibrated tool can lead to dimensional errors or to incorrect cutting values.

4.2.7 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 in the main screen for MM+.



2. Edit the programmed value using the numeric keys.

3. Press this hardkey.
   The value is activated.

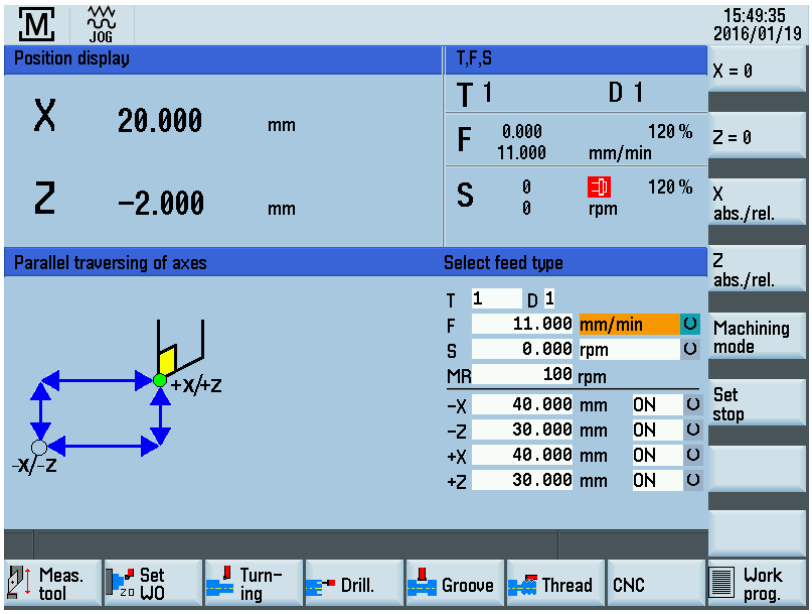
Note

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

4.2.8 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 (with an orange background).



By pressing the above hardkey, 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.

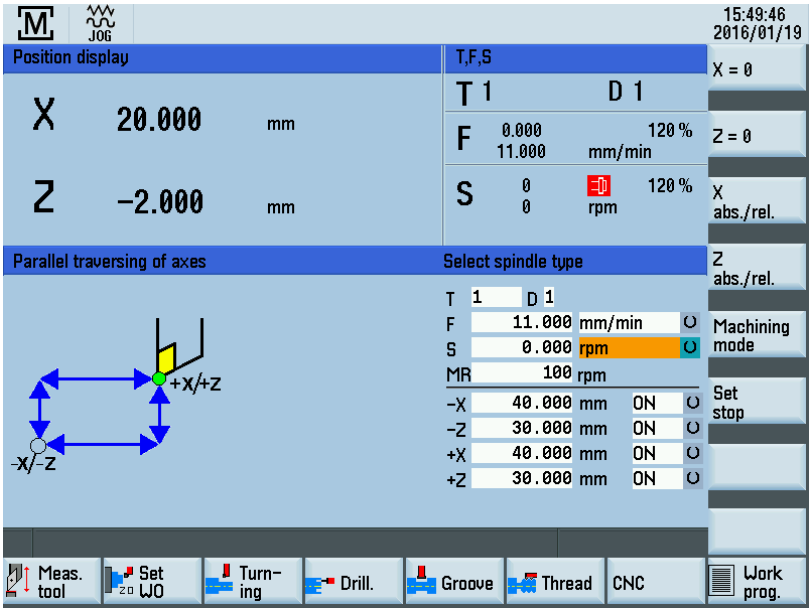
  The time feed is only possible for a constant spindle speed.

- Rotary feedrate (mm/rev)

  In the "spindle speed + revolutional feedrate" or "constant 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 (with an orange background).



By pressing the above hardkey, you can choose one of the following spindle types:

- Constant spindle speed (rpm)

  This value defines the programmed spindle speed for machining in the "spindle speed + time feed" or "spindle speed + revolutional feedrate" mode.

  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 in the "cutting speed + revolutional feedrate" mode. 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.

Note

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

4.2.9 Change the speed limitation for constant cutting rate

Change the speed limitation operating sequences

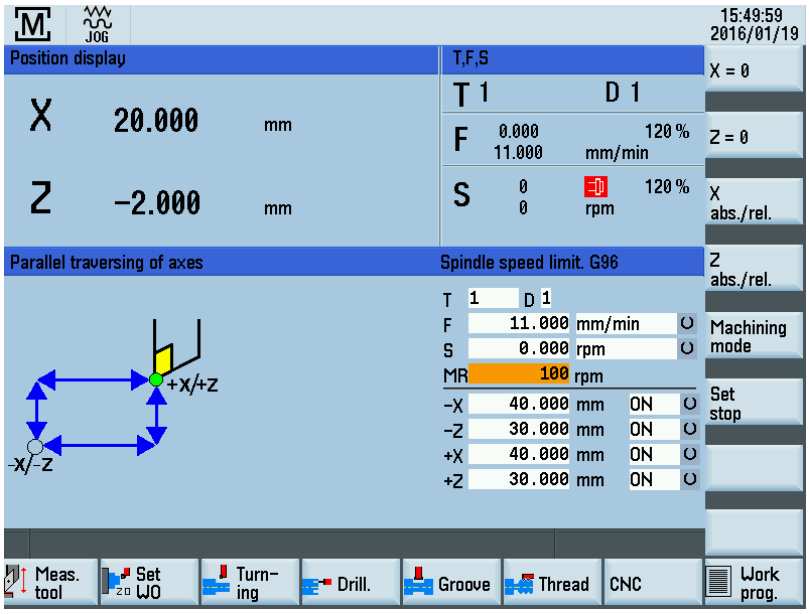
When a constant cutting rate (G96) is programmed, the maximum permissible spindle speed, corresponding to the fitted tool chucking device, must be entered in the input field "MR" (spindle speed limitation).

⚠ WARNING

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 MM+ main screen.



2. Edit the programmed value using the numeric keys.
3. Press this hardkey.
   The value is activated.

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

4.3 Manual machining with machining types

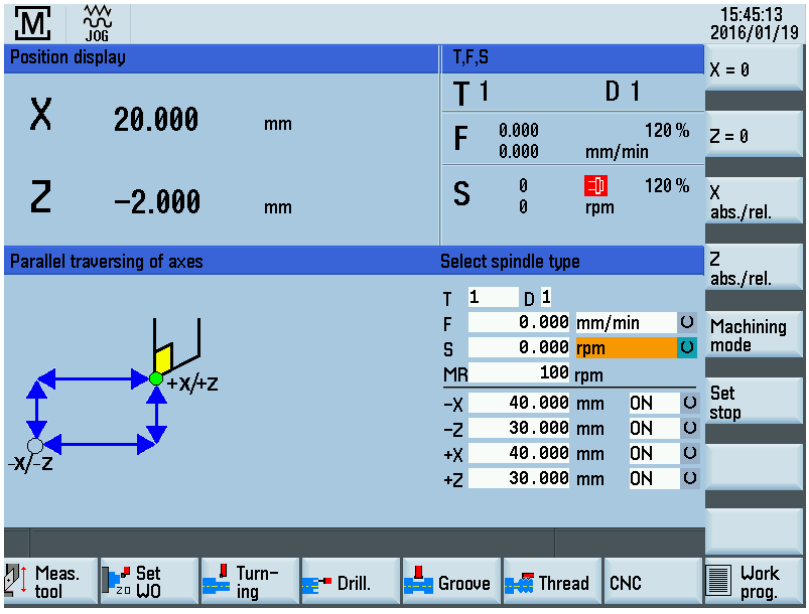
4.3.1 Axis-parallel traversal

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 axis-parallel traversal function via the main screen MM+.
2. If a different machining mode is active, press this softkey until the following mode screen is displayed.



4.3.2 Manual taper turning

Functionality

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

For this machining type, 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.

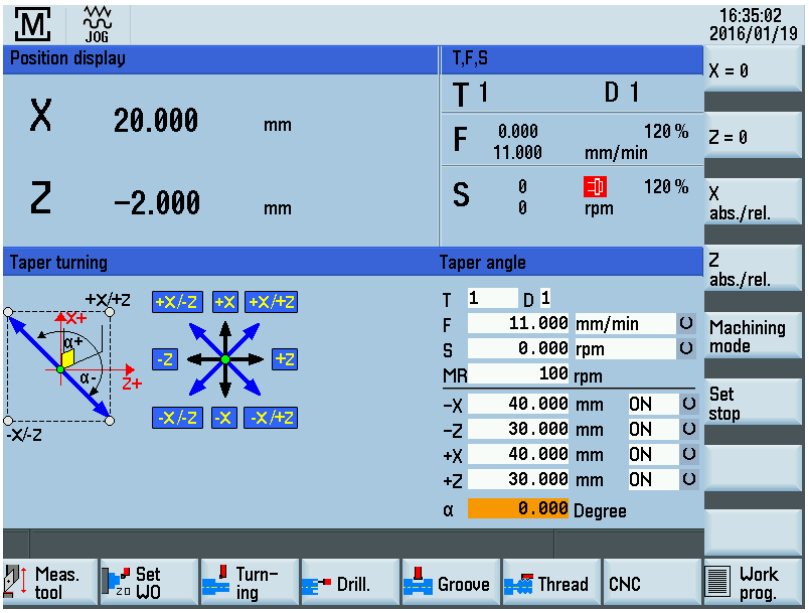
Note

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 MM+.
2. Press this softkey until the following screen is displayed.



3. The input field for the taper angle "α" is immediately displayed with an orange 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 this hardkey.

INPUT

The taper angle remains active until you exit this machining mode by pressing this softkey.

Machining mode

4.3.3 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.

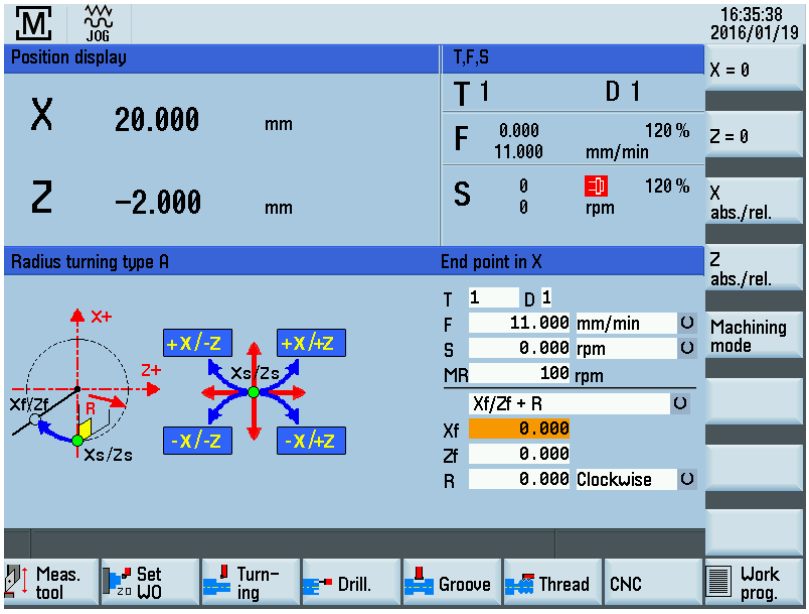
Note

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 MM+.
2. Press this softkey until the following machining mode screen is displayed.

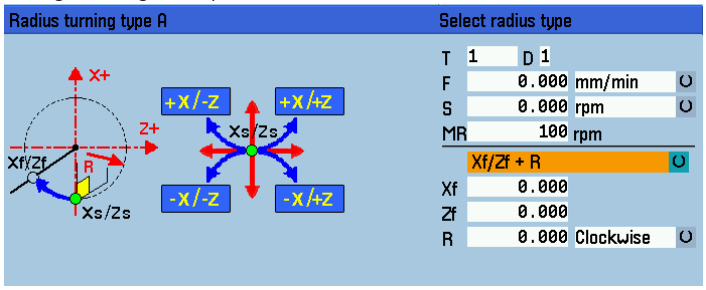


You can exit the machining mode by pressing this softkey again.

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 (with an orange background).



4. By pressing this hardkey, you can select the radius type.

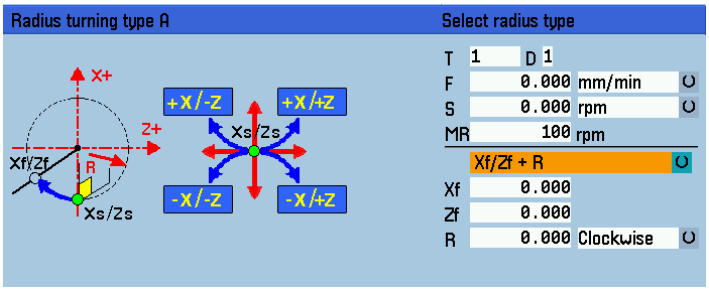
NOTICE
Damage to equipment due to incorrectly input values Omitting or using the wrong sign for the input values or entering the wrong arc direction can lead to a collision and may destroy the tool or the workpiece. • Input values correctly.

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.

4.3.3.1 Radius turning type A

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

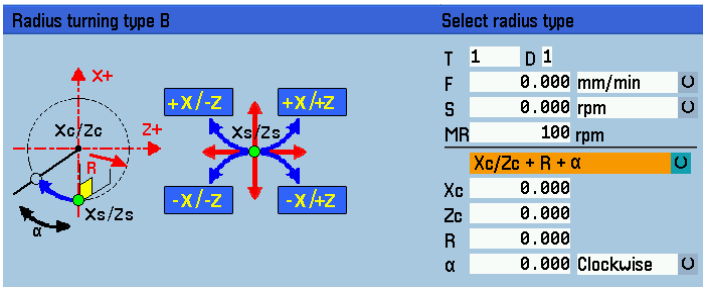


Parameter

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 counter-clockwise direction.

4.3.3.2 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.

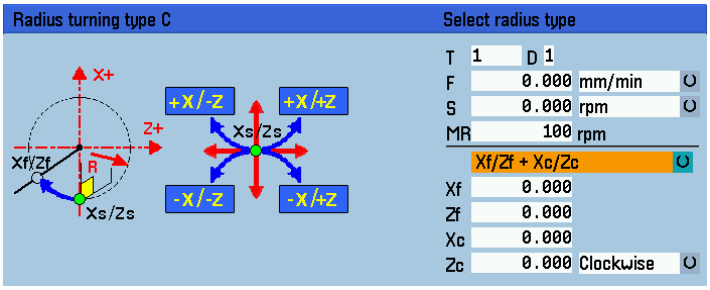


Parameter

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 counter-clockwise direction.

4.3.3.3 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.



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.

4.4 Manual machining using cycles (functions)

4.4.1 Principle operating sequence

Functionality

You can perform the following functions manually:

- Drilling centric
- Tapping
- Groove cycles/Expan. groove
- 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:

NOTICE
Damage to equipment due to spindle rotating in the wrong direction In machining operations, 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 the following key. 

- The spindle is rotating in the correct direction.
- Any axis position from which the workpiece position is machined can be approached without risk of collision.
- All parameters for the cycles are correctly assigned.

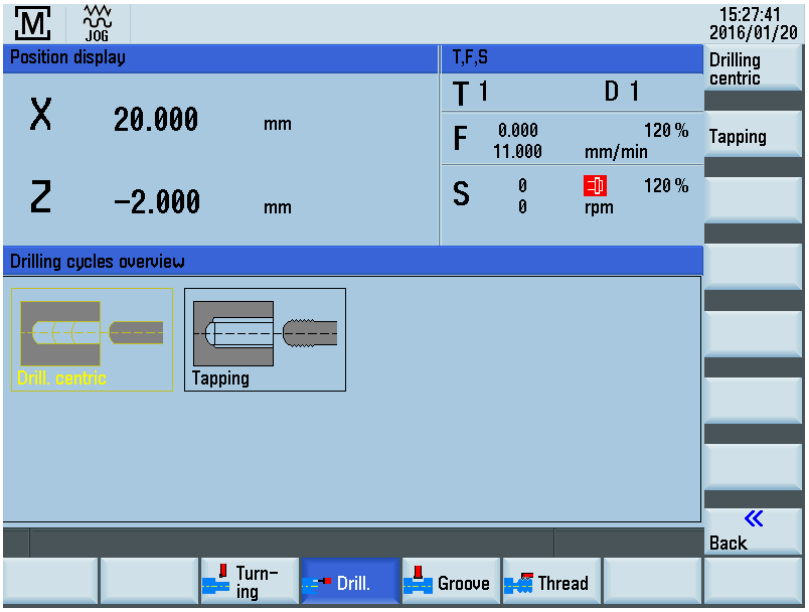
Note

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

Operating sequences

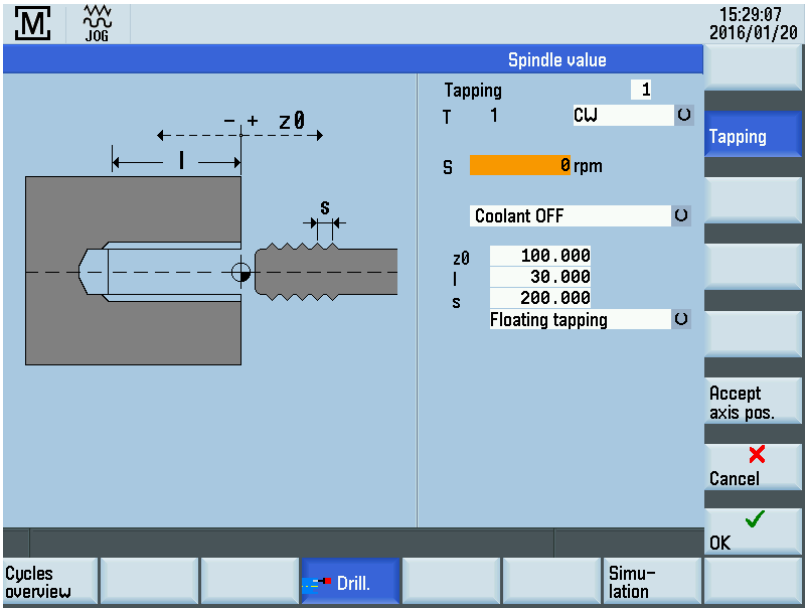
Drill.

1. Press this softkey in the main screen of MM+.



Tapping

2. Press this softkey and then parameterize the function.



Accept axis pos.

3. This softkey will support you with the parameterization and execution of functions.

   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, an error message is displayed when you accept the axis position, saying that the value cannot be accepted.

Cancel

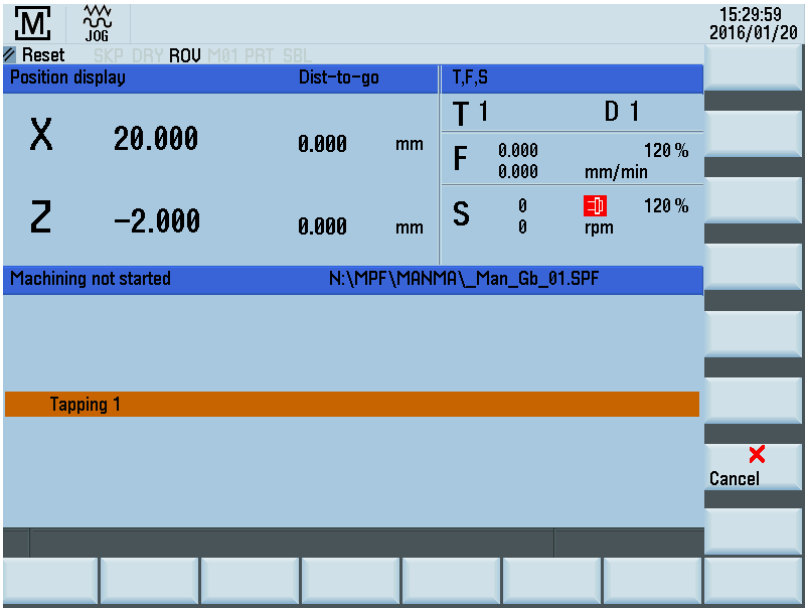
   This softkey takes you back to the main screen.

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

4. Your inputs are accepted when you press this softkey.

Your inputs are discarded when you press this softkey.

5. The function was parameterized (for example, thread tapping).
Activate the function using this softkey.
The following executing screen appears:



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

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

6. Start machining using this hardkey.
The machining operation on the workpiece is executed.

Press this hardkey if you want to interrupt the machining operation. The selected direction of the spindle rotation continues to be activated.
By pressing this hardkey, "JOG" operating mode is automatically changed, that is, you can traverse the axes manually. By continuing executing a cycle start , the interruption point is approached again and execution of the program is continued.

7. If machining was terminated (for example, completed), the execution screen can be exited using this softkey.

Note

In step 2, a detailed parameter description of each function can be found in the relevant sections.

4.4.2 General parameters

General parameters

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

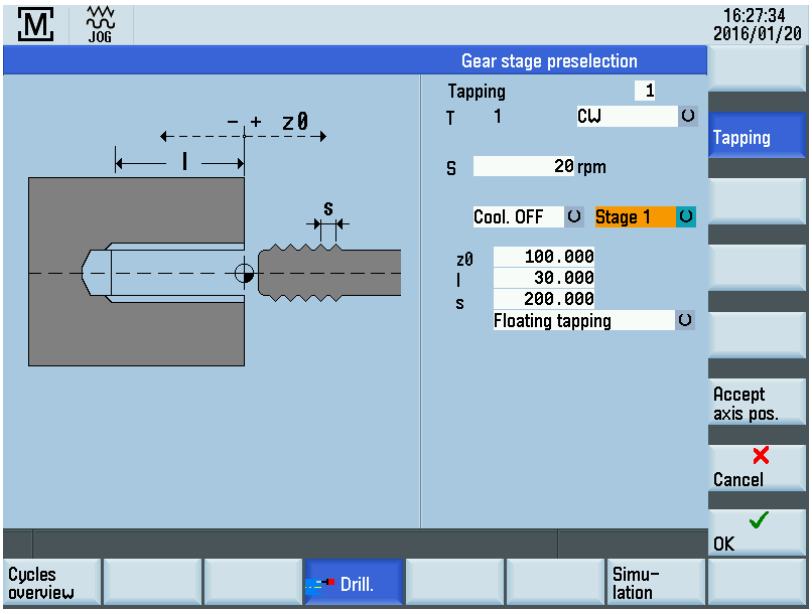
Parameter	Description
Function name	Number of the selected function
Tool (T)	Tool number
Compensation (D)	Tool offset number
Direction of spindle rotation	Toggle field for spindle direction of rotation (clockwise/counterclockwise)
Feed value (F)	Feed value
Type of feedrate	Toggle field for feed type (mm/min or mm/rev)
Spindle speed (S)	Spindle speed value
Spindle type	Toggle field for spindle type (rev/min)
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, and finishing only.
Position	The machining direction can be selected with the first toggle field. The following options are available: outside right, inside right, and 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	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 pre-selection

In the input fields for the relevant manual machining, for example, thread tapping, it is possible to pre-select the gear stage.



If a gear unit is installed on the machine, you can select the gear stage by using the following hardkey:

Note

The selection of the gear stages can be changed by the following general machine data:

- $MN14512 USER_DATA_HEX[31]
  - 1's digit -> Number of selectable gear stages
  - 10's digit -> = 1 -> With selection of the automatic gear stage

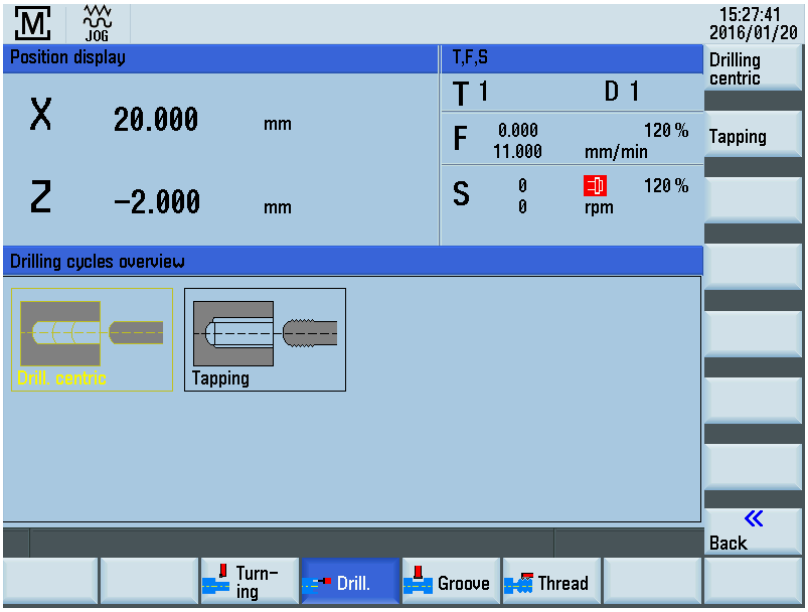
4.4.3 Manual drilling centered

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

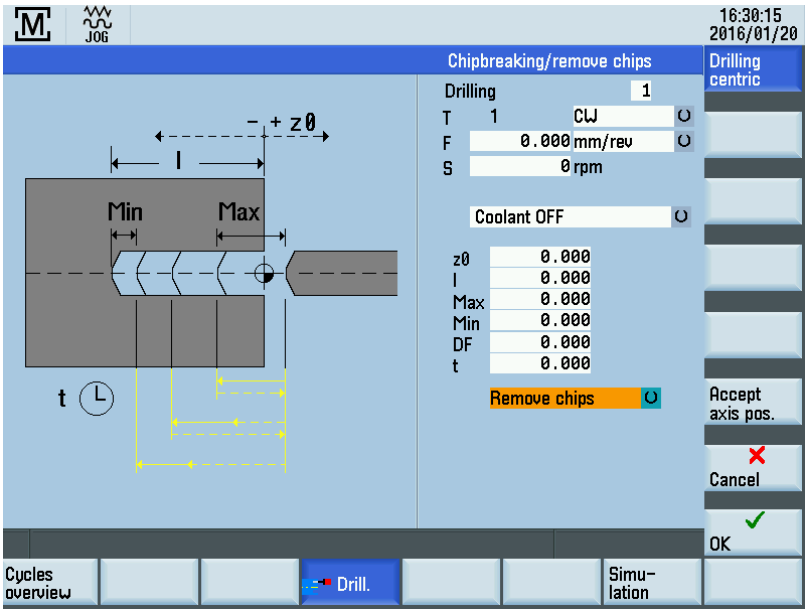
1. You can access the drilling cycle overview by pressing this softkey in the basic screen for MM+.



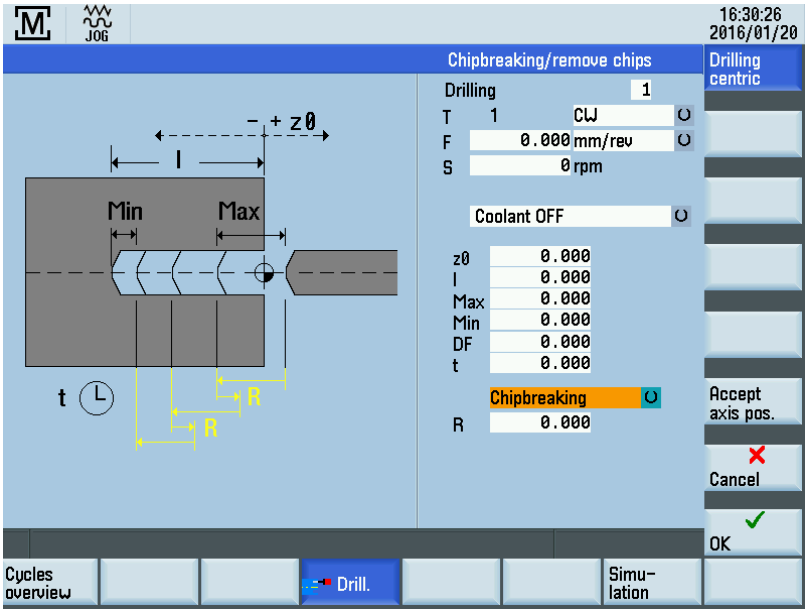
2. You can access the "Manual center drilling" function by pressing this softkey in the drilling cycle overview.

Alternatively, you can select this function with cursor keys and activate it with this hardkey.

Remove chips:



Chip breaking:



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

Parameter	Description
Return travel distance (chip breaking only) (R)	Return travel distance in the longitudinal axis for chip breaking.
Chip breaking/remove chips	With this toggle field you can select between machining with "chip breaking" or "remove chips". 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 "remove chips" 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.

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 "remove chips" has been selected. With "chip breaking", the tool is retracted in the longitudinal axis by the value set in the "return travel" parameter; with "remove chips", 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/remove chips" 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.

4.4.4 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 the 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, that is, X=0)

NOTICE

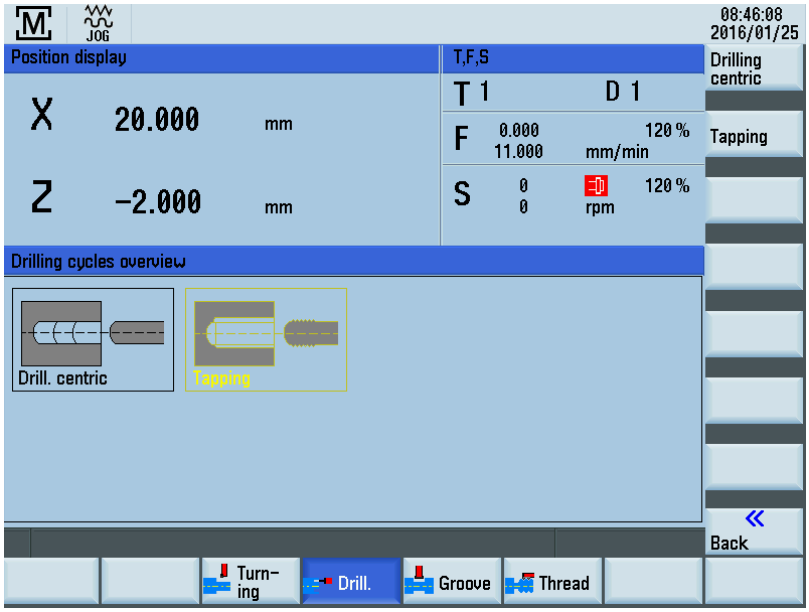
Setting the spindle override weighting to 100%

If the time feed is selected in the input fields of machining technology data, for the sake of 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 pressing the following hardkey, check that the spindle override weighting is set to 100%.

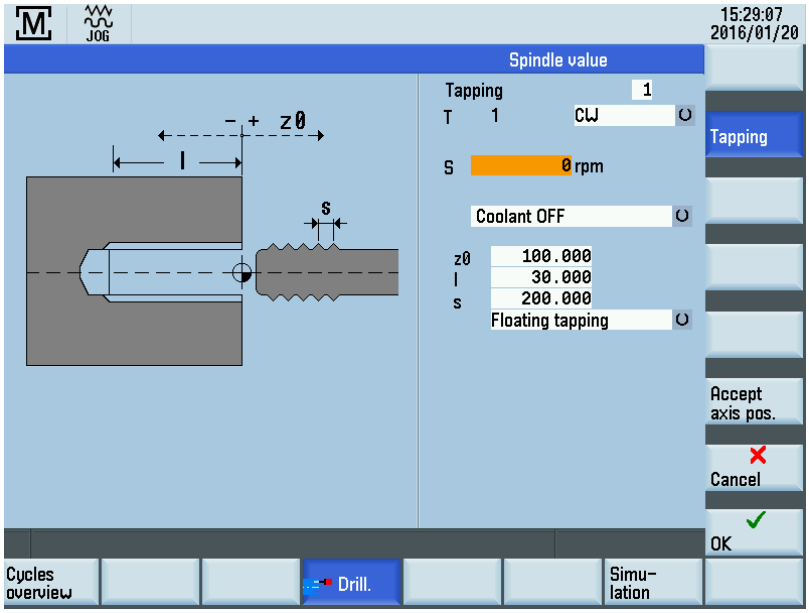
Operating sequences

1. You can access the drilling cycle overview by pressing this softkey in the basic screen for MM+.



2. You can access the manual tapping function by pressing this softkey in the drilling cycle overview.

   Alternatively, you can select this function with cursor keys and activate it with this hardkey.



Parameters

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 a 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.

4.4.5 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.

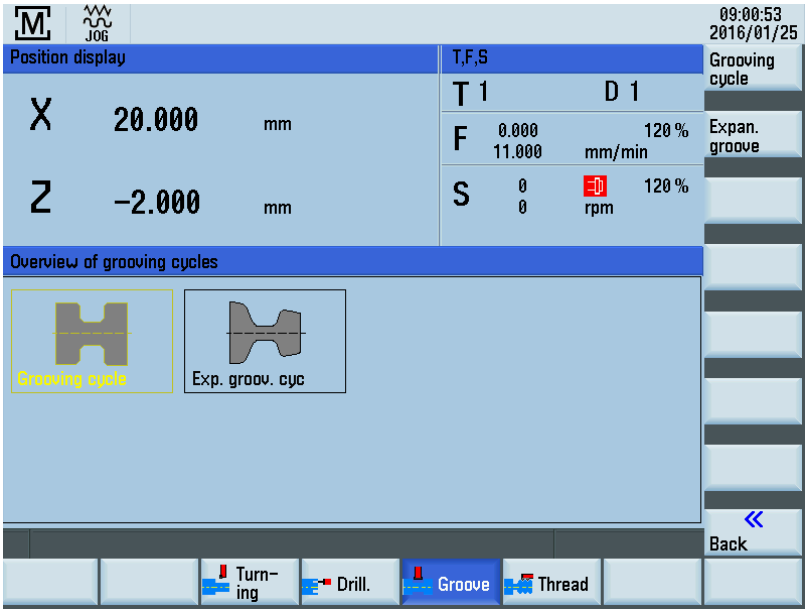
Note

The grooving tool must be configured in the tool list; in doing so, the width of the tool must be programmed via the panel width or via cutting edge 1 (D1) or cutting edge 2 (D2).

4.4.5.1 Groove cycle - single

Operating sequences

1. You can access the grooving cycle overview by pressing this softkey in the basic screen for MM+.



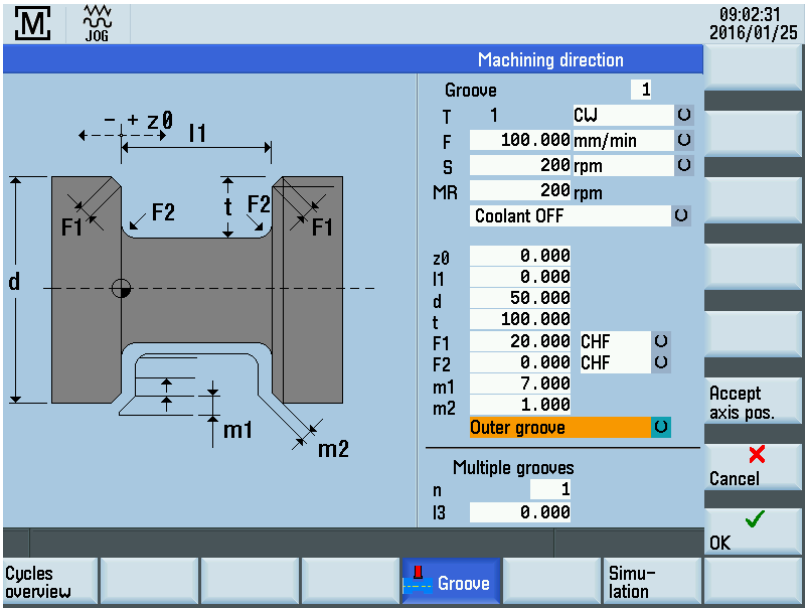
Grooving cycle

INPUT

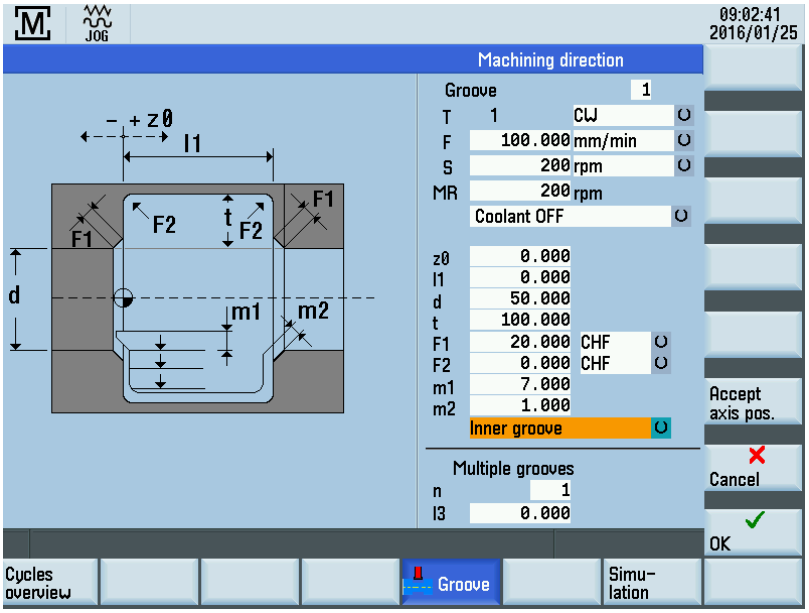
2. You can access the manual grooving function by pressing this softkey in the grooving cycle overview.

Alternatively, you can select this function with cursor keys and activate it with this hardkey.

Outer groove:



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 indicates 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 "CHF" and "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 "RND") or an input chamfer (display "CHF") of less than 45^0 on both sides of the groove. You can toggle between RND and 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 "RND") or a chamfer (display "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 and 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 input fields of machining technology data before the start of the cycle.
7. Finally, the original position of the axes before the start of machining is approached diagonally.

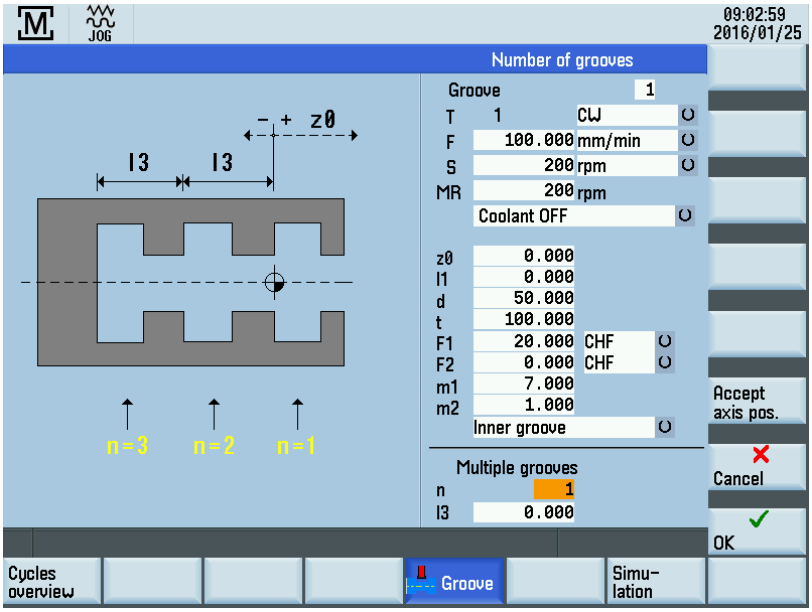
4.4.5.2 Groove cycle - multiple

Functionality

Note

The multiple groove function supplements the single groove option. This function can be used only if all the parameters for the single groove 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:



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.

NOTICE

Collision between the tool and the chuck due to insufficient clearance from the spindle

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.

- Reserve sufficient clearance from the spindle.
- Check the plausibility of the input values before pressing the following key.

Multiple grooves

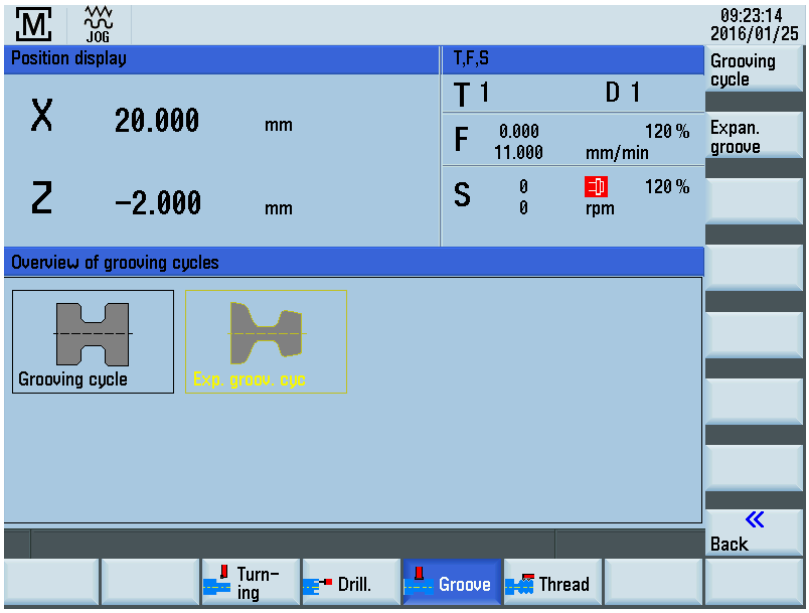
The machining sequence is as follows:

1. Starting from the current axis position, the first groove is produced as described for the single groove function.
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 for the single groove function).
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.

4.4.5.3 Extended grooving

Operating sequences

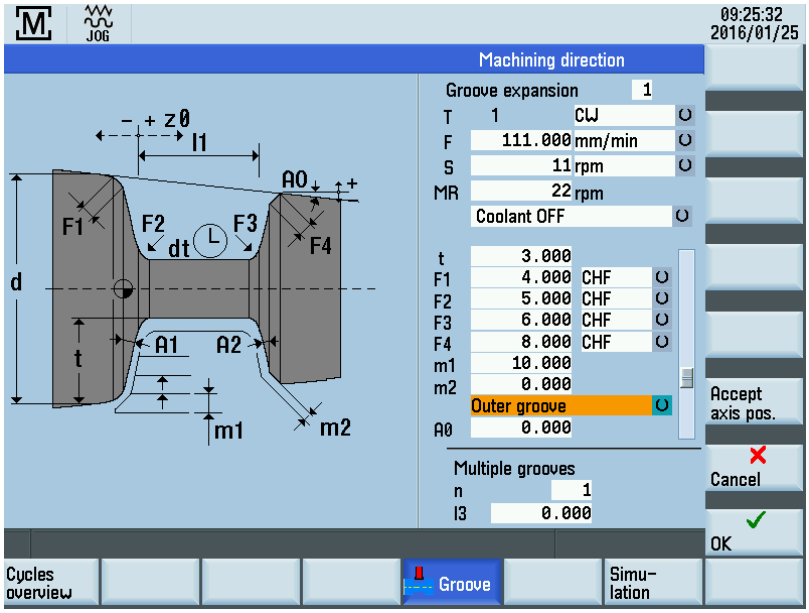
1. You can access the grooving cycle overview by pressing this softkey in the basic screen for MM+.



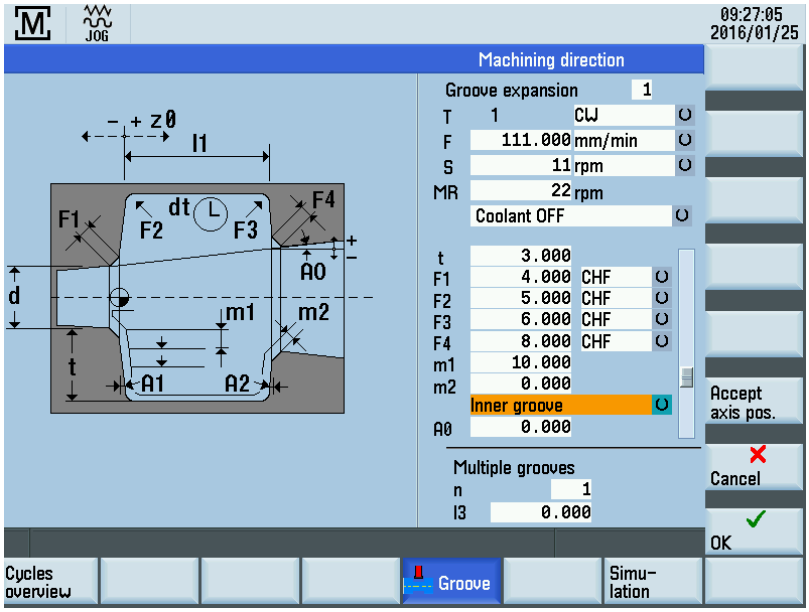
2. You can access the extended grooving function by pressing this softkey in the grooving cycle overview.

Alternatively, you can select this function with cursor keys and activate it with this hardkey.

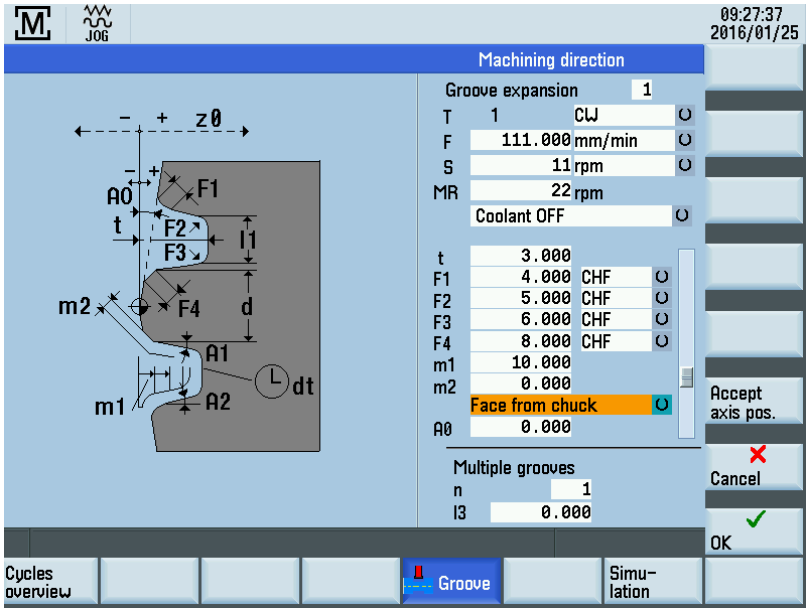
Outer groove:



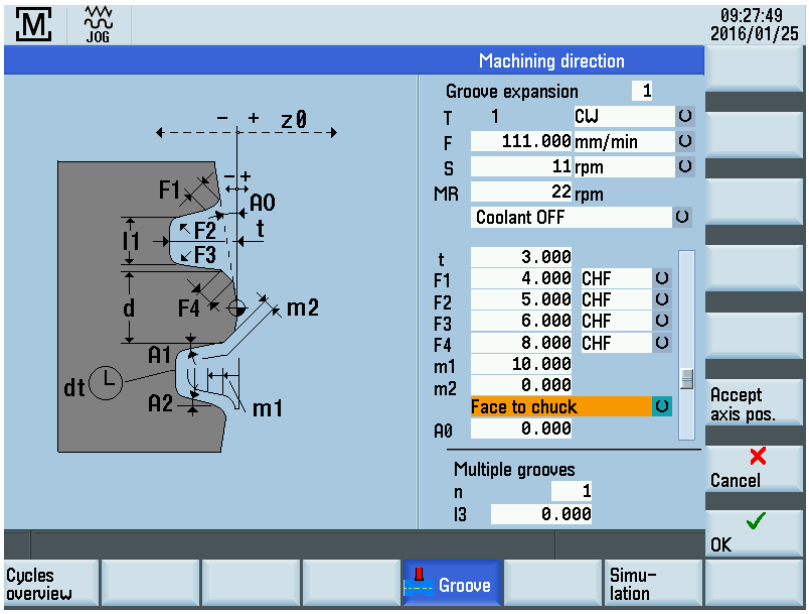
Inner groove:



Face from chuck:



Face to chuck:



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 indicates 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.
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.

Parameter		Description
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 input fields of machining technology data before the start of the cycle.
7. Finally, the original position of the axes before the start of machining is approached diagonally.

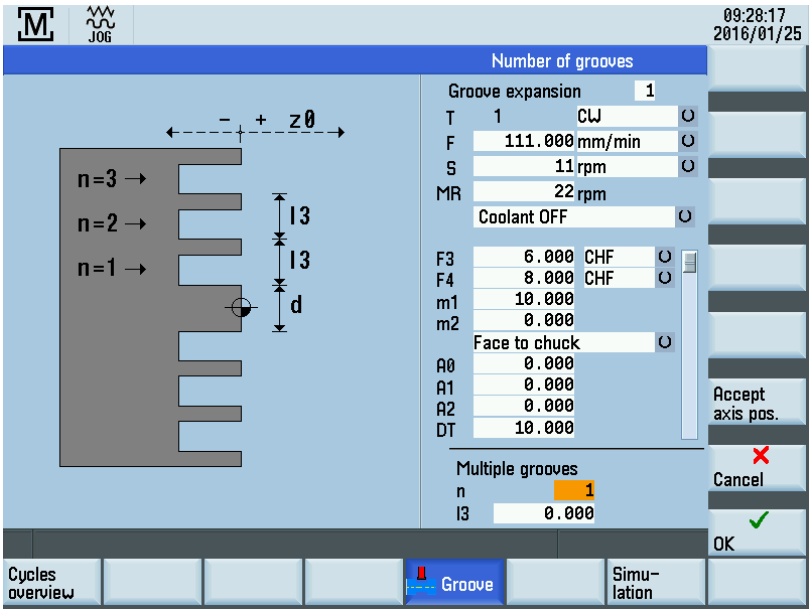
4.4.5.4 Multiple extended grooving

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:



Parameter

Parameter		Description
Distance	l3	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 l3" 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 for the extended grooving function.
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 for the extended grooving function).
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.

4.4.6 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

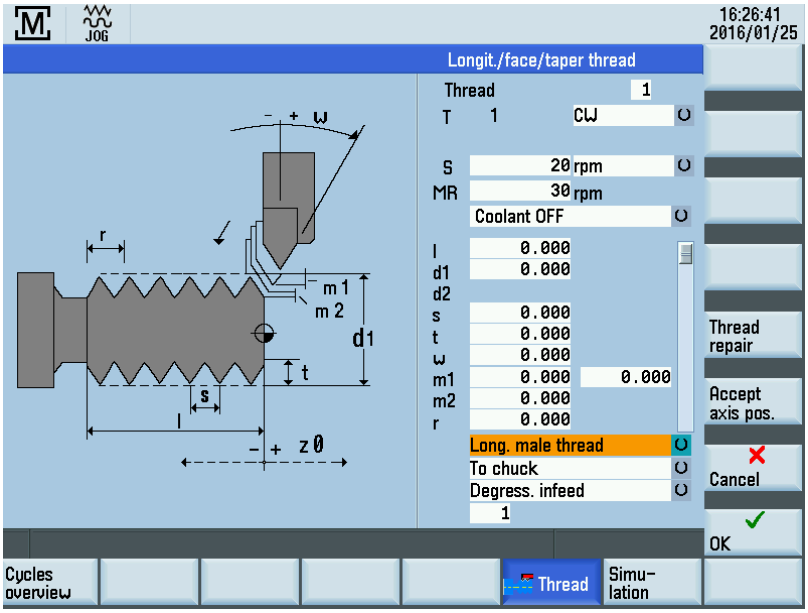
Any limit stops that are activated should be disabled before starting thread cutting or set to a value outside the traversing range needed for thread cutting.

4.4.6.1 Thread cutting

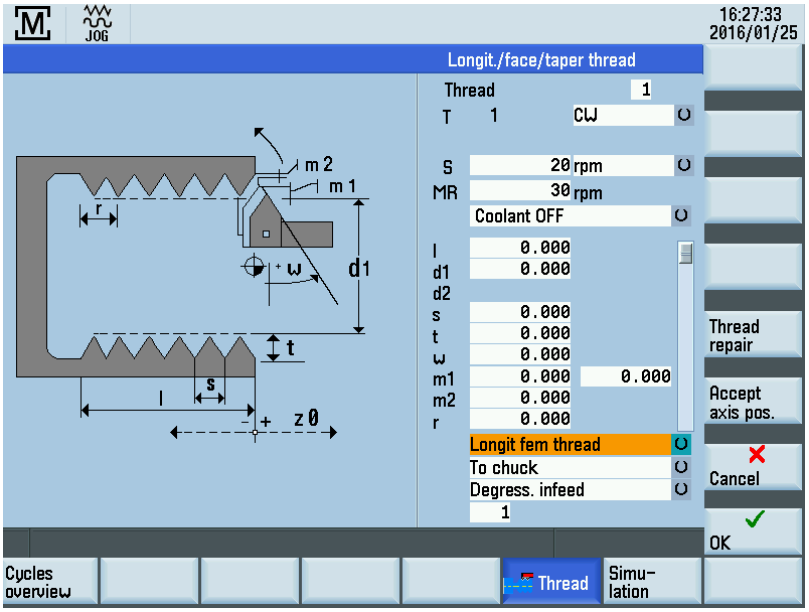
Operating sequences

1. You can access the manual thread cutting function by pressing this softkey in the main screen for MM+.

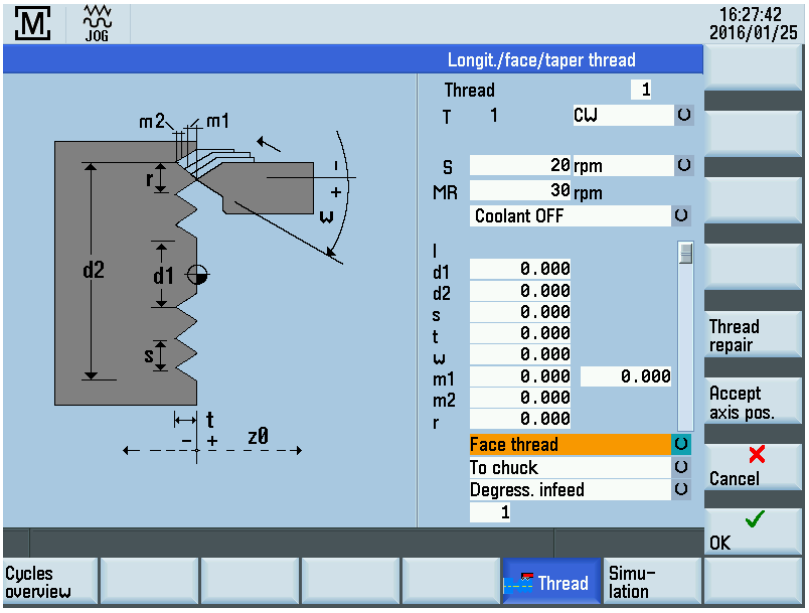
Longitudinal male thread:



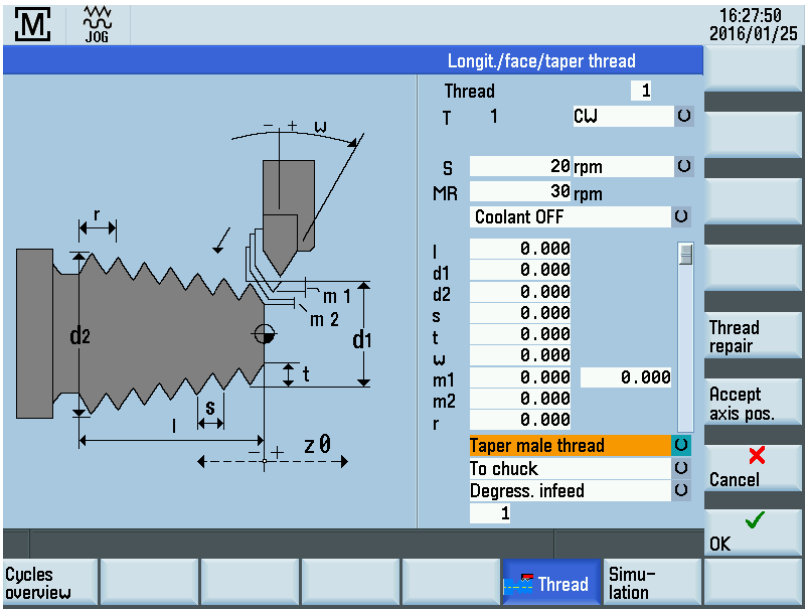
Longitudinal female thread:



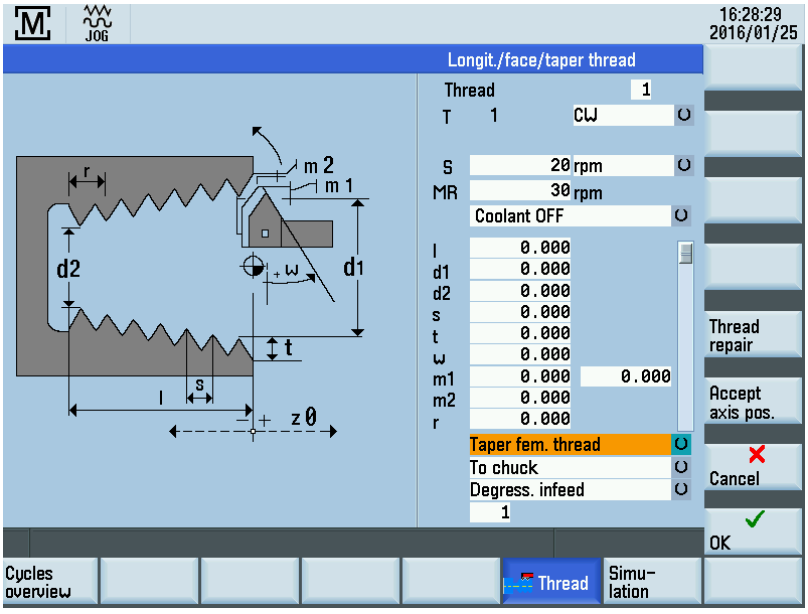
Face thread:



Taper male thread:



Taper female thread:



Parameters

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 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.

Parameter		Description
Depth	t	This parameter is used to set the thread depth. **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.
Finishing allowance	m2	Finishing allowance
Thread run-out path	r	Thread run-out path
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.
Number of thread starts		The number of thread starts is defined here.

Softkey

The above 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 two options:
   – Machining is now complete and the execution screen can be closed with the following softkey:

   – If you wish to continue machining the thread, for example, if thread finishing is required, press the following hardkey again:

4.4.6.2 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.

To make thread recutting to proceed correctly, you have to enter the appropriate values in the thread cutting screen form.

Operating sequences

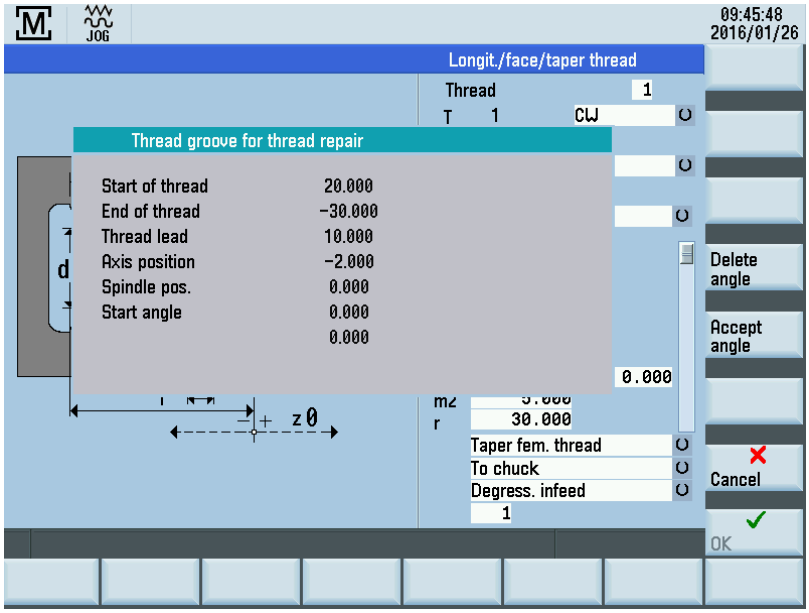
Note

The thread recutting function uses the entry values from the thread cutting screen form. This screen form must therefore have been completed so that thread cutting can proceed correctly.

Thread repair

1. You can call the thread recutting function with this softkey.

   The following screen form appears:



   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:

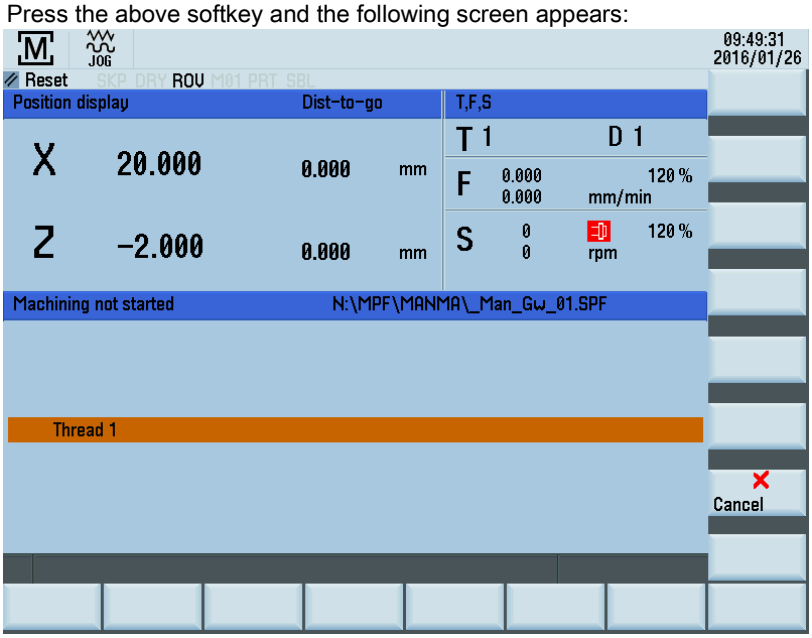
- Appropriate values must already have been entered in the thread cutting screen form at this point.
- The screen above is displayed.
- The spindle must be stationary (switched off) and must already have been synchronized, in other words, it must have been turned through at least one full revolution 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 value shown on the screen must be between the values for the thread initial position and thread end position.

Accept angle

- Press the above softkey. 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 above softkey and the following screen appears:



The rest of the thread cutting process is exactly the same as that described for the manual thread cutting function.

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.

4.4.7 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.

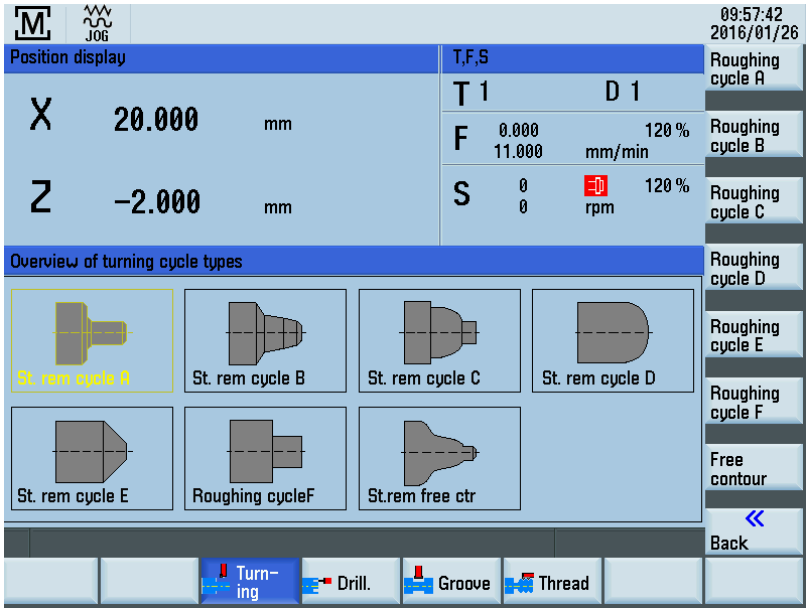
The contour can be machined using the following position of the contour:

- Outside right
- Inside right
- Outside left

Roughing is either longitudinal or planar.

Operating sequences

1. Starting from the main menu for MM+ you can reach the roughing cycle functions via this softkey.



The following roughing cycles can be used via the vertical softkey bar or by choosing 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

4.4.7.1 Roughing cycle A

Functionality

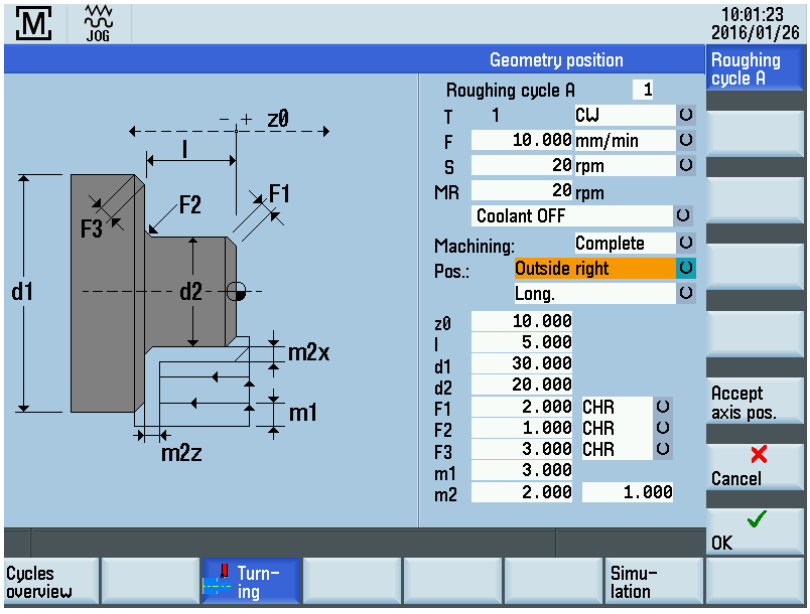
The roughing cycle A function is used to produce a simple stepped contour (step), with the option of working the transitions to adjacent faces as a radius or chamfer.

Operating sequences

1. You can access the turning cycle overview by pressing this softkey in the basic screen for MM+.

2. In the displayed screen, press this softkey.

Alternatively, you can select this function from the turning cycle overview with cursor keys and activate it with this hardkey.



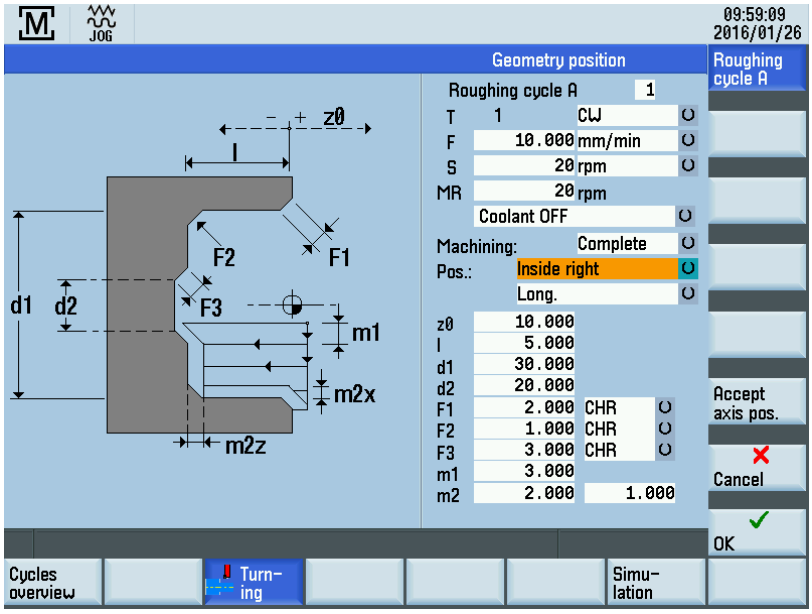
Input fields

The input fields in the roughing cycle A screen form have the following meanings:

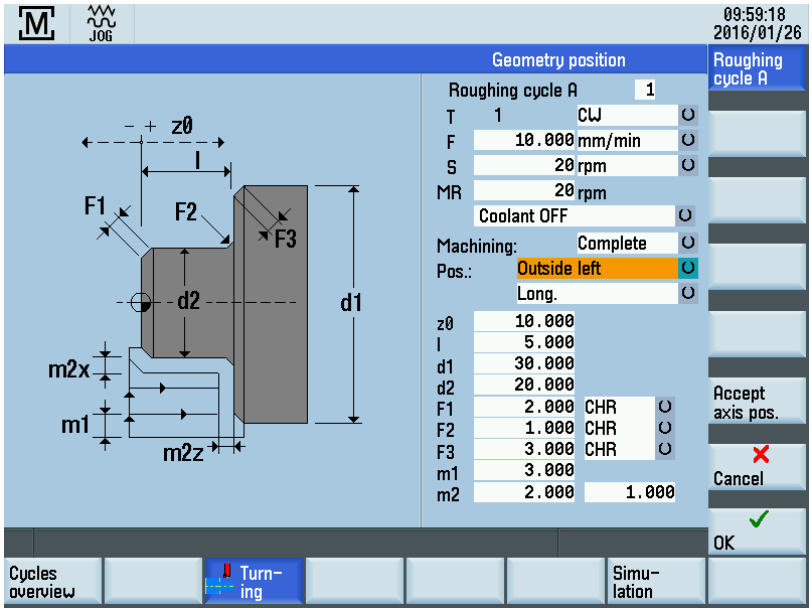
Parameter		Description
Length	l	Enter the length of the "step" to be produced, taking the contour start position ("Reference z0") in the axial axis (Z axis) as the starting point.
Diameter	d1	Outside diameter of the "step" to be machined in the radial axis (absolute position of the X axis in the diameter).
Diameter	d2	Inside diameter of the "step" to be machined in the radial axis (absolute position of the X axis in the diameter).
Chamfer/radius	F1	Depending on the option selected, this value forms either a transition radius (display "RND") or a transition chamfer (display "CHR" or "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 "CHR" or "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 "CHR" or "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:

Inside right:



Outside left:



4.4.7.2 Roughing cycle B

Functionality

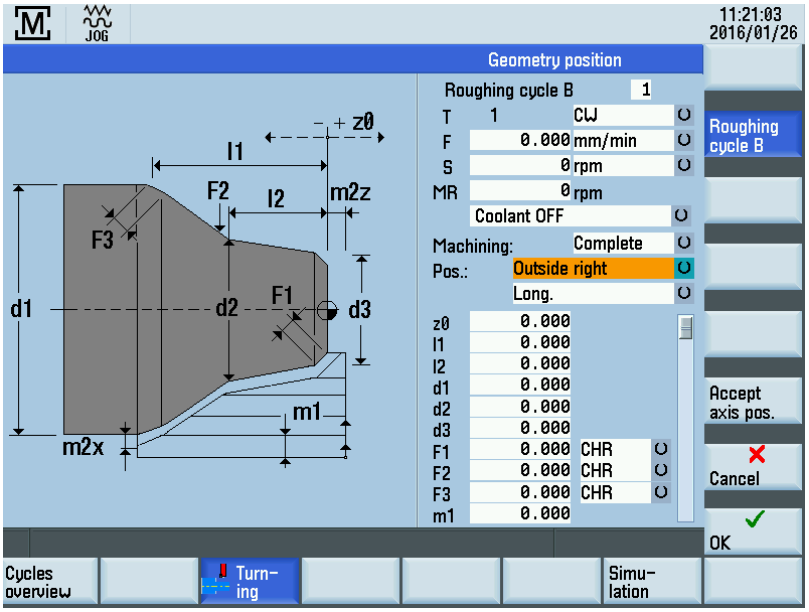
The roughing cycle B function is used to produce a simple cutting contour, with an additional interpolation point allowing beveled or tapered contours. Transitions to adjacent faces can again be worked as a radius or chamfer.

Operating sequences

1. You can access the turning cycle overview by pressing this softkey in the basic screen for MM+.

2. In the displayed screen, press this softkey.

Alternatively, you can select this function from the turning cycle overview with cursor keys and activate it with this hardkey.



Input fields

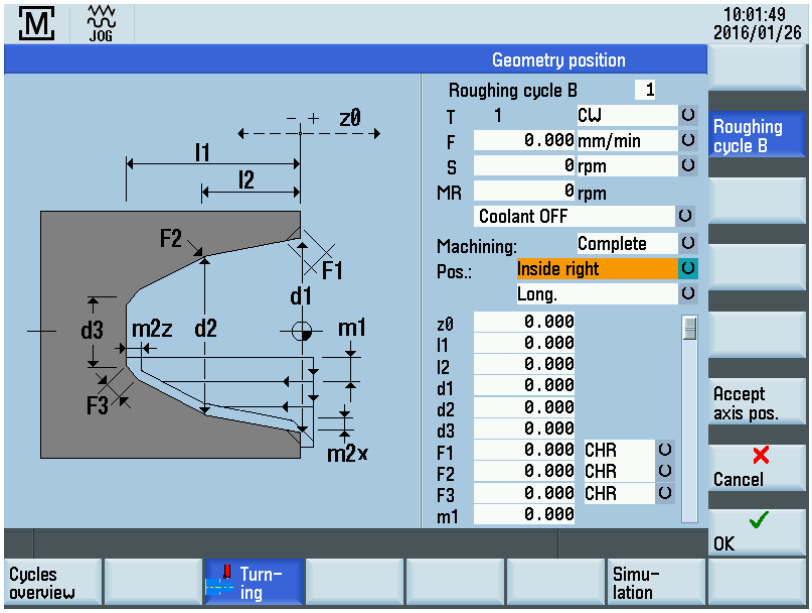
The input fields in the roughing cycle B screen form have the following meanings:

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).

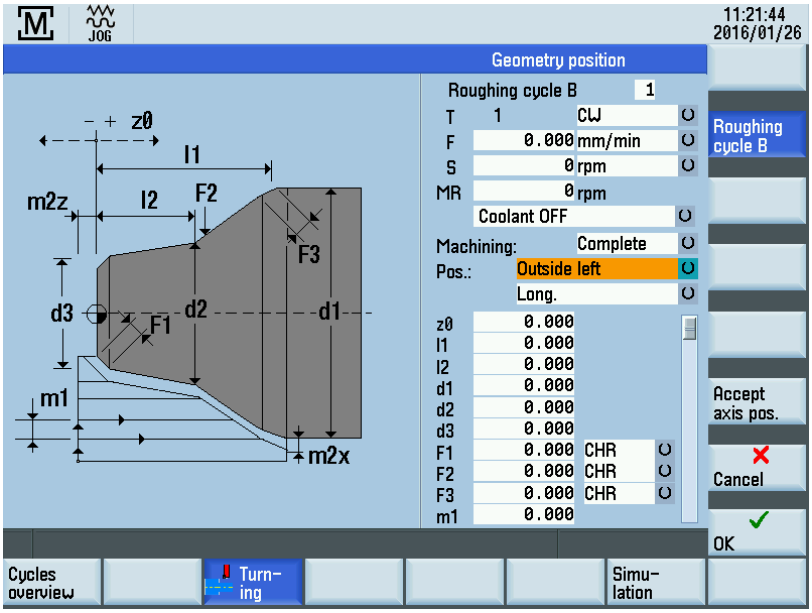
Parameter		Description
Chamfer/radius	F1	Depending on the option selected, this value forms either a transition radius (display "RND") or a transition chamfer (display "CHR" or "CHF") of less than 45^0 between the end face and the inside diameter of the "step". You can toggle between RND/CHR/CHF using 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 "CHR" or "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 "CHR" or "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:

Inside right:



Outside left:



4.4.7.3 Roughing cycle C

Functionality

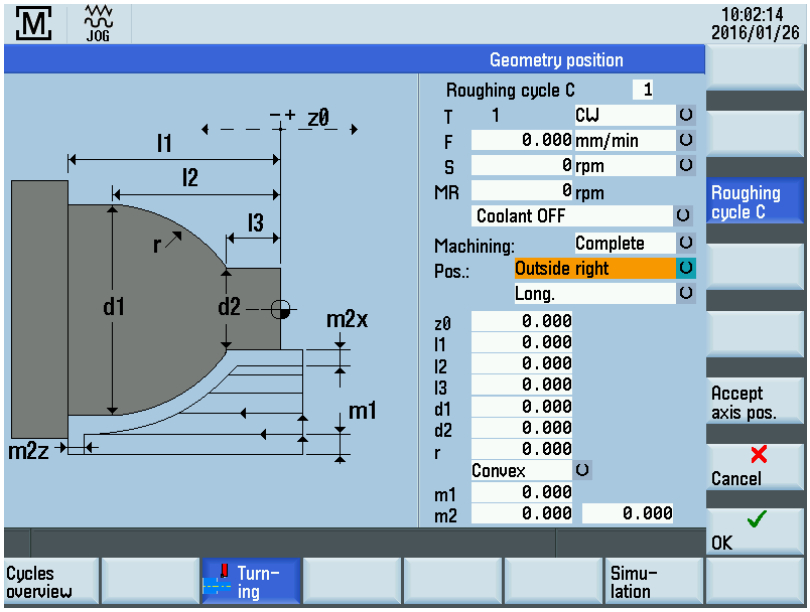
The roughing C function is used to produce a special cutting contour, with a filleted transition between the inside and outside diameter of the contour. Other chamfers or radii cannot be included.

Operating sequences

1. You can access the turning cycle overview by pressing this softkey in the basic screen for MM+.

2. In the displayed screen, press this softkey.

Alternatively, you can select this function from the turning cycle overview with cursor keys and activate it with this hardkey.



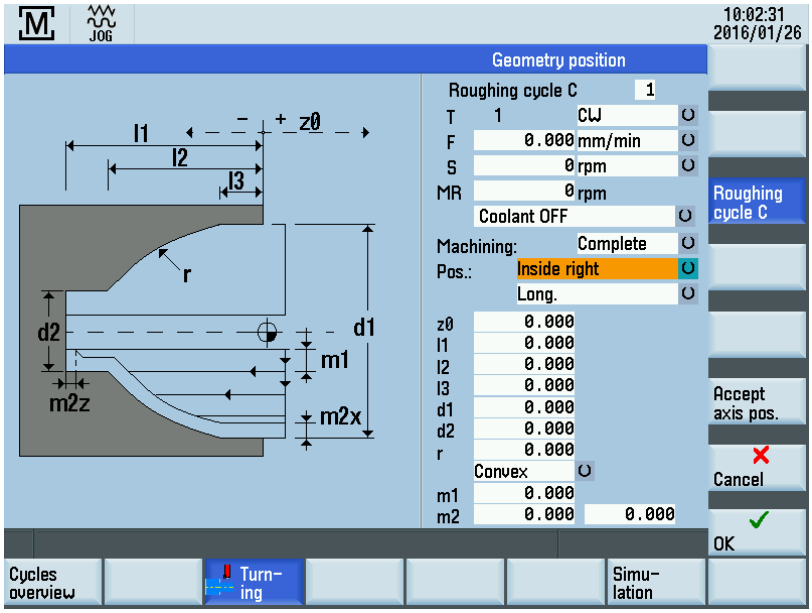
Input fields

The input fields in the roughing cycle C screen form have the following meanings:

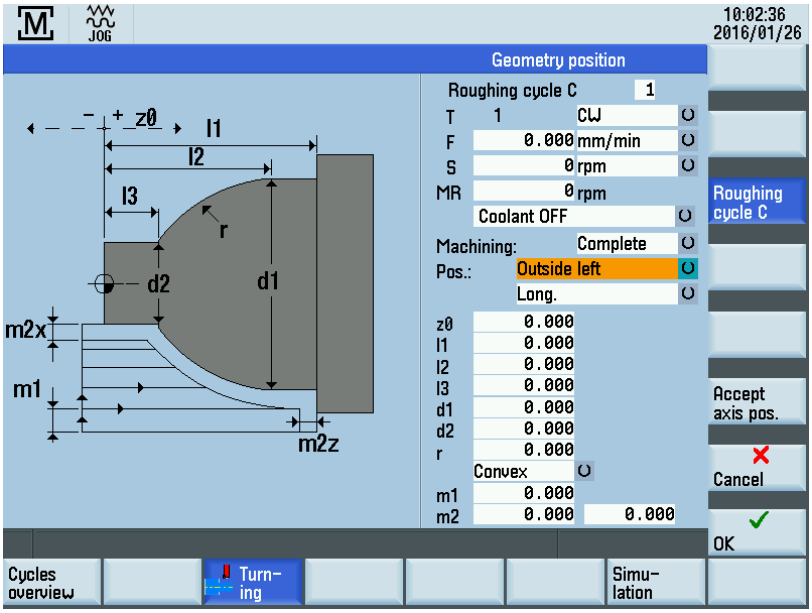
Parameter		Description
Length	l1	Enter the end point of the contour in the axial axis here, taking the contour start position ("Reference z0") in the axial axis (Z axis) as the starting point.
Length	l2	End point of filleting in the longitudinal axis (Z axis).
Length	l3	Start point of filleting in the 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 a cycle 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:

Inside right:



Outside left:



4.4.7.4 Roughing cycle D

Functionality

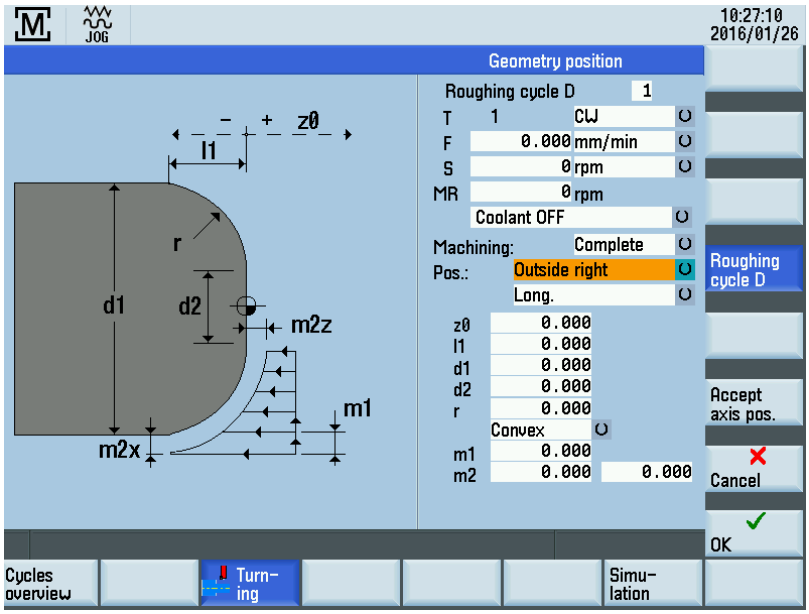
The roughing cycle D function allows a single radius contour to be machined, supported by cycles.

Operating sequences

1. You can access the turning cycle overview by pressing this softkey in the basic screen for MM+.

2. In the displayed screen, press this softkey.

Alternatively, you can select this function from the turning cycle overview with cursor keys and activate it with this hardkey.



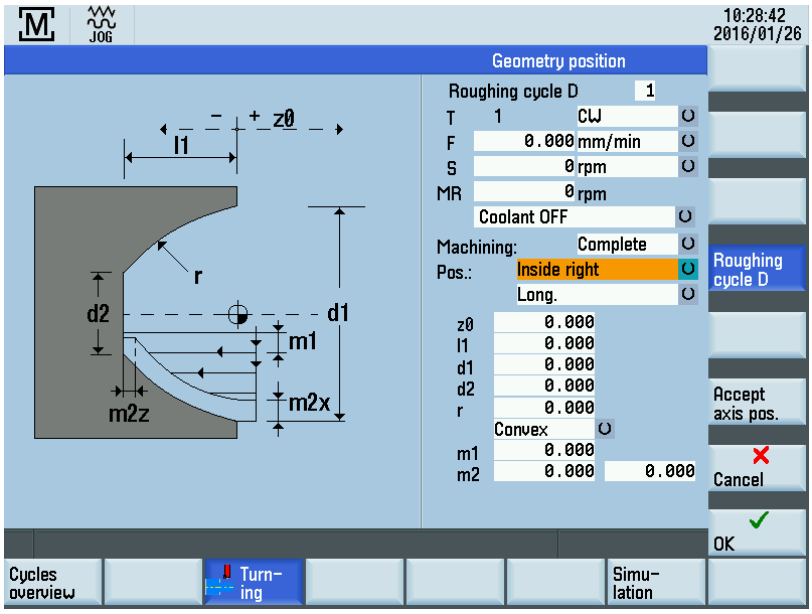
Input fields

The input fields in the roughing cycle D screen form have the following meanings:

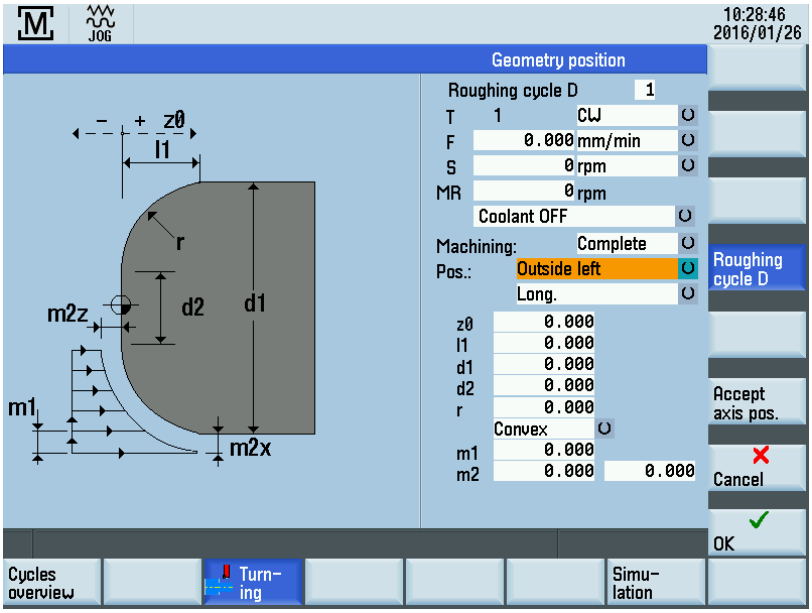
Parameter		Description
Length	l1	Enter the end point of the contour in the axial axis here, taking the contour start position ("Reference z0") in the axial axis (Z axis) as the starting point.
Diameter	d1	Outside diameter of the contour to be machined in the transverse axis (absolute position of the 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 a cycle 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:

Inside right:



Outside left:



4.4.7.5 Roughing cycle E

Functionality

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

Operating sequences

Turn-ing

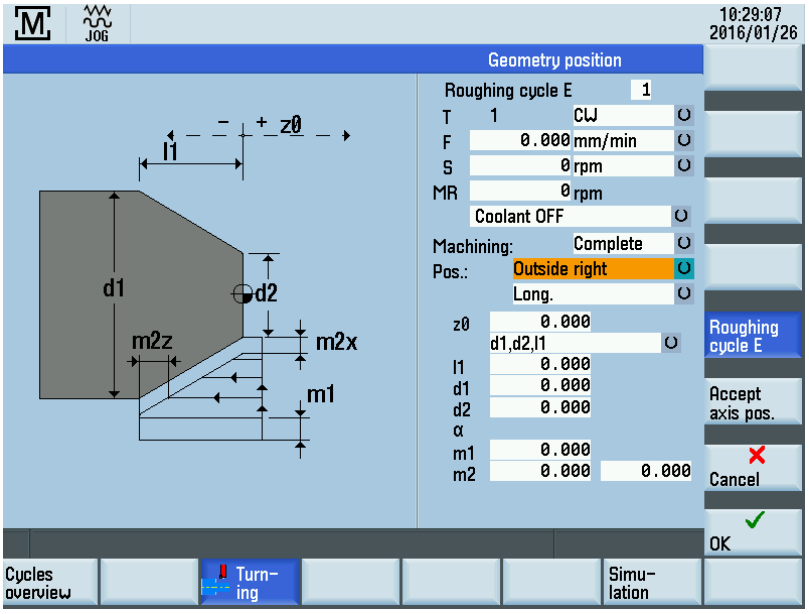
1. You can access the turning cycle overview by pressing this softkey in the basic screen for MM+.

Roughing cycle E

2. In the displayed screen, press this softkey.

INPUT

Alternatively, you can select this function from the turning cycle overview with cursor keys and activate it with this hardkey.



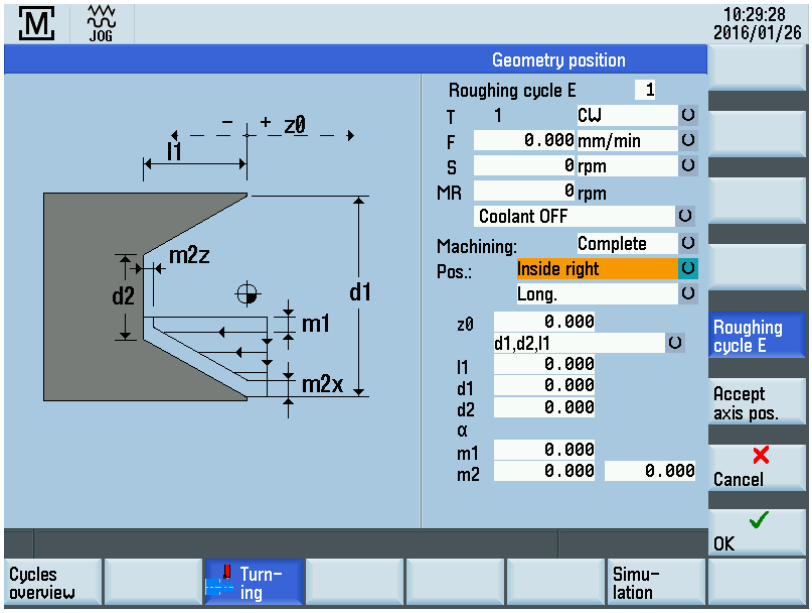
Input fields

The input fields in the roughing cycle E screen form have the following meanings:

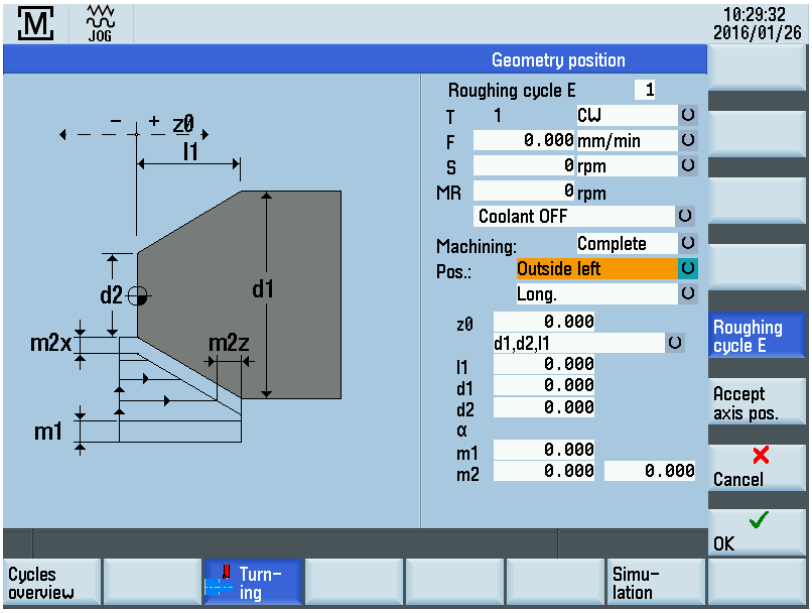
Parameter		Description
d1, d2,...		The dimensioning type can be selected in this toggle field. The following options are available: "d1,d2,l1" -> "d1,l1,angle" -> "d2,l1,angle" -> "d1,d2,angle(d1)" -> "d1,d2,angle(d2)" The selection is indicated in the diagram displayed on the screen.
Length	l1	Enter 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).
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:

Inside right:



Outside left:



4.4.7.6 Roughing cycle F

Functionality

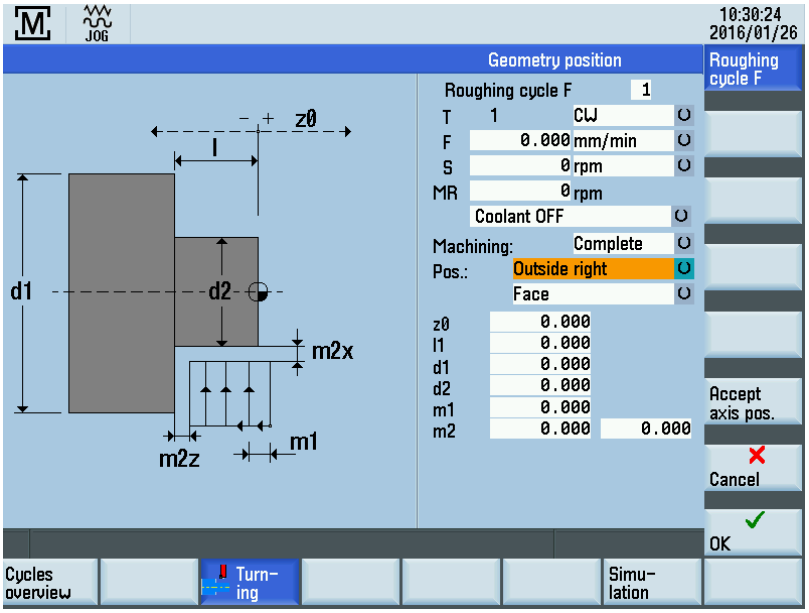
The roughing cycle F function allows cycle-supported production of an end face (cutting direction is planar) or of a peripheral surface (cutting direction is longitudinal).

Operating sequences

1. You can access the turning cycle overview by pressing this softkey in the basic screen for MM+.

2. In the displayed screen, press this softkey.

Alternatively, you can select this function from the turning cycle overview with cursor keys and activate it with this hardkey.



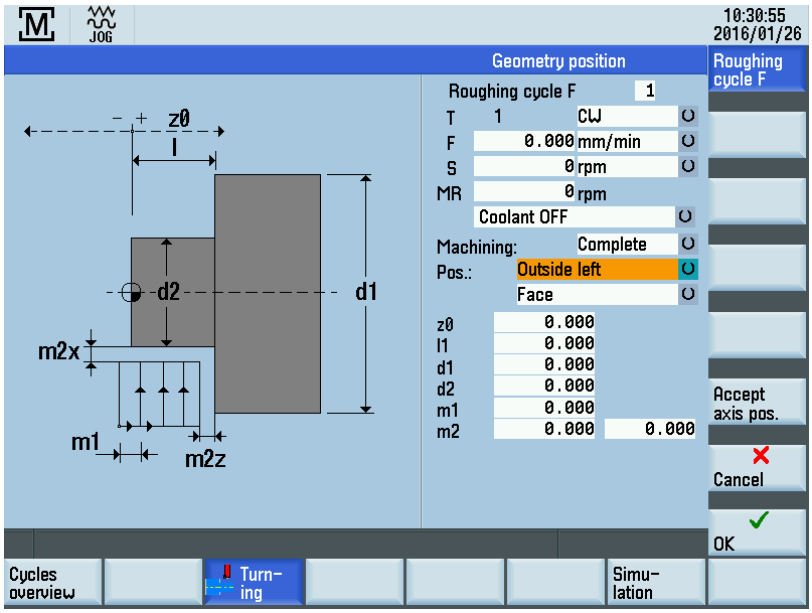
Input fields

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:

Outside left:



4.4.7.7 Roughing cycle, free contour

Functionality

The free contour cycle function is used for the input and for the processing of an arbitrary contour path.

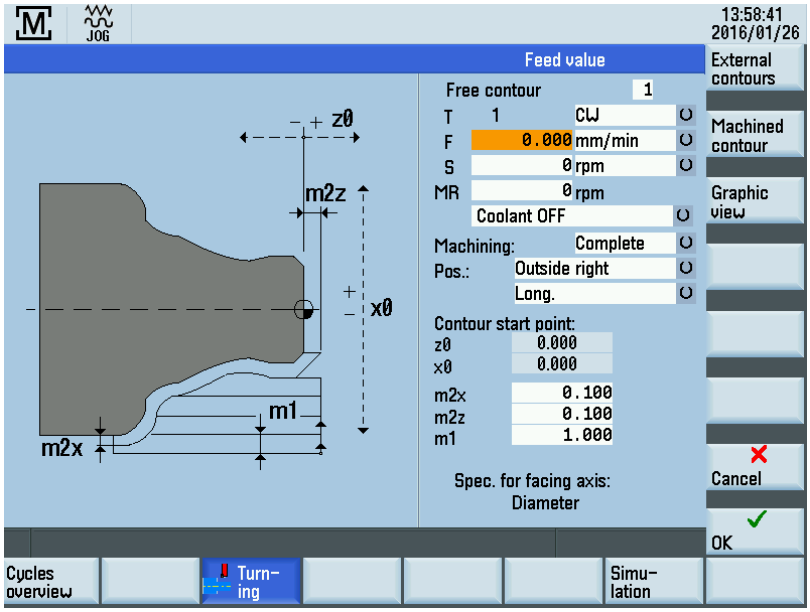
Operating sequences

1. You can access the turning cycle overview by pressing this softkey in the basic screen for MM+.

2. In the displayed screen, press this softkey.

INPUT

Alternatively, you can select this function from the turning cycle overview with cursor keys and activate it with this hardkey.

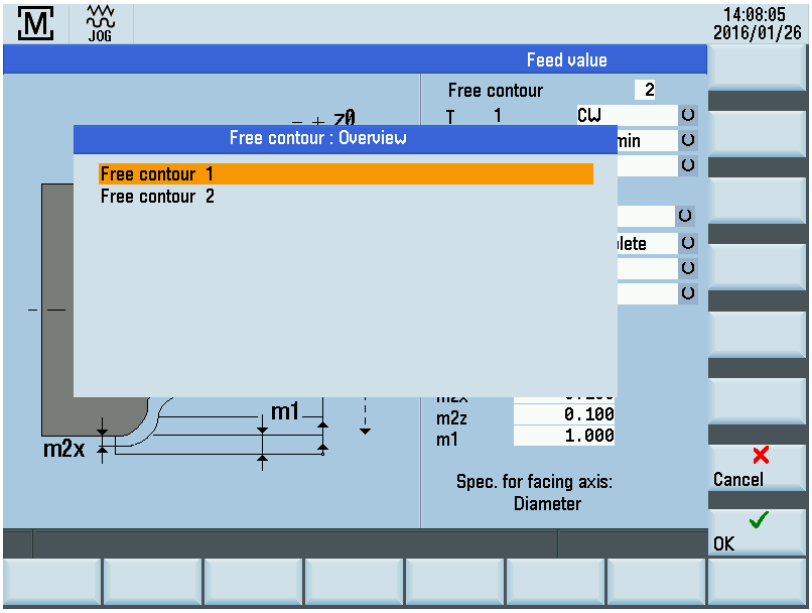


Parameter	Description
x0	Outside diameter of the shaft to be machined in the transverse axis (absolute position of the X axis in the diameter).
m2x	Finishing allowance horizontal to the contour.
m2z	Finishing allowance perpendicular to the contour.
m1	Enter the maximum infeed depth for roughing. The 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

Cycles overview

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

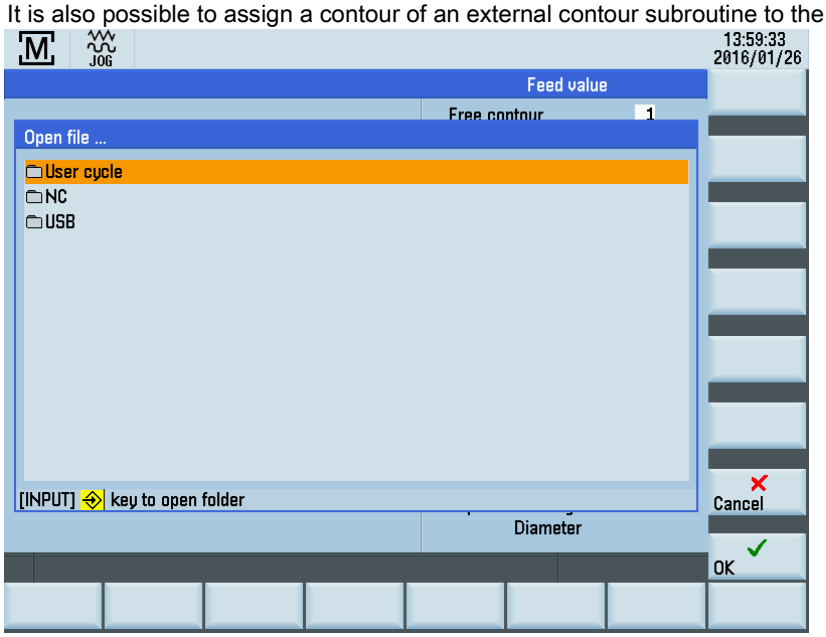


To choose, place the cursor on the appropriate line and press the following softkey:

OK

External contours

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



This function opens a dialog box with which the contour subroutine can be selected. Pressing the following softkey links the selected program with the cycle:

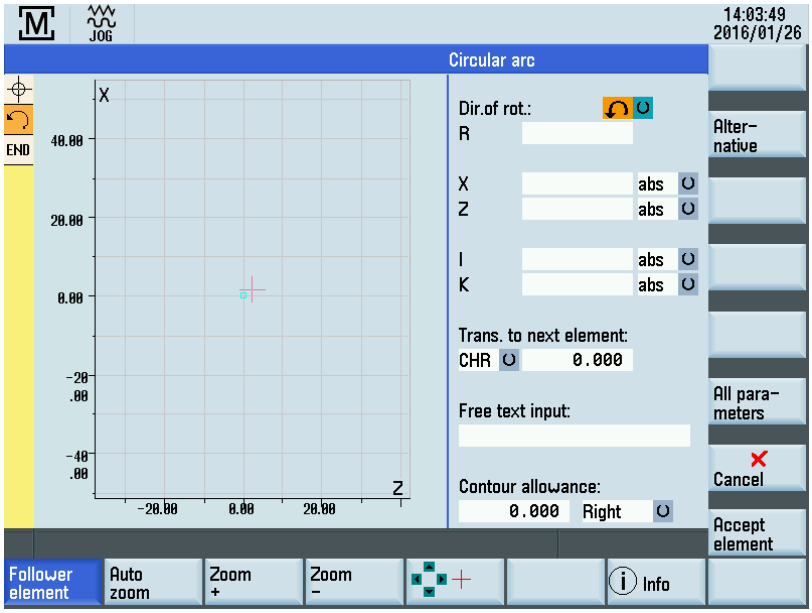
OK

Machined contour

The machined contour function branches to the contour input.

Note

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



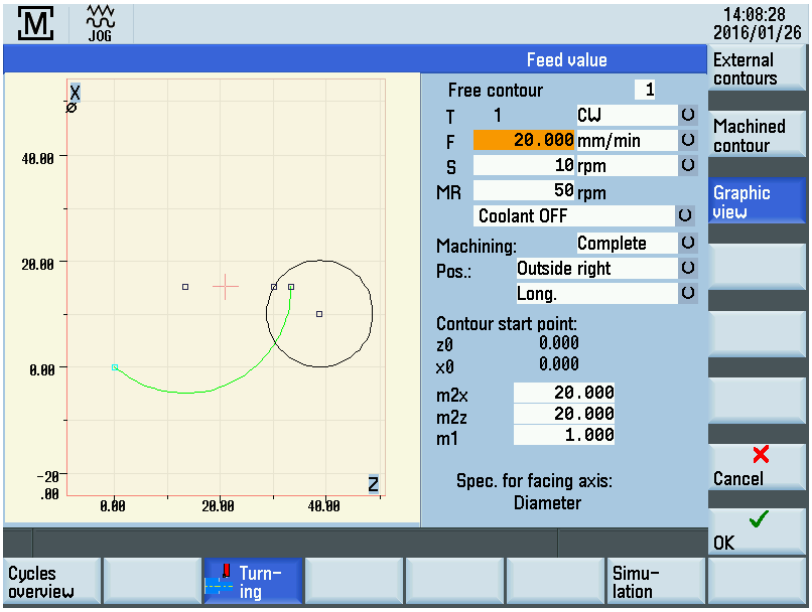
First, define the contour starting point.

References

For more information on the machined contour function, see Section "Defining a start point" of the SINUMERIK 808D ADVANCED Programming and Operating Manual (Turning).

Graphic view

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



4.4.7.8 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 the planar or longitudinal cutting direction 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° 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°) to the contour start point.
3. Execution of the finishing motion along the parameterized contour.
4. Retraction by the clearance distance below 45° 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.

5 Machining the machining step program manually

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.

WARNING

Danger to life and/or damage to machine due to insecure part programs

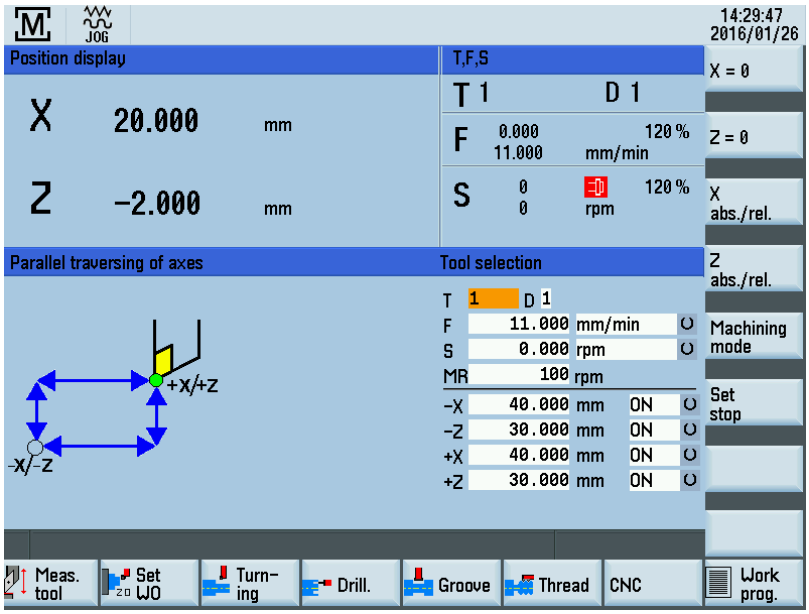
Running an insecure part program on your machine may cause unexpected attacks to the machine, which in turn can lead to death, personal injuries, and/or machine damage.

- Make sure you use part programs that come from trusted sources.

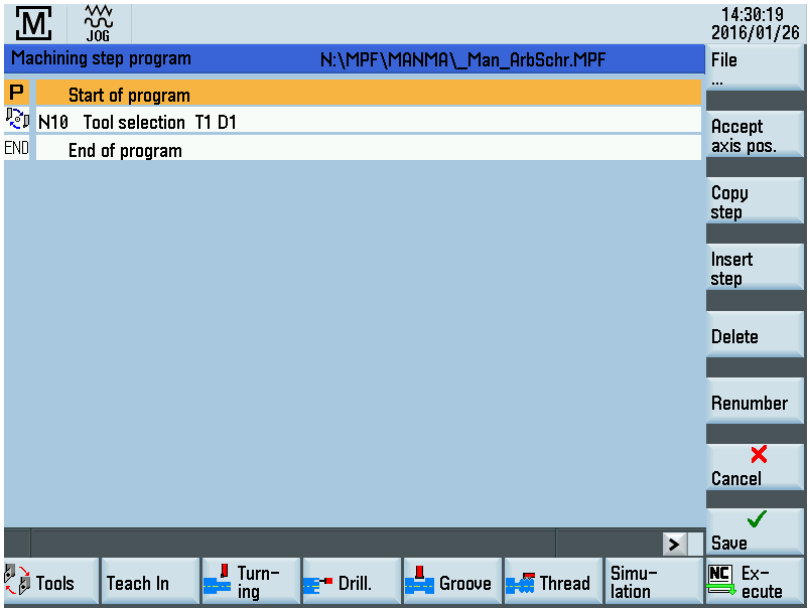
Operating sequences

Manual

1. Press this softkey to access the basic screen for MM+.



2. You can access the screen for input into the list by pressing this softkey in the main screen for MM+.



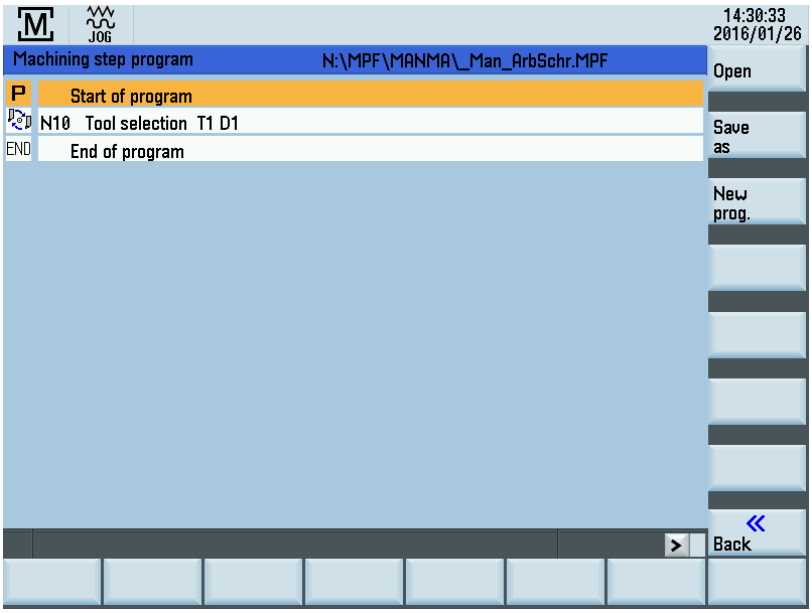
Screen handling functions

- To select machine steps, press the <cursor up> or <cursor down> key on the PPU. The selected step is displayed on an orange background.
- If you have selected a machining cycle, the input screen for this cycle or the taught block opens automatically when you press the <cursor right> key.

Softkeys

Other inputs in the screen are made with softkeys:

Opens the following dialog box:



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 (control system storage), ensure that the external medium is not removed during the machining.

Save as

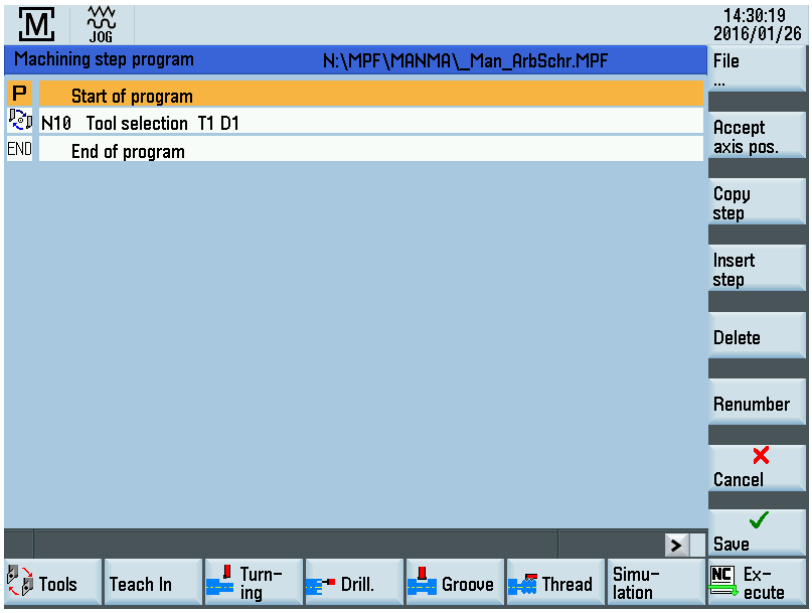
Displays a save dialog.

New prog.

Displays a dialog used to create a program.

Back

Returns to the menu for the machining step program



Accept axis pos.

Inserts a positioning block at the current machine axis position in the selected machining step.

Delete

Deletes the currently selected machining step.

Cancel

Interrupts the machining step program function.

This softkey returns you to the main screen for MM+.

If you have changed any values, a prompt window informs you accordingly.

OK: Press it to save the values.

Cancel: Press it to discard the settings.

Saves the machining step program.

This softkey returns you to the main screen for MM+.

Inserts a tool change into the program.

This softkey shows you the dialog box for Tool change in the machining step program (Page 68).

Inserts a traversing block in the program.

Inserts a roughing cycle in the program.

This softkey shows you the dialog box for Roughing cycles (Page 47).

Inserts a drilling cycle in the program.

This softkey shows you the dialog box for the Manual thread tapping (Page 32).

Inserts a groove/cutting cycle into the program.

This softkey shows you the dialog box for the Manual grooving/parting (Page 34).

Inserts a thread cycle.

This softkey shows you the dialog box for the Manual thread cutting (Page 42).

This softkey shows you the dialog box for machining Simulate machining (Page 71).

Saves the selected step program.

This softkey opens the screen form in which the Executing the machining step program (Page 73) step program is actually executed.

Changes the softkey function as follows.

From 

to 

5.1 Tool change in the machining step program

Functionality

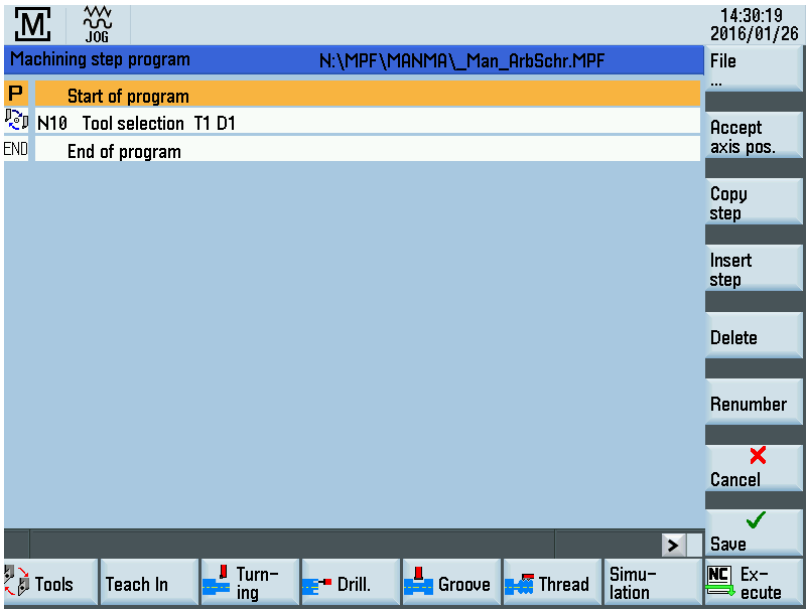
You add a tool change step to the machining step program.

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.

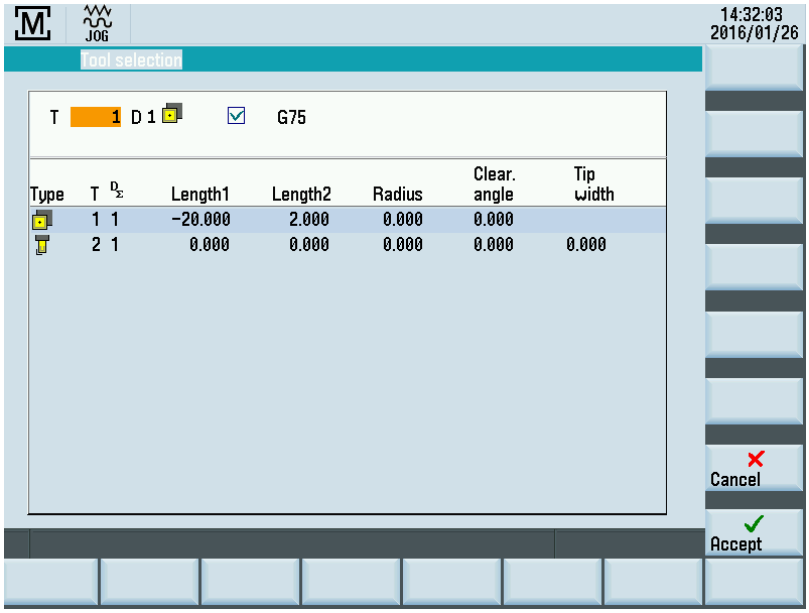
Operating sequences

1. You have opened a machining step program.

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



2. Press this softkey and a list with all tools created in the control system will be displayed.



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.

   Alternatively, perform the following operations to select the tool:

   Use this key to change to the list and position the cursor on the appropriate tool.

   Then press this key to confirm the selection.

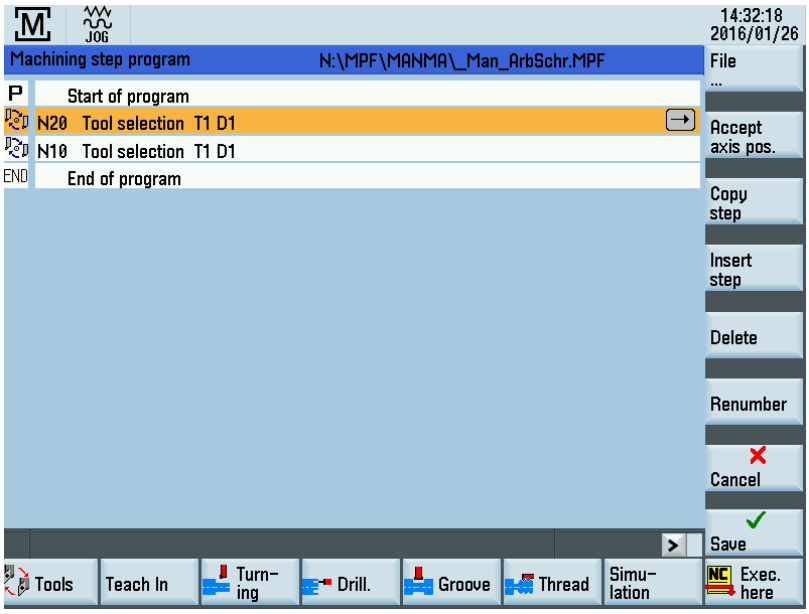
   The selected tool is copied into the "T" input field.

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 this softkey.

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



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.

5.2 Teach In

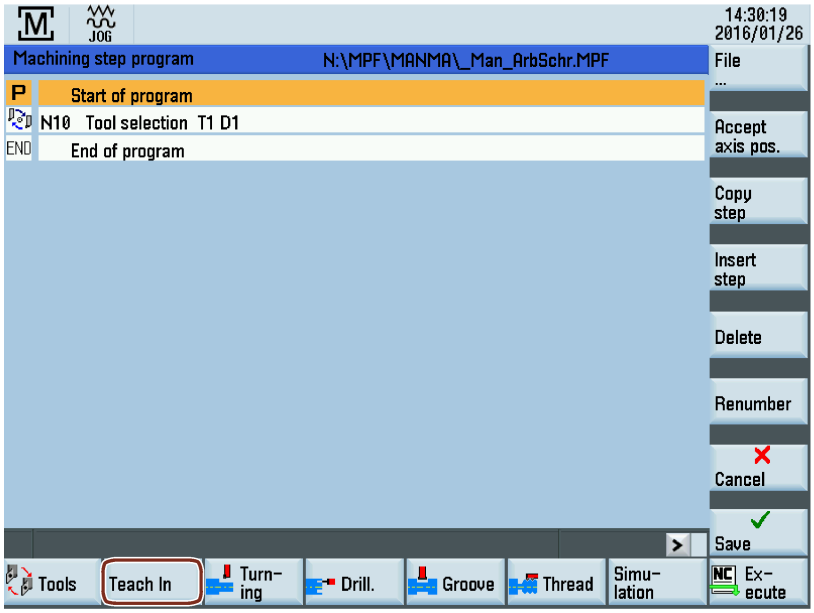
Functionality

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

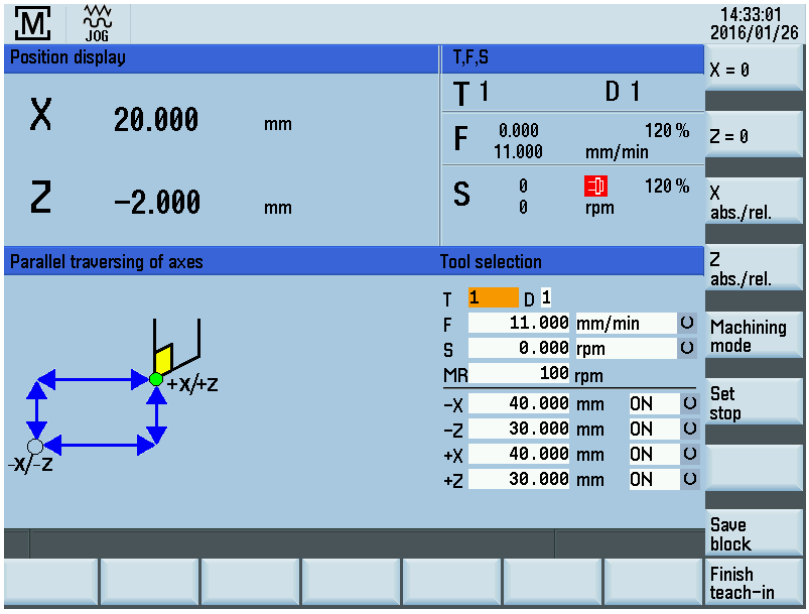
Operating sequences

Teach In

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

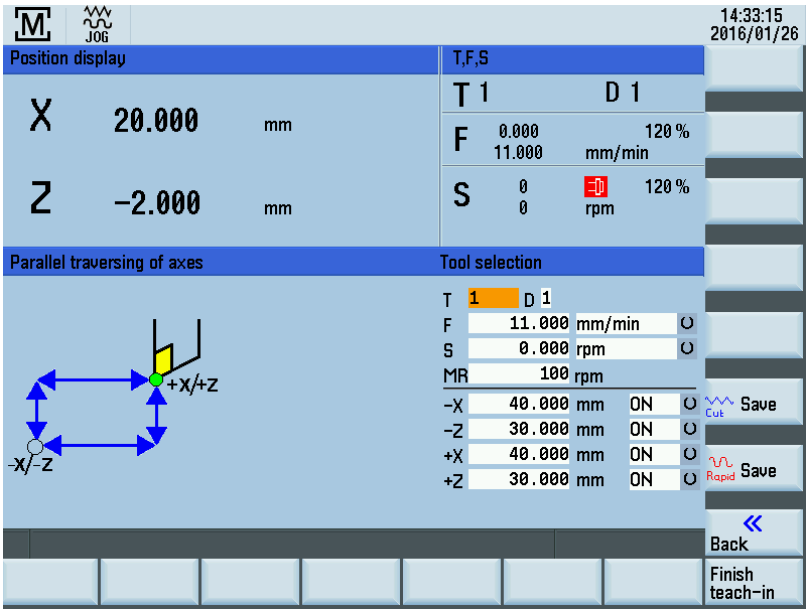


The controller switches to the manual machining screen forms of axis-parallel turning, taper turning and radius turning.



Save block

2. Traverse to a position that is to be taught in and press this softkey.



Save

3. You can save the position with path feed with this softkey.

Save

4. You can save the position with rapid traverse via this softkey.

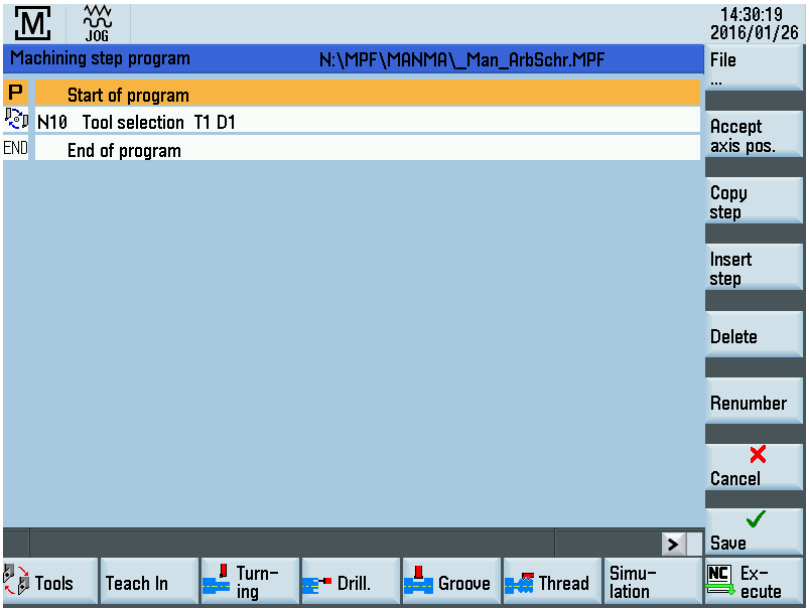
Save block

After the control acknowledged the action with a screen message (for example, saying that the block was inserted as N20), a new position could be traversed to and this is in turn taught in using this softkey.

Finish teach-in

5. Exit "Teach In" mode via this softkey.

The menu returns to the machining step program.



5.3 Simulate machining

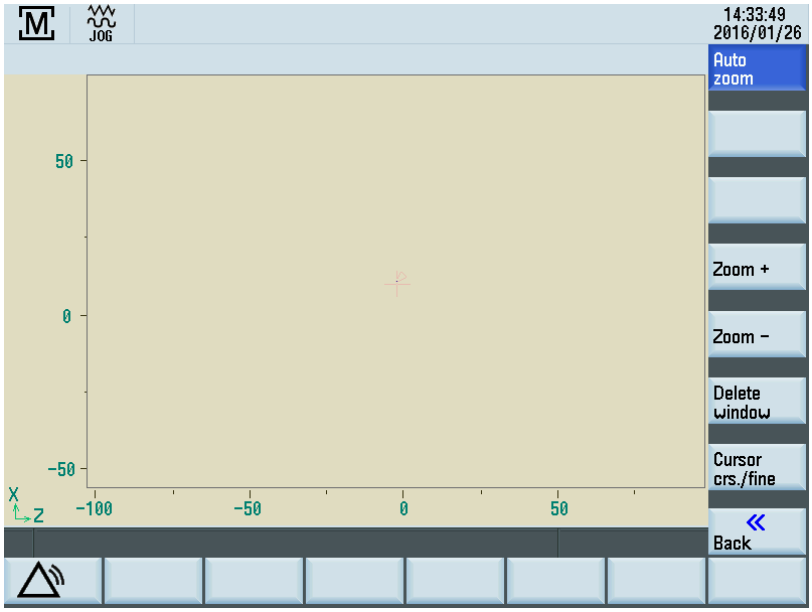
Function

You can use this function to graphically display the execution of a single cycle on the screen.

Operating sequences

Simu-
lation

The start screen is opened.



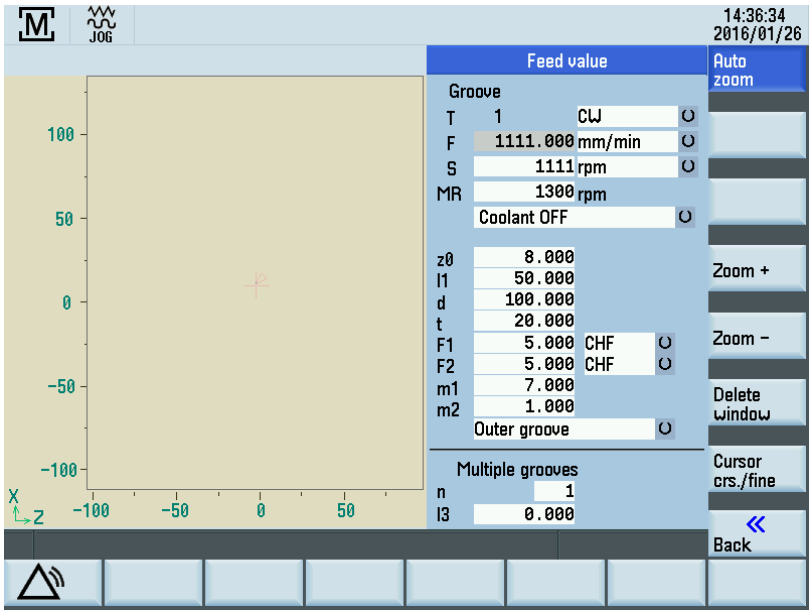
CYCLE START

Press the above hardkey to start the simulation for the selected part program.

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.

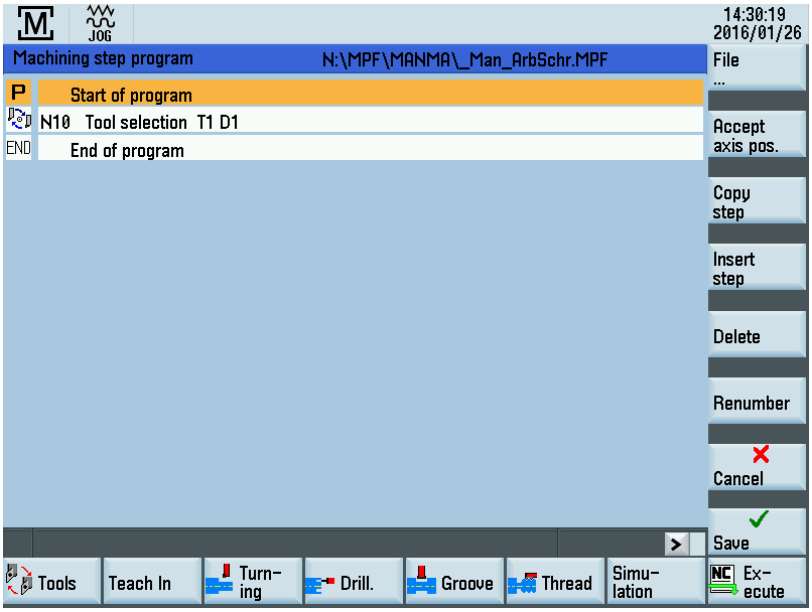


References

For more information on further operating options for a simulation, see Chapter "Program simulation" of the SINUMERIK 808D ADVANCED Programming and Operating Manual (Turning).

5.4 Executing the machining step program

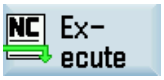
Functionality



In the "Machining step program" function, you can toggle between the following horizontal softkey functions using the above hardkey.

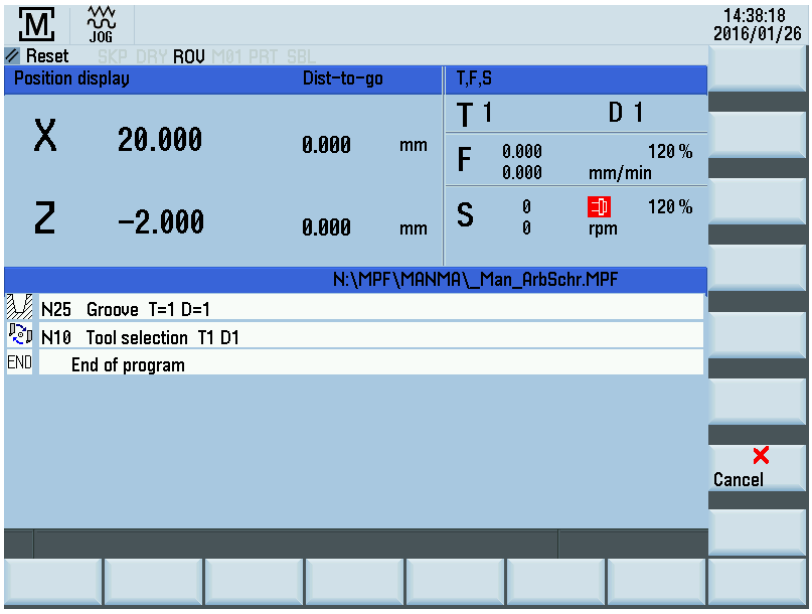
and


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



With the above softkey, the complete machining step program is executed.

With the above softkey, the program is executed from the actual cursor position in the machining step program (step that is marked in the program).



Operating sequences, executing the machining step program

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

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

1. Start machining using this hardkey.
   The machining operation on the workpiece is executed.
   The individual machining steps are executed in the order in which you program them.
   Press this hardkey if you want to interrupt the machining operation.

2. If machining was terminated (for example, completed), the execution screen can be exited using this 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.

6 Messages

Functionality

The meanings of the messages listed below differ from those given in the SINUMERIK 808D ADVANCED Diagnostics Manual:

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.

700049 Reference point of X axis not reached
700050 Reference point of Z axis not reached
700051 Spindle direction illegal
700052 Monitoring time for JOG operation exceeded
700053 Spindle override not 100%
700054 Spindle not started
700055 Feed override = 0%
700056 Spindle dir. change not allowed in thread mach.

Siemens AG
Division Digital Factory
Postfach 48 48
90026 NÜRNBERG
GERMANY

Programming and Operating Manual (Manual Machine Plus (MM+), Turning)
6FC5398-3DP10-0BA5, 07/2018