Operation/Programming 10/2004 Edition

sinumerik

SINUMERIK 840D/840Di/810D
ShopMill

SIEMENS

SIEMENS

SINUMERIK 840D/840Di/810D

ShopMill

Operation/Programming

Valid for

Control	*Software version*
SINUMERIK 840D powerline	7
SINUMERIK 840DE powerline	7
SINUMERIK 840Di	3
SINUMERIK 840DiE (Export Version)	3
SINUMERIK 810D powerline	7
SINUMERIK 810DE powerline	7

10.04 Edition

SINUMERIK® Documentation

Printing history

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

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

Status code in the "Remarks" column:

A .... New documentation.
B .... Unrevised reprint with new Order No.
C .... Revised edition with new status.

Edition	Order No.	Remarks
10.97	6FC5298-2AD10-0BP0	A
11.98	6FC5298-2AD10-0BP1	C
03.99	6FC5298-5AD10-0BP0	C
08.00	6FC5298-5AD10-0BP1	C
12.01	6FC5298-6AD10-0BP0	C
11.02	6FC5298-6AD10-0BP1	C
11.03	6FC5298-6AD10-0BP2	C
10.04	6FC5298-6AD10-0BP3	C

Trademarks
SIMATIC®, SIMATIC HMI®, SIMATIC NET®, SIROTEC®, SINUMERIK®, and SIMODRIVE® are registered trademarks of Siemens AG. Other names in this publication might be trademarks whose use by a third party for his own purposes may violate the rights of the registered holder.

More information is available on the Internet at:
http://www.siemens.com/motioncontrol

This publication was produced with with WinWord 2000
and Designer V 7.1.

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

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

Subject to change without prior notice

6FC5298-6AD10-0BP3
Printed in Germany

Siemens Aktiengesellschaft

Preface

Structure of the Documentation	The SINUMERIK documentation is organized in 3 parts: • General Documentation • User Documentation • Manufacturer/Service Documentation
Audience	This documentation is intended for use by operators of vertical machining centers or universal milling machines controlled by the SINUMERIK 840D/840Di/810D system.
Validity	This Operation/Programming Guide is valid for ShopMill SW 6.4.
Hotline	Please address any queries to the following hotline: A&D Technical Support Tel.: +49 (0) 180 5050-222 Fax: +49 (0) 180 5050-223 Email: adsupport@siemens.com If you have any queries (suggestions, corrections) concerning the documentation, please send them to the following fax number or email address: Fax: +49 (0) 9131 98-2176 Fax form at the end of the documentation Email: motioncontrol.docu@erlf.siemens.de
Internet address	http://www.cnc-werkstatt.de http://www.siemens.com/motioncontrol
SINUMERIK 840D powerline	Since 09.2001, improved-performance variants SINUMERIK 840D powerline and SINUMERIK 840DE powerline are available. For a list of available powerline modules, please refer to the following Hardware Description: **Reference**: /PHD/, SINUMERIK 840D Configuration Manual
SINUMERIK 810D powerline	Since 12.2001, improved-performance variants SINUMERIK 810D powerline and SINUMERIK 810DE powerline are available. For a list of available powerline modules, please refer to the following Hardware Description: **Reference**: /PHC/, SINUMERIK 810D Configuration Manual
Standard scope	This Operator's/Programming Guide describes the functionality of the ShopMill operator interface. Extensions or changes made by the machine tool manufacturer are documented by the machine tool manufacturer. More detailed information about other publications relating to SINUMERIK 840D/840Di/810D and publications that apply to all SINUMERIK controls (e.g. Universal Interface, Measuring Cycles...) can be obtained from your local Siemens branch office.

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

Principle

Your SIEMENS 840D/840Di/810D with ShopMill has been designed and constructed according to state-of-the-art technology and approved safety regulations and standards.

Additional equipment

SIEMENS offers special add-on equipment, products and system configurations for the focused expansion of SIEMENS controls in your field of application.

Personnel

Only **suitably trained, authorized, reliable personnel** should be allowed to handle the equipment. Persons who are not qualified should never be allowed to work on the control, even for a short time.

The relevant **responsibilities** of personnel who set up, operate and maintain the equipment must be clearly **defined** and adherence to these responsibilities **monitored**.

Procedure

Before the control is started up, it should be ensured that the Operator's Guides have been read and understood by the people responsible. The operator also has a **permanent obligation to continuously monitor** the overall technical condition (externally recognizable defects and damage and changes in the operating behavior) of the control.

Servicing

Repairs must be carried out by personnel who are **specially trained and qualified** in the relevant technical subject according to the information supplied in the service and maintenance guide. All appropriate safety specifications must be observed.

The following is deemed to be **improper usage** and **exempts the manufacturer from any liability**:

- **Any** application deviating from the above points or usage extending beyond the given limits.
- Cases where the control is **not maintained in perfect technical condition**, or is operated without due regard to safety or danger, and cases where any or all of the instructions in the Operator's Guide have not been observed.
- If faults that might affect the safety of the equipment are not rectified **before** the control is started up.

- Any **modification**, **bypassing** or **disabling** of items of equipment on the control that are required to ensure fault-free operation, unlimited use and active and passive safety.

Structure of the documentation

This documentation uses the following information blocks, identified by pictograms:

Function

Background information

Operating sequence

Explanation of parameters

Additional notes

Software option
The function described is a software option. This means that the function will only run on the control if you have purchased the relevant option.

Warnings

The following 5 warnings with varying degrees of severity are used in this documentation.

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

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

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

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

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

Machine manufacturer

If changes or additions exist for a particular topic, they are referenced here:

Please observe the details provided by the machine manufacturer.

References

Further references for particular topics are indicated here:

Reference:

A complete list of available literature is included in the Appendix of this Operator's Guide.

Terms

The meanings of several fundamental terms used in this documentation are defined below:

Program
A program is a sequence of instructions for the CNC control, which produce a particular workpiece at the machine.

Contour
A contour outlines a workpiece.
The term "contour" is also used to denote the section of a program that uses individual elements to define the outline of a workpiece.

Cycle
A cycle, for example, mill rectangular pocket, is a subroutine specified by ShopMill to execute a repetitive machining process.
(a cycle is sometimes also called a "function".)

"Unit of measurement"

The parameter units are always specified in metric units in this documentation. The corresponding inch measures are given in the table below.

Metric	Inch
mm	in
mm/tooth	in/tooth
mm/min	in/min
mm/rev	in/rev
m/min	ft/min

Contents

Introduction

1.1 ShopMill

ShopMill is operating and programming software for milling machines that makes it easy for you to operate the machine and to program workpieces.

These are some of the features the software provides:

Setting up the machine

Special measurement cycles make it easier to measure the tools and the workpiece.

Creating a program

3 different programming methods are available:

- G code programs for mold-making applications imported from CAD/CAM systems.
- G code programs that you create directly at the machine.
  You can use all technology cycles for programming:
- Sequential control programs that you create directly at the machine (software option).
  The workpiece is programmed with ease because graphical techniques are used and no knowledge of G codes is required.
  ShopMill displays the program as a clearly understandable process plan and presents the individual cycles and contour elements in a dynamic graphical display.

Irrespective of the programming method you use, the following functions will simplify programming and processing:

- A powerful contour calculator lets you enter any contours.
- A stock removal cycle complete with detection of residual material saves unnecessary machining (software option).
- A swivel cycle allows multiple-surface machining and machining on inclined surfaces, irrespective of the machine kinematics of the machine.

Executing a program

You can display the execution of programs on the screen three-dimensionally.
This makes it easy for you to check the result of programming and to observe the progress of workpiece machining at the machine (software option).

The execution of sequential control programs is a software option.

Executing a program

You can display the execution of programs on the screen three-dimensionally.
This makes it easy for you to check the result of programming and to observe the progress of workpiece machining at the machine (software option).

The execution of sequential control programs is a software option.

Tool management

ShopMill stores your tool data. The software can also manage the data for tools that are not in the tool magazine.

Program management

Programs can be created simply by copying and modifying similar programs; there is no need to start again from the beginning.

With ShopMill you can implement multiple clamping of identical or different (software option) workpieces with optimization of the tool sequence.

You can access external programs from a network or from a diskette drive (software option).

1.1.1 Sequence of operations

Two typical working situations are considered separately in this Guide.

- You want to execute a program for the purpose of automatically machining a workpiece.
- You want to create the program to be used for machining a workpiece.

Executing a program

Before you execute a program, you have to set up your machine. You must perform the following steps with the support of ShopMill (see Sec. "Operation"):

- approach the reference point of the machine
  (only for incremental position measuring systems)
- gauge the tools
- define the workpiece zero
- enter any other work offsets

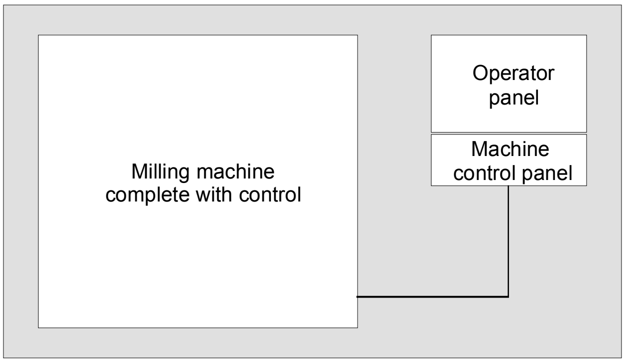
When you have finished setting up the machine, you can select a program and execute it automatically (see Sec. "Automatic operation").

Creating a program

As you create a new program, you can choose whether it will be a sequential control program or a G code program (see "Creating a ShopMill program" or "G code program").
During creation of a sequential control program, ShopMill prompts you to enter all the relevant parameters. Programming progress is automatically indicated in a dashed-line diagram. Help screens that explain the parameters in each operation also support you with programming.
You can, of course, also insert G code commands in a sequential control program.
A G code program, however, must be created entirely out of G code commands.

1.2 Workstation

A ShopMill workstation comprises the milling machine complete with a CNC/positioning control plus an operator panel and a machine control panel.



Workstation configuration

Milling machine

You can use ShopMill on vertical or universal milling machines with up to 10 axes (including rotary axes and spindles). Of the 10 axes, 3 linear and 2 rotary axes plus 1 spindle can be displayed at any one time.

Machining step and G code programs are suitable for 2D to 2½D machining; for 3D machining, use G code programs from CAD/CAM systems.

Control

ShopMill runs on the SINUMERIK 840D/840Di/810D CNC with PCU 20 and PCU 50.

Operator panel

You communicate with ShopMill via the operator panel.

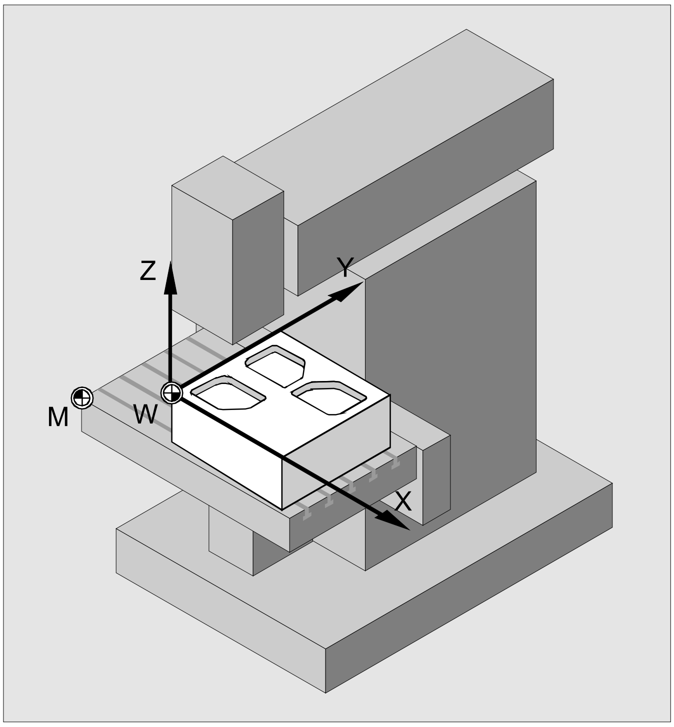
Machine control panel

You operate the milling machine via the machine control panel.

1.2.1 Coordinate system

The basic coordinate system used to machine a workpiece on a milling machine is right-angled. It consists of the three coordinate axes X, Y, and Z that are parallel to the machine axes.

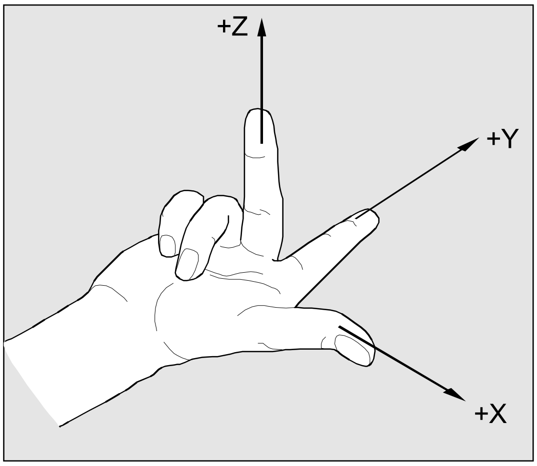
The positions of the coordinate system and the machine zero depend on the type of machine used.



Position of the coordinate system, machine zero and workpiece zero (example)

The axis directions are governed by the "right-hand rule" (according to DIN 66217).
Seen from in front of the machine, the middle finger of the right hand points in the opposite direction to the infeed of the main spindle.
Therefore:

- the thumb points in the +X direction
- the index finger points in the +Y direction
- the middle finger points in the +Z direction



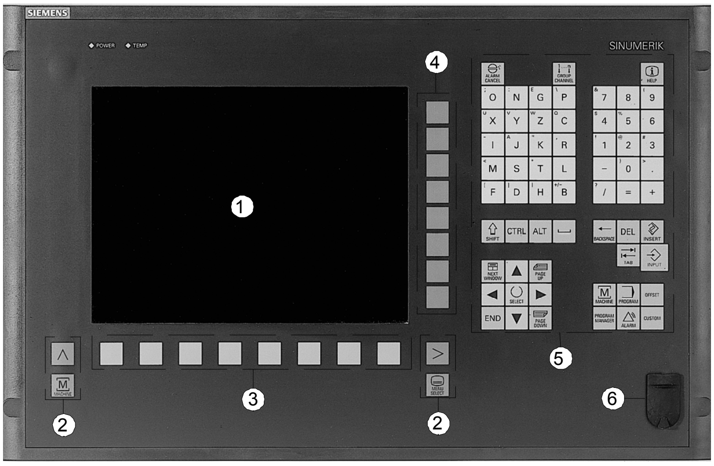
Right-hand rule

1.2.2 Operator panels

You can use one of the following operator panels for the PCU:

OP 010
OP 010C
OP 010S with OP 032S full CNC keyboard
OP 012
OP 015 with 19" full CNC keyboard

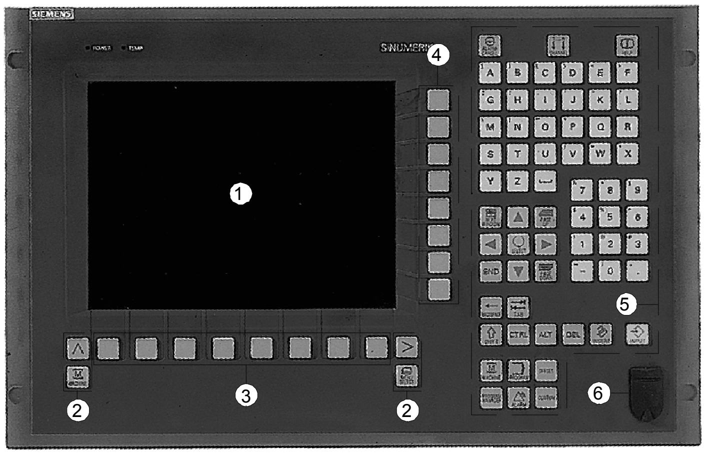
Operator panel OP 010



Operator panel OP 010

1 10" screen
2 Screen keys
3 Horizontal softkey bar
4 Vertical softkey bar
- Alphanumeric keypad
  Correction/cursor pad with control keys and input key
- USB interface

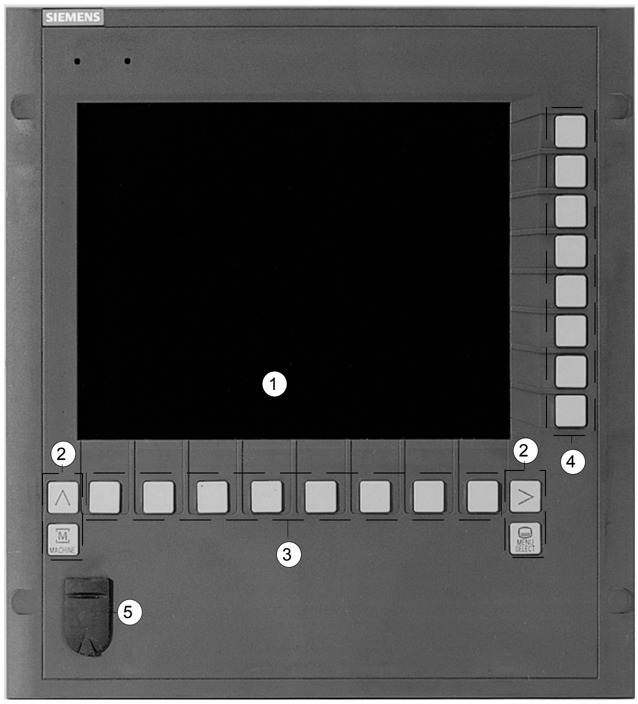
Operator panel OP 010C



Operator panel OP 010C

1 10" screen
2 Screen keys
3 Horizontal softkey bar
4 Vertical softkey bar
5 Alphanumeric keypad
Correction/cursor pad with control keys and input key
6 USB interface

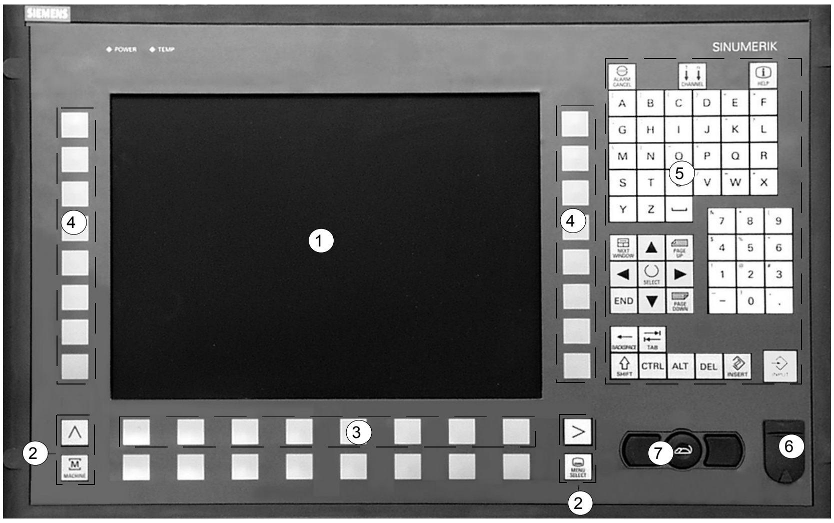
OP 010S slimline operator panel



OP 010S operator panel

1 10" screen
2 Screen keys
3 Horizontal softkey bar
4 Vertical softkey bar
5 USB interface

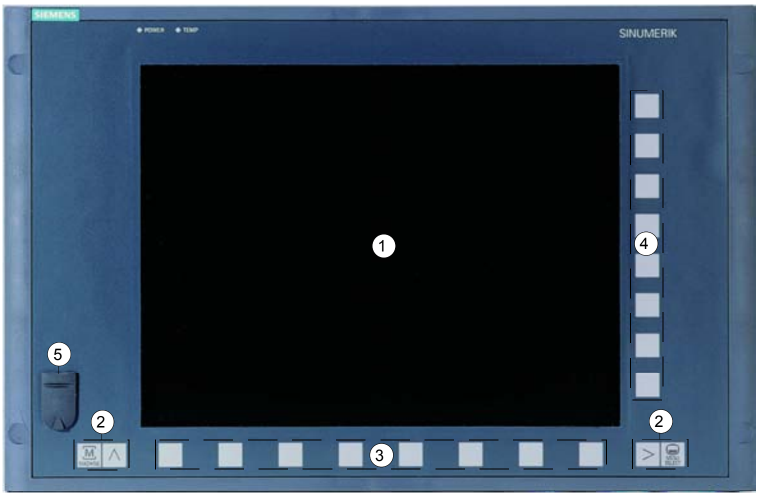
OP 012 operator panel



OP 012 operator panel

1 12" screen
2 Screen keys
3 Horizontal softkey bar
4 Vertical softkey bar
5 Alphanumeric keypad
Correction/cursor pad with control keys and input key
6 USB interface
7 Mouse

Operator panel OP 015

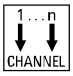


Operator panel OP 015

1 15" screen
2 Screen keys
3 Horizontal softkey bar
4 Vertical softkey bar
5 USB interface

1.2.3 Operator panel keys

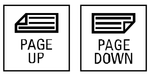
Alarm Cancel
Cancels the alarm that is marked with this symbol.



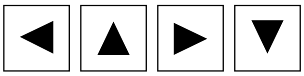
Channel
Irrelevant in ShopMill.

Help
Toggles between the process plan and programming graphics as well as between the parameterization screen form with programming graphics and the parameterization screen form with the help display.

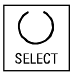
Next Window
Irrelevant in ShopMill.



Page Up or **Page Down**
Page upward or downward in the directory or in the process plan.



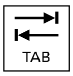
Cursor
Navigate between different fields or lines.
Use Cursor right to open a directory or program.
Use Cursor left to switch to the next highest level in the directory tree.



Select
Chooses one of a number of options presented.
This key has the same function as the "Alternat." softkey.

End
Moves the cursor to the last input field in a parameterization screen form.

Backspace
- Deletes the value in the input field.
- In insertion mode, it deletes the character after the cursor.



Tab
Irrelevant in ShopMill.

Shift
Depress the Shift key to enter the upper character shown on the dual input keys.

Ctrl
Use the following key combinations to navigate in the process plan and in the G code editor:

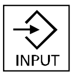
- Ctrl + Home: Jump to the beginning.
- Ctrl + End: Jump to the end.

Alt
Irrelevant in ShopMill.

Del - not with OP 031

- Deletes the value in the parameter field.
- In insertion mode, it deletes the character marked by the cursor.

Insert
Activates insertion mode or the pocket calculator.



Input

- Terminates entry of a value in the input field.
- Opens a directory or program.

Alarm - only OP 010 and OP 010C
Opens the "Messages/Alarms" operating area.
This key has the same function as the "Alarm list" softkey.

Program - only OP 010 and OP 010C
Opens the "Program" operating area.
This key has the same function as the "Prog. edit" softkey.

Offset - only OP 010 and OP 010C
Opens the "Tools/Offsets" operating area.
This key has the same function as the "Tools WOs" softkey.

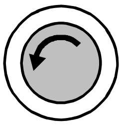
Program Manager - only OP 010 and OP 010C
Opens the "Program Manager" operating area.
This key has the same function as the "Program" softkey.

1.2.4 Machine control panels

You can equip your milling machine with a SIEMENS machine control panel or with a specific machine control panel supplied by the machine manufacturer.

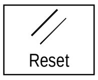
You use the machine control panel to initiate actions on the milling machine such as traversing an axis or starting the machining of a workpiece.
When functions are active, the LEDs on the corresponding keys on the machine control panel light up.

1.2.5 Elements of the machine control panels

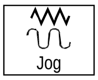


Emergency Stop button
Press this pushbutton in an emergency, i.e. when there is a danger to life or there is a risk of damage to the machine or workpiece.
All drives will be stopped with the greatest possible braking torque.

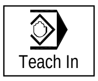
For additional responses to pressing the Emergency Stop button, please refer to the machine manufacturer's instructions.



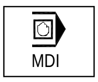
Reset
- Interrupts execution of the current program.
  The NC control remains synchronized with the machine. It is in its initial state and ready for a new program run.
- Cancels an alarm



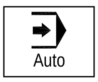
Jog
Selects Machine Manual operating mode.



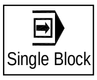
Teach In
Irrelevant in ShopMill.



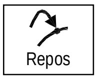
MDI
Selects MDI mode.



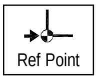
Auto
Selects Machine Auto operating mode.



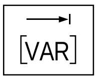
Single Block
Executes the program block by block (single block).



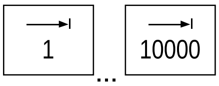
Repos
Repositions, re-approaches the contour.



Ref Point
Approaches the reference point.



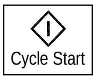
Inc Var (incremental feed variable)
Incremental mode with variable increment size.



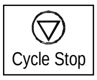
Inc (incremental feed)
Incremental mode with predefined increment size of
1, ..., 10000 increments.

A machine data code defines how the increment value is interpreted.

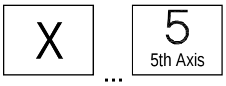
Please refer to the machine manufacturer's instructions.



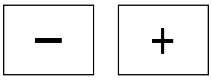
Cycle Start
Starts execution of a program.



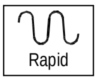
Cycle Stop
Stops execution of a program.



Axis keys
Selects an axis.

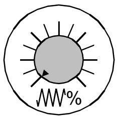


Direction keys
Traverses axis in negative or positive direction.

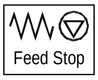


Rapid
Traverses axis at rapid traverse (fastest speed).

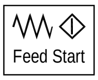
WCS MCS
Switches between the workpiece coordinate system (WCS = work) and machine coordinate system (MCS = machine).



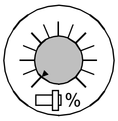
Feedrate/Rapid Traverse Override
Raises or lowers the programmed feedrate or rapid traverse.
The programmed feedrate or rapid traverse is set to 100% and can be adjusted between 0% and 120% (only up to 100% for rapid traverse). The new feedrate setting appears in the feedrate status display on the screen as an absolute value and as a percentage.



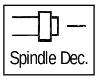
Feed Stop
Stops execution of the running program and shuts down axis drives.



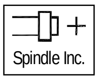
Feed Start
Continues execution of the program in the current block and ramps up to the feedrate specified in the program.



Spindle Override
Increases or decreases the programmed spindle speed.
The programmed spindle speed is set to 100% and can be controlled from 50 to 120%. The new spindle speed setting appears in the spindle status display on the screen as an absolute value in percent.

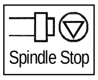


Spindle Dec. – only OP032S machine control panel
Decreases the programmed spindle speed.

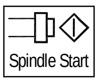


Spindle Inc. – only OP032S machine control panel
Increases the programmed spindle speed.

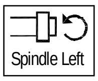
100 % – only OP032S machine control panel
Restores the programmed spindle speed.



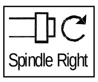
Spindle Stop
Stop spindle.



Spindle Start
Start spindle.



Spindle Left – machine control panel OP032S only
Starts spindle (CCW rotation).



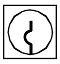
Spindle Right – machine control panel OP032S only
Starts spindle (CW rotation).

Keyswitch

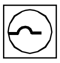
You can use the keyswitch to set various access rights. The keyswitch has four settings for protection levels 4 to 7.
Machine data can be programmed to interlock access to programs, data, and functions at various protection levels.

Please refer to the machine manufacturer's instructions.

The keyswitch has three keys of different colors that you can remove in the specified positions:

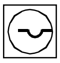


Position 0
No key
Protection level 7



Position 1
Key 1 **black**
Protection level 6

Position 2
Key 1 **green**
Protection level 5



Position 3
Key 1 **red**
Protection level 4

Lowest
access authorization

↓

Increasing
access authorization

↓

Highest
access authorization

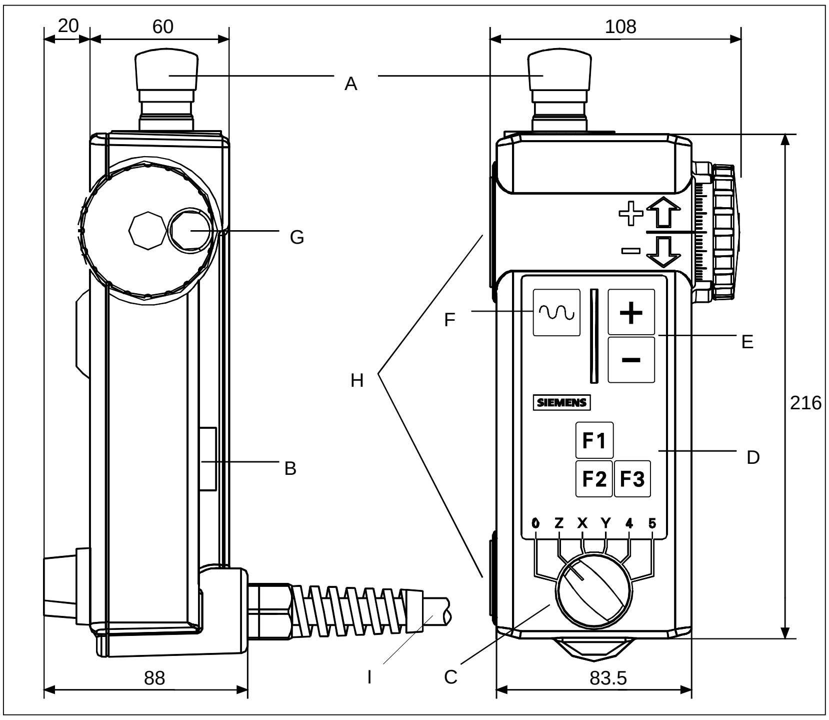
When you change the key position to change the access authorization, this is immediately not visible on the operator interface. You have to initiate an action first (e.g. close or open a directory).

If the PLC is in the STOP state (LEDs on the machine control panel are flashing), ShopMill will not read the keyswitch settings as it boots.

The machine manufacturer can set up protection levels 0 to 3 using a password. If this password is set, ShopMill does not read the keyswitch setting.

Please refer to the machine manufacturer's instructions.

1.2.6 Mini handheld unit



A EMERGENCY STOP button, two-channel
B Enabling key, two-channel
C Axis selector switch for 5 axes and neutral position
D Function keys F1, F2, F3
E Traversing keys, directions +, -
F Rapid traverse key for high-speed travel with traversing keys or handwheel
G Handwheel
H Magnets for attachment to metal parts
I Connecting cable 1.5 m ... 3.5 m

Control elements

EMERGENCY STOP button

The EMERGENCY STOP button must be pressed in an emergency

1. when a person is at risk,
2. when there is a danger of the machine or workpiece being damaged.

Enabling button

The enabling button is designed as a 2-way switch. It must be pressed to initiate traversing movements.

Axis selection switch

You can select up to 5 axes with the axis selector switch.

Function keys

The function keys can be used to trigger machine-specific functions.

Traversing keys

The +, - traversing keys can be used to trigger traversing movements on the axis selected via the axis selection switch.

Handwheel

The handwheel can be used to initiate movements at the axis selected using the axis selection switch. The handwheel supplies two guide signals with 100 I/U.

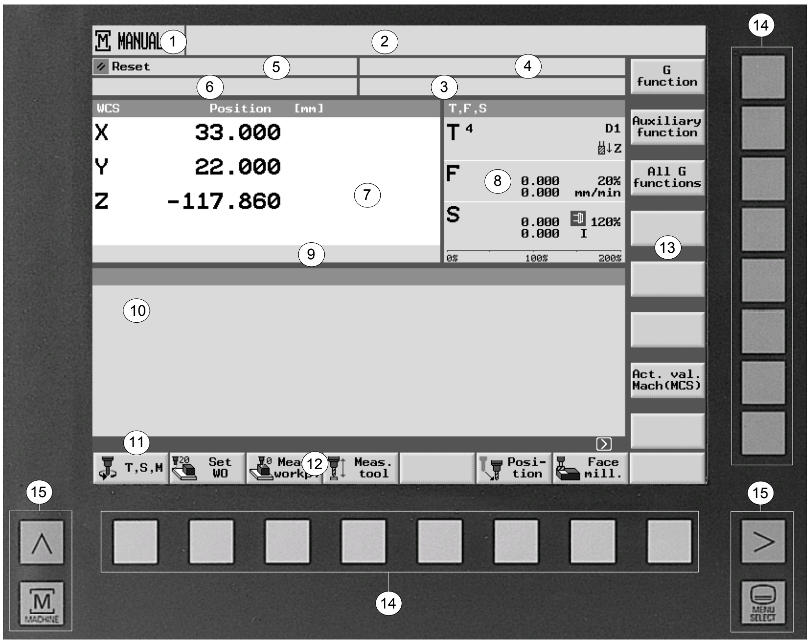
Rapid traverse key

The rapid traverse key increases the traversing speed of the axis selected with the axis selector switch. The rapid traverse key acts both on travel commands from the +/- keys and on the handwheel signals.

1.3 User interface

1.3.1 Overview

Screen layout



User interface

1 Active operating mode/operating area and secondary mode
2 Alarm and message line
3 Program name
4 Program path
5 Channel state and program control
6 Channel operational messages
7 Position display of the axes
8 Display for
- active tool T
- current feedrate F
- spindle S
- spindle utilization in percent

9 Display of active work offsets and rotation
10 Working window
11 Dialog line for additional explanatory text
12 Horizontal softkey bar
13 Vertical softkey bar
14 Softkeys
15 Screen buttons

Secondary mode

REF: Approaching a reference point
REPOS: Repositioning
INC1 ... INC10000: Fixed increment
INC_VAR: Variable increment

Channel status

RESET
Active
Interrupted

Program control

SKP: Skip G code block
DRY: Dry run feedrate
!ROV: Feedrate override only (not feedrate and rapid traverse override)
SBL1: Single block (stop after every block that triggers a function on the machine)
SBL2: Not possible to select in ShopMill (stop each every block)
SBL3: Single block fine (stop after every block, even within a cycle)
M01: Programmed stop
DRF: DRF Offset
PRT: Program test

Channel operational messages

Stop: An operator action is required.
Wait: No operator action is required.

If a dwell time is active, the remaining dwell time is displayed. It is either displayed in seconds or as spindle revolutions.

Position display of the axes

The actual value display in the position display refers to the SZS coordinate system (settable zero system). The position of the active tool relative to the workpiece zero is displayed.

Symbols used for axis display
Linear axis clamped
Rotary axis clamped

Feedrate status

Feedrate is not enabled

Spindle status

Spindle not enabled

Spindle is stationary

Spindle is turning clockwise

Spindle is turning counterclockwise

The display of the spindle utilization as a percentage can be 200 %.

Please refer to the machine manufacturer's instructions.

Key to the meaning of the symbol colors:

Red: Machine is stationary

Green: Machine is running

Yellow: Waiting for operator to take action

Gray: Miscellaneous

Screen buttons

Machine

Call active operating mode (Machine Manual, MDI, or Machine Auto).

Return

Irrelevant in ShopMill.

Expansion

Changes the horizontal softkey bar.

Menu Select

Calls the main menu:

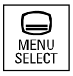
The machine manufacturer can display defined symbols instead of the program path (4). The program path is then displayed together with the program name (3).

Please refer to the machine manufacturer's instructions.

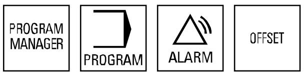
1.3.2 Operation by means of softkeys and hardkeys

The ShopMill user interface consists of different screens featuring eight horizontal and eight vertical softkeys. You operate the softkeys with the keys next to the softkey bars.
Each softkey displays a new screen form.

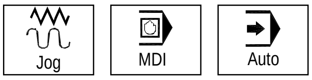
ShopMill has 3 operating modes (Machine Manual, MDI, and Machine Auto) and 4 operating areas (Program Manager, Program Messages/ Alarms, and Tools/Work Offsets).



To switch from one operating mode/operating area to another, press the "Menu Select" key. The main menu is displayed, in which you can select the appropriate operating area via a softkey.

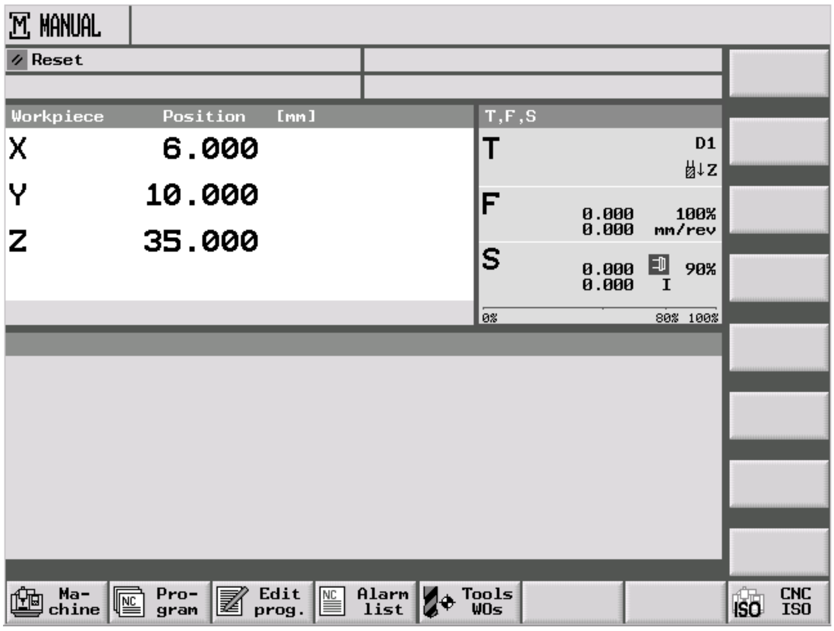


Alternatively, you can call the operating areas via the hardkeys on the operator panel.

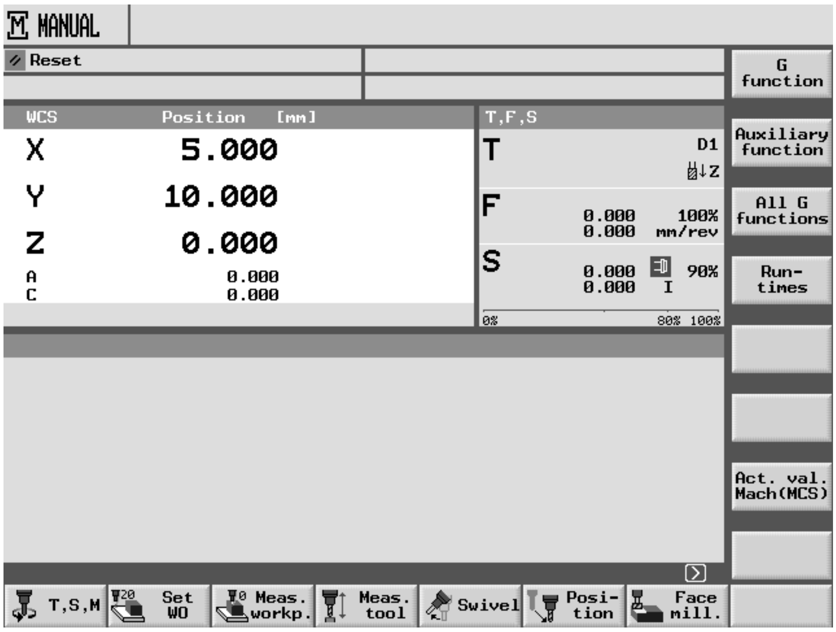


You can activate an operating mode directly at any time via the keys on the machine control panel.
If you select the "Machine" softkey in the main menu, the screen form for the currently active mode appears.

If you select another operating mode or operating area, the horizontal and vertical softkey bars change.

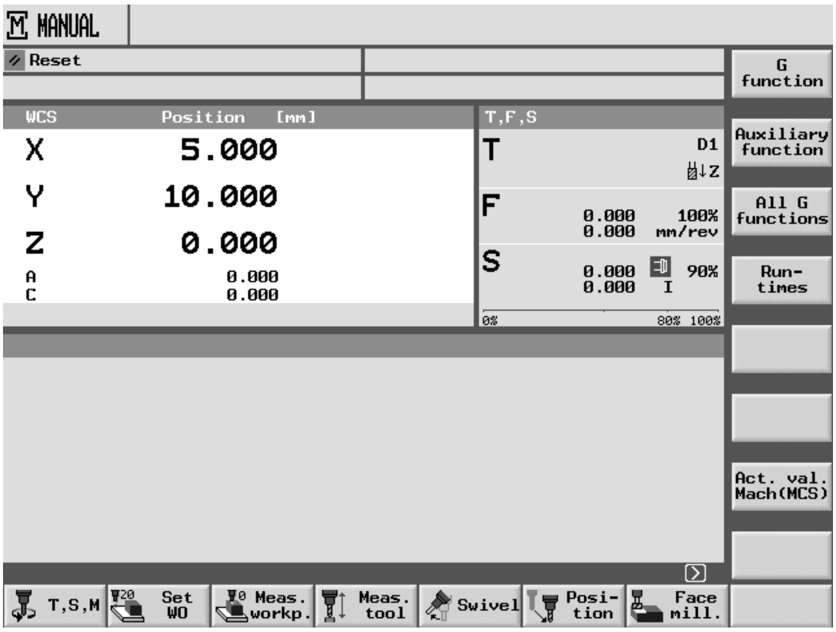


Main menu

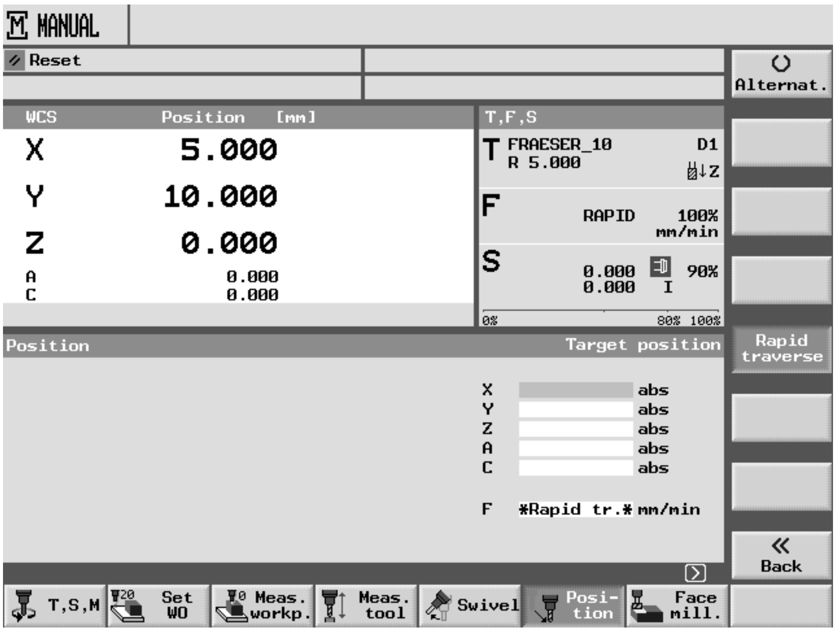


Machine Manual operating mode

If you press a horizontal softkey within an operating mode or operating area, only the vertical softkey bar will change.



Machine Manual operating mode



Function within Machine Manual operating mode

>

When the symbol appears to the right of the dialog line on the operator interface, you can change the horizontal softkey bar within an operating area. This is done by pressing the "Expansion" key. Pressing the "Expansion" key again will take you back to the original horizontal softkey bar.

Back

Within an operating mode or operating area, you can use the "Back" softkey to return to the next highest screen form.

Abort

Use the "Abort" softkey to exit a screen form without accepting the entered values and return to the next highest screen form.

Accept

When you have entered all the necessary parameters in the parameterization screen form correctly, you can close the screen form and save the parameters using the "Accept" softkey.

OK

Use the "OK" softkey to initiate an action immediately, e.g. to rename or delete a program.

Program test On

Program test Off

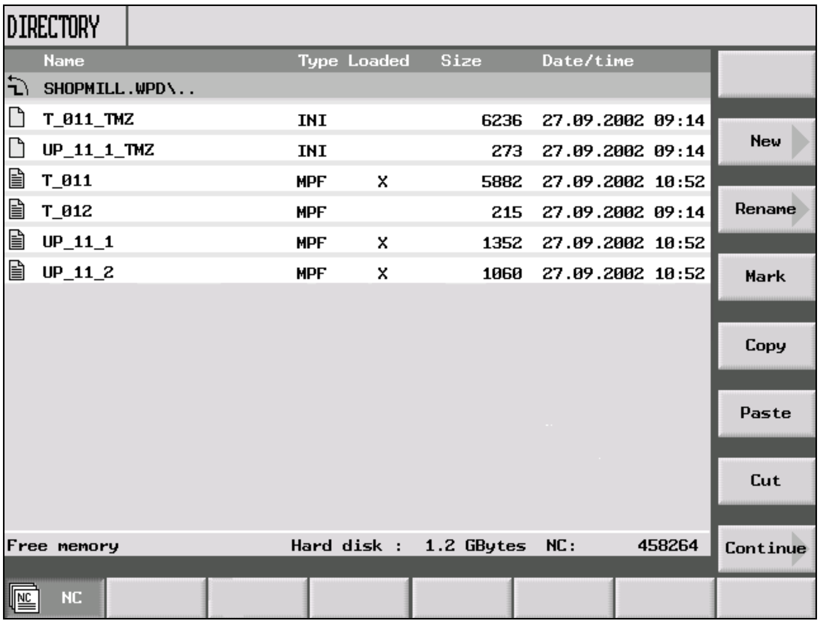
Some softkeys are displayed with a black background when you activate the function assigned to them.
In that case, you can deactivate the function by pressing the softkey again. The softkey will then have a gray background again.

1.3.3 Program views

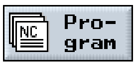
You can display a sequential control program in various views.

Program manager

In the program manager, you manage all your programs. You can also select a program here for machining the workpiece.

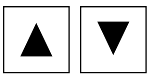


Program manager

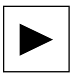


-or-

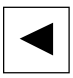

Select the program manager with the "Program" softkey or "Program Manager" key.



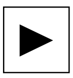
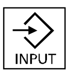
You can move around within a directory using the "Cursor up" and "Cursor down" keys.



Use the "Cursor right" key to open a directory.



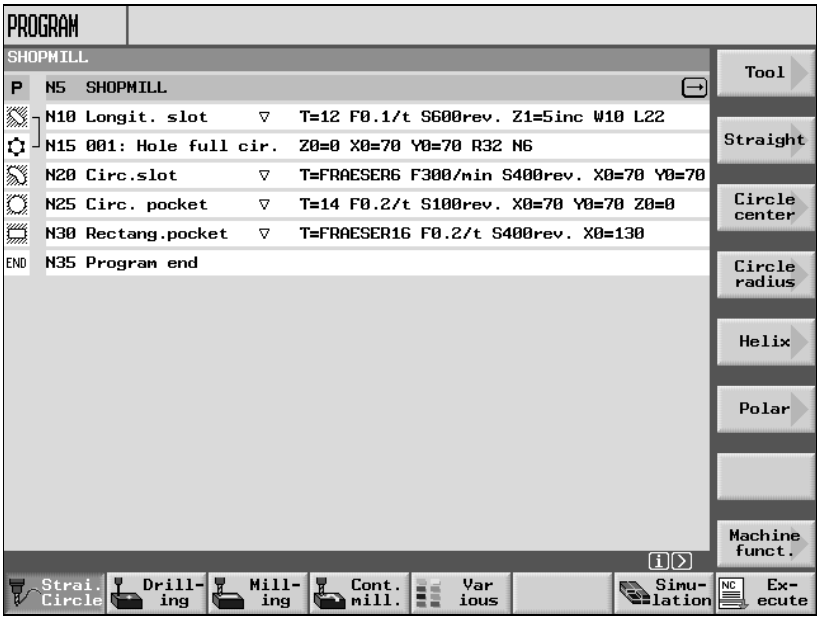
Use the "Cursor left" key to move up to the next-higher directory level.


-or-


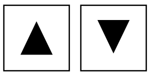
Use the "Cursor right" or "Input" key to open the process plan for a program.

Process plan

The process plan provides an overview of the separate machining steps of a program.



Process plan

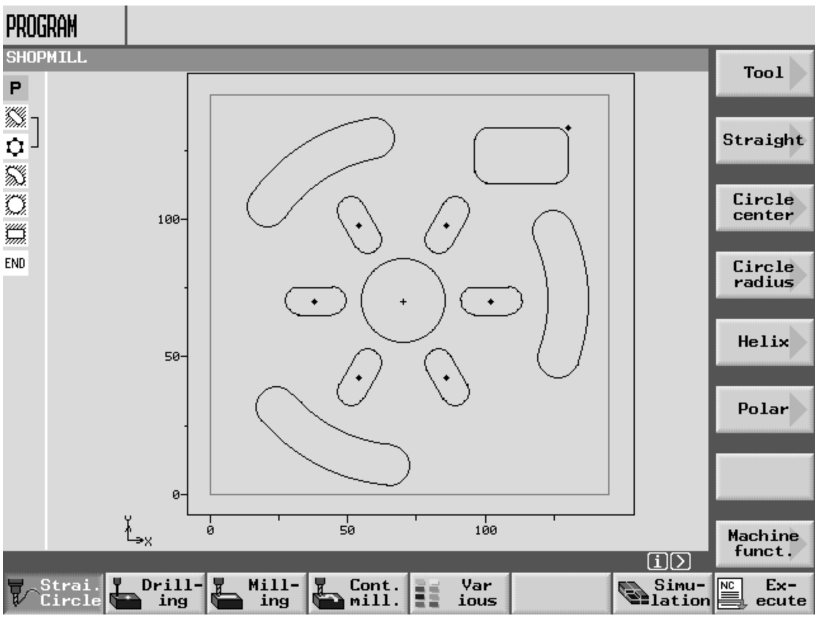


You can move between the program blocks in the process plan using the "Cursor up" and "Cursor down" keys.

Use the "Help" key to switch between the process plan and the programming graphics.

Programming graphics

The programming graphics display a dynamic broken-line top view of the workpiece. The program block selected in the process plan is color-highlighted in the programming graphics.

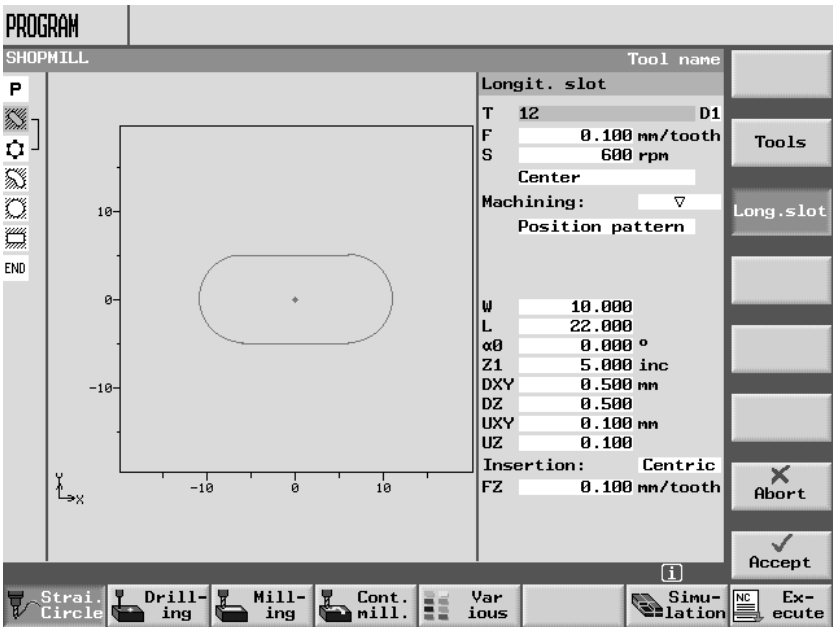


Programming graphics

Use the "Cursor right" key to open a program block in the process plan. The appropriate parameterization mask complete with programming graphics is then displayed.

Parameter screen with programming graphics

The programming graphics in a parameterization screen form show the contour of the current machining step in broken-line graphics complete with the parameters.



Parameter screen with programming graphics

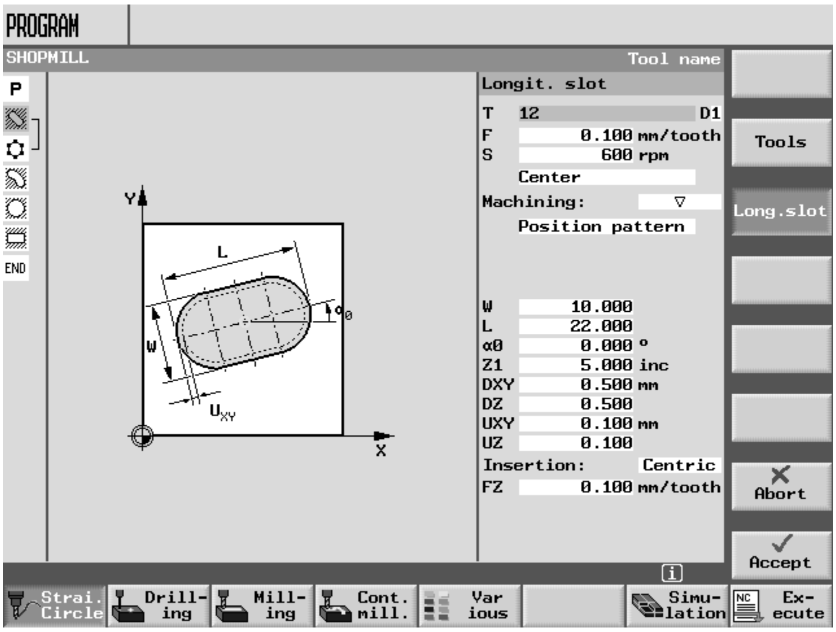
Use the cursor keys to move between the input fields within a parameterization screen form.

HELP

Use the "Help" key to switch between the programming graphics and the help display.

Parameter screen with help display

The help display in the parameterization screen form explains the parameters of the machining step individually.

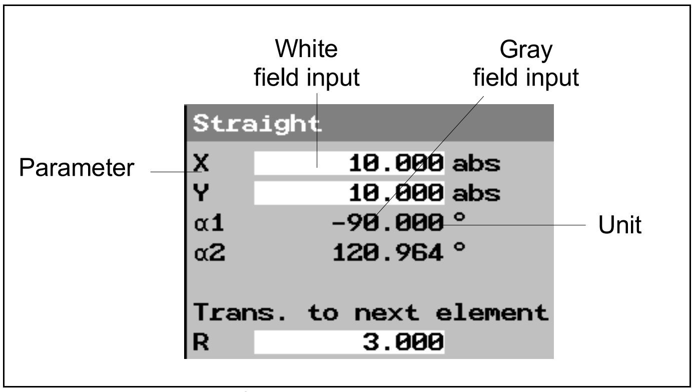


Parameter screen with help display

The colored symbols in the help displays have the following meaning:
Yellow circle = reference point
Red arrow = tool traveling at rapid traverse
Green arrow = tool traveling at machining feedrate

1.3.4 Entering parameters

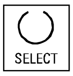
On setting up the machine and during programming, you must enter values in the white fields for various parameters.
Parameters that have a gray input field are automatically calculated by ShopMill.



Parameterization screen form

Selecting a parameter

Some parameters require you to select from a number of options in the input field. Fields of this type do not allow you to type in a value.

-or-


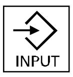
➢ Press the "Alternat." softkey or the "Select" key until the required setting is displayed.

The "Alternat." softkey is only visible when the cursor is positioned on an input field that presents a choice of options. The "Select" key is also only active in this situation.

Entering a parameter

For the remaining parameters, enter a numerical value in the input field using the keys on the operator panel.

➢ Enter the desired value.



➢ Press the "Input" key to terminate entry.

-or-


If you do not want to enter a value, i.e. not even "0", press the "Backspace" or "Del" key.

Selecting the unit

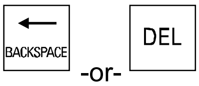
For certain parameters, you can choose between different units.

➢ Press the "Alternat." softkey or the "Select" key until the required unit is displayed.

The "Alternat." softkey is only visible when you have a choice of units for this parameter. The "Select" key is also only active in this situation.

Deleting a parameter

If an input field contains an invalid value, you can delete it completely.



➢ Press the "Backspace" or "Del" key.

Changing or calculating parameters

If you only want to change individual characters in an input field rather than overwriting the entire entry, switch to insert mode. In this mode, the pocket calculator is also active. You can use it during programming to calculate parameter values.

➢ Press the "Insert" key.

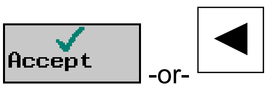
Insert mode and the pocket calculator are activated.

You can move around within the input field using the "Cursor left" and "Cursor right" keys.
Use the "Backspace" or "Del" key to delete individual characters.

For more information on the pocket calculator, see Sec. "Pocket calculator".

Accepting a parameter

When you have correctly entered all the necessary parameters in the parameterization screen form, you can close the screen form and save the parameters.



➢ Press the "Accept" softkey or the "Cursor left" key.
If there are several input fields in a line and you want to use the "Cursor left" key to accept the parameters, you must position the cursor in the leftmost input field.

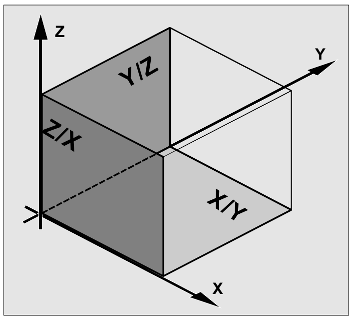
You cannot accept the parameters if they are incomplete or obviously erroneous. In this case, you can see from the dialog line which parameters are missing or were entered incorrectly.

1.4 Fundamentals

1.4.1 Plane designation

A plane is defined by means of two coordinate axes. The third coordinate axis (tool axis) is perpendicular to this plane and determines the infeed direction of the tool (e.g. for 2½-D machining).

When programming, it is necessary to specify the working plane so that the control system can calculate the tool offset values correctly. The plane is also relevant to certain types of circular programming and polar coordinates.



Working planes are defined as follows:

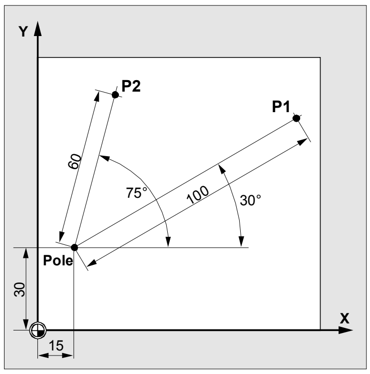
Plane	Tool axis
X/Y	Z
Z/X	Y
Y/Z	X

1.4.2 Polar coordinates

The rectangular coordinate system is suitable in cases where dimensions in the production drawing are orthogonal. For workpieces dimensioned with arcs or angles, it is better to define positions using polar coordinates. This is possible if you are programming a straight line or a circle (see Section "Programming simple path motions").

Polar coordinates have their zero point in the "pole".

Example:
Points P1 and P2 can then be described – with reference to the **pole** – as follows:
`P1`: radius =100 plus angle =30°
`P2`: radius =60 plus angle =75°

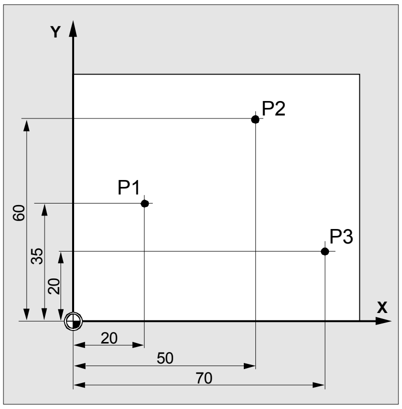


1.4.3 Absolute dimensions

With absolute dimensions, all the positional data refer to the currently valid zero point. Applied to tool movement this means:

The absolute dimensions describe the position to which the tool is to travel.

Example:
The positional parameters for points P1 to P3 in absolute dimensions **relative to the zero point** are the following:
`P1:` X20 Y35
`P2:` X50 Y60
`P3:` X70 Y20

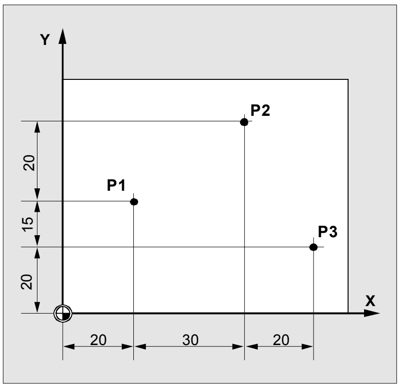


1.4.4 Incremental dimensions

In the case of production drawings in which dimensions refer to some other point on the workpiece rather than the zero point, it is possible to enter an incremental dimension.

With incremental dimension input, each item of positional data refers to a point programmed beforehand.

Example:
The positional data for points P1 to P3 in incremental dimensions are:
`P1:` X20 Y35 ;(relative to the zero point)
`P2:` X30 Y20 ;(relative to P1)
`P3:` X20 Y -35 ;(relative to P2)



1.4.5 Pocket calculator function

Function

Precondition

The cursor is positioned on a parameter field.

Press the "Insert" key

or
Equals key
to switch to **pocket calculator mode**.
Once you have pressed this key, enter one of the basic arithmetic operators (+, -, *, /), then enter a value,
then press "Input", and then enter a second value to obtain the result of the arithmetic operation.

Example:
Suppose we want to add a tool wear of + 0.1 in length L for a tool.

- Place the cursor in the appropriate parameter setting field,
- Press the Equals key to open the parameter field and
- Add the new wear value to the existing value,
  e.g. 0.5 **+ 0.1**
- Complete the calculation by pressing the "Input" key.
  Result: 0.6

■

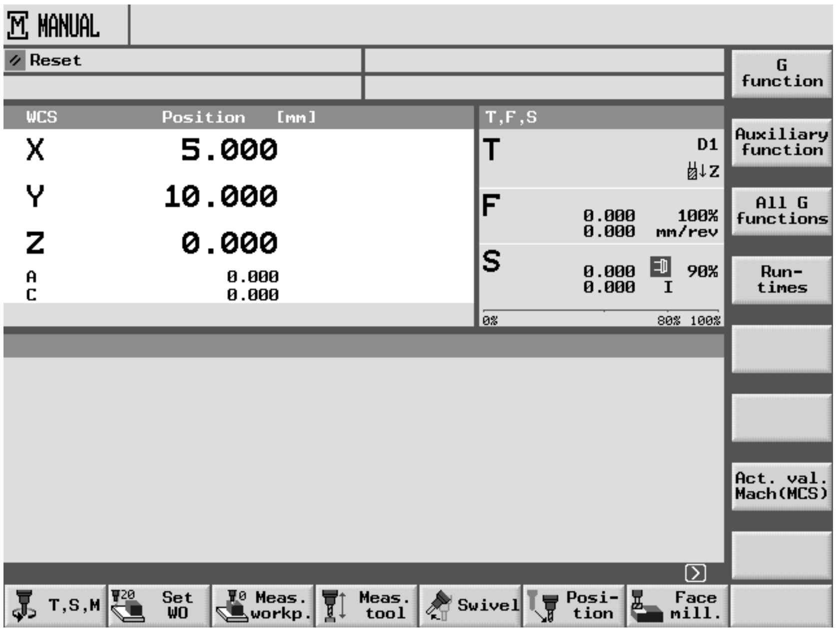
Operation

2.1 Switching on and off

There are different ways of switching the control and the entire system on and off.

Please refer to the machine manufacturer's instructions.

After power ON, the main "Machine Manual" display appears on the screen



Main "Machine Manual" display

2.2 Reference point approach

The "Ref Point" function ensures that the control and machine are synchronized after power ON.
Various reference point approach methods may be employed.

Please refer to the machine manufacturer's instructions.

- Reference point approach can only be performed by machine axes. The actual value display does not match the real position of the axes when the control is switched on.
- Reference point approach is necessary on machines without an absolute measuring system!

Warning

If the axes are not positioned safely, then you must reposition them accordingly. You must observe the axis motions directly on the machine!
Ignore the actual value display until the axes have been referenced!
The software limit switches are not active!

Referencing axes

Jog

➢ Select "Machine Manual" mode.

Ref Point

➢ Press the "Ref Point" key.

X ...

➢ Select the axis to be traversed.

− +

➢ Press the "+" or "-" key.

The selected axis moves to the reference point. The direction and sequence is defined by the machine-tool manufacturer in the PLC program.
If you have pressed the wrong direction key, the action is not accepted and the axes do not move.
The display shows the reference point value.

No symbol appears for axes that have not been referenced.

This symbol is shown next to an axis if it has been referenced.

Interrupting axis motion

Feed Stop

➢ Press the "Feed Stop" key.

The axis stops.

Re-approaching an axis

X ...

➢ Select the axis to be traversed.

− +

➢ Press the "+" or "-" key.

The selected axis moves to the reference point.

The machine is synchronized as soon as the reference point is reached. The actual value display is set to the reference point value. The display is the difference between the machine zero and the slide reference point. From now on path limits, such as software limit switches, are active.

End the function via the machine control panel by selecting operating mode "Machine Auto" or "Machine Manual".

- You can reference all axes simultaneously (depending on the PLC program supplied by the machine tool manufacturer).
- The feedrate override is operative.

The machine manufacturer can define the order in which the axes will be referenced.

Only when all axes with a defined reference point have reached this point will you be able to start with the "Cycle Start" key in "Machine Auto".

2.2.1 User agreement in Safety Integrated

If you are using Safety Integrated (SI) on your machine, you will need to confirm that the current displayed position of an axis corresponds to its actual position on the machine when you reference an axis. Your confirmation is the precondition for the availability of other Safety Integrated functions.

You can only give your user agreement for an axis after it has approached the reference point.

The displayed axis position always refers to the machine coordinate system (MCS = machine).

User agreement with Safety Integrated is only possible with a software option.

For more information on user agreement, please refer to:
References: /FBSI/, Description of Functions SINUMERIK Safety Integrated

Jog

➢ Select "Machine Manual" mode.

Ref Point

➢ Press the "Ref Point" key on the machine control panel.

X ...

➢ Select the axis to be traversed.

− +

➢ Press the "+" or "-" key.

The selected axis moves to the reference point and stops. The coordinate of the reference point is displayed. The axis is marked .

User agreement

➢ Press the "User agreement" softkey.

The "User agreement" window opens.
It shows a list of all machine axes with their current and SI positions.

➢ Position the cursor in the "Agreement" field for the axis in question.

Alternat.
- or -
SELECT

➢ Give your agreement by pressing the "Alternat." softkey or the "Select" key.

The selected axis is marked with a cross meaning "safely referenced" in the "Agreement" column.

Pressing the "toggle keys" again removes your agreement.

2.3 Operating modes

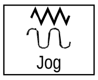
Three different operating modes can be used in ShopMill:

- Manual mode
- MDI (Manual Data Automatic)
- Automatic mode

Manual mode

Manual mode is used for the following preparatory actions:

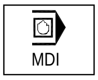
- Reference point approach, i.e. calibration of the position measuring system
- Swiveling the workpiece
- Preparing a machine for executing a program in automatic mode, i.e. measuring tools, measuring the workpiece and, if necessary, defining the work offsets used in the program
- Traversing axes, e.g. during a program interruption
- Positioning axes
- Face milling a workpiece



You can select manual mode by pressing the "Jog" softkey.
The parameters set under "T, S, M..." affect all movements in manual mode with the exception of reference point approach.

MDI

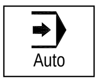
In MDI mode, you can enter and execute G code commands non-modally to set up the machine or to perform a single action.



You can select MDI mode via the "MDI" key.

Automatic mode

In automatic mode, you can execute a program completely or only partially. You can also trace execution of the program in a graphical display on the screen.



You can select "Machine Auto" via the "Auto" key.

2.4 Settings for the machine

2.4.1 Switching over the unit (millimeter/inch)

This function enables you to switch between the metric and inch dimension systems depending on the dimension units used in your production drawing.

Every dimension system switchover applies to the entire machine, i.e. all relevant measurement data are automatically converted to the new dimension system, e.g.

- Positions
- Tool offsets
- Work offsets

Jog >

➢ Open the extended horizontal softkey bar in "Machine Manual" mode.

ShopM. sett.

➢ Press the "ShopM. sett." softkey.

Inch

➢ Press the "Inch" softkey to switch to **Inch**.

The "Inch" softkey is active.

Inch

➢ Press the "Inch" softkey to switch to **metric**.

The "Inch" softkey is not active.
When you press the "Inch" softkey, a box appears asking you to confirm switchover.

OK

The dimension system is adjusted accordingly if you confirm with the "OK" softkey.

2.4.2 Switching over the coordinate system (MCS/WCS)

The machine coordinate system (MCS) is the original system of your machine. Unlike the workpiece coordinate system (WCS), it does not allow for tool offsets, work offset, scalings, etc.

You can switch between the machine and workpiece coordinate systems by following the sequence below.

WCS MCS

- Press the "WCS MCS" key on the machine control panel

-or-

Jog Auto

- Select "Machine Manual" or "Machine Auto" mode.

Actual value MCS

- Press the "Actual value MCS" softkey to switch to **MCS**.

The "Actual value MCS" softkey is active.

Actual value MCS

- Press the "Actual value MCS" softkey to switch from MCS to **WCS**.

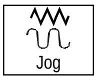
The "Actual value MCS" softkey is not active.

2.5 Setting a new position value

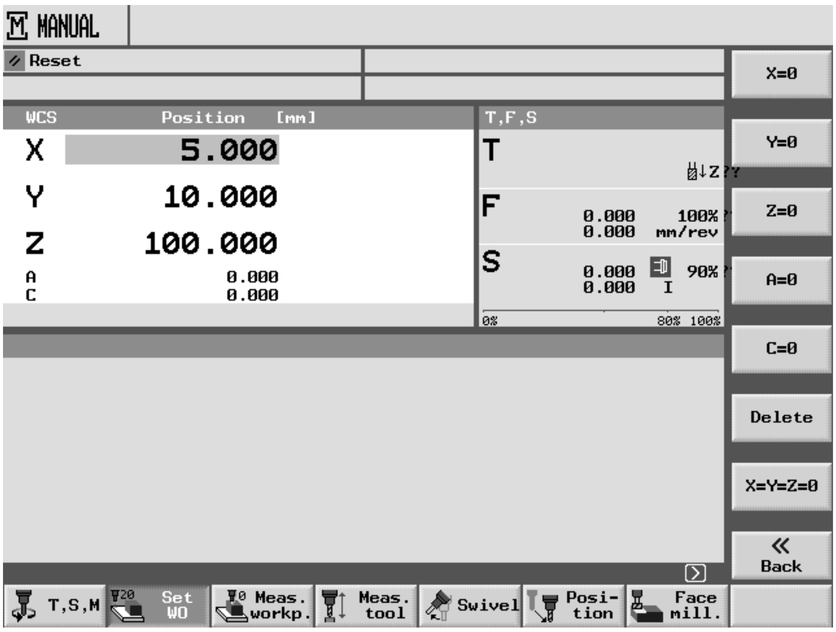
You can use the "Set work offset" function to enter a new position value for each individual axis in the actual value display.
The difference between the position value in the machine coordinate system **MCS** and the new position value in the workpiece coordinate system **WCS** is saved in the currently active work offset or in the basic offset.

Please refer to the machine manufacturer's instructions.

If the values are stored in the active work offset, they are stored in the coarse offset and existing values in the fine offset are deleted.
The currently active work offset is displayed under the position window for the axes.



- Move the machine axes to the desired position (e.g. workpiece surface).
- Select the "Set WO" menu in "Machine Manual" operating mode.



Base offset menu

Setting a position value

- Enter the new position values with the keyboard.
  You can use the cursor keys to switch between positions.
- Press the "Input" key to complete your entry.

-or-

- Press softkeys "X=0","Y=0", and "Z=0", to set the position values to zero.

Resetting an offset

Delete

- Press the "Delete" softkey.

The offset is canceled again.

The work offsets (WO1 etc.) are based on the base offset.

2.6 Measuring workpiece zero

The reference point for programming a workpiece is always the workpiece zero. You can determine the workpiece zero on the following workpiece elements:

- Edges
- Corner
- Pocket/hole
- Spigot
- Plane

You can measure the workpiece zero either manually or automatically.

Manual measurement

To measure the zero point manually, you need to traverse your tool manually up to the workpiece. You can use edge probes, sensing probes, or dial gauges with known radii and lengths. You can also use any other tool of which you know the radius and length.
The tools used for measuring must not be of type 3D probe.

Automatic measurement

For automatic measurement always use electronic 3D measuring probes, which you must first calibrate.
When measuring automatically, first position the tool manually. As soon as you start the process with the "Cycle Start" key, the probe automatically approaches the workpiece at measuring feedrate and then returns to the starting position at rapid traverse.

For automatic workpiece zero measurement, the machine manufacturer must have first set up the measuring cycles. This includes defining the measuring feedrate in the machine data.

Please refer to the machine manufacturer's instructions.

To obtain the desired measuring results, you must keep to the measuring point sequence shown in the help displays.
You can reject measuring points and then measure them again. This is done by pressing the softkey that is currently active (measured value). In manual measurement, you can reset values in any order but in automatic measurement, only in reverse order.

Measurement only

If you "only" want to measure the workpiece zero, the measured values are merely displayed without changing the coordinate system.

Work offset

You usually store the measured workpiece zero in a work offset. With ShopMill you can measure rotations and offsets. If necessary, you may first have to measure the rotations of your workpiece to align your workpiece and then define the zero point by measuring the offsets.

Aligning

Alignment can be performed either by rotating the coordinate system or by rotating the workpiece with a rotary axis. If your machine is equipped with two rotary axes and the "swivel" function is set up, you can also align an inclined plane.

Zero point

The measurement values for the offsets are stored in the coarse offset and the relevant fine offsets are deleted. If the zero point is stored in a non-active work offset, an activation window is displayed with which you can activate this work offset directly.

Rotary axes

If your machine has rotary axes, you can include these rotary axes in the measurement and setup procedure. If you store the workpiece zero in a work offset, rotary axis positioning may be necessary in the following cases.

- Correcting the work offset requires you to position the rotary axes to align the workpiece parallel with the coordinate system, e.g. with "Align edge".
- Correcting the work offset rotates the workpiece coordinate system, which should align the tool perpendicular to the plane, e.g. for "Align plane".

You are supported by one or two activation windows when you position the rotary axes (see Sec."Corrections after measuring the zero point").

You can only select "Rotary axis A, B, C" for the "Angle corr." parameter, if your machine has rotary axes. They must also be assigned to geometry axes in the machine data.

Please refer to the machine manufacturer's instructions.

Sequence of operations

To measure the workpiece zero, the tool must always be perpendicular to the machining plane (e.g. with "Align plane"). In some measuring methods, the workpiece must first be aligned parallel to the coordinate system (set edge, distance 2 edges, rectangular pocket, rectangular spigot).
To do this, it may be necessary to perform the measurement in several stages.

1. "Align plane" (to align the tool perpendicular to the plane)
2. "Align edge" (to align the workpiece parallel with the coordinate system)
3. "Set edge", "Distance 2 edges", "Rectangular pocket", or "Rectangular spigot" (to determine the zero point)

Or

1. "Align plane" (to align the tool perpendicular to the plane)
2. "Corner", "Holes", or "Spigots" (to align the tool parallel with the coordinate system and define the zero point)

Prepositioning

If you want to preposition a rotary axis before measuring with "Align edge", move the rotary axis so that your workpiece is already approximately parallel to the coordinate system. Set the relevant rotary axis angle to zero with "Set WO" Measurement with "Align edge" will then correct the value for rotary axis offset or include it in the coordinate rotation and align the workpiece edge precisely.

If you want to preposition your workpiece with "Align edge" prior to measurement, you can set the angle values under "Manual swivel". With "Set zero plane" you transfer the resulting rotations into the active work offset. The measurement with "Align edge" will then correct the value for the coordinate rotations and align the workpiece precisely.

If the function "Swivel" is set up on your machine, we recommend effecting a swivel motion to zero before starting measurement. In that way, you will ensure that the rotary axis positions comply with the current coordinate system.

Examples

Two typical examples are given below that demonstrate the interaction between and the use of "Measure workpiece" and "Manual swivel" when measuring and aligning workpieces:

Example 1:
Remachining on a cylinder head with 2 holes on an inclined plane.

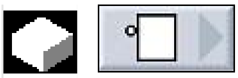
1. Clamp the workpiece
2. Insert probes T, S, M and activate the required work offset.
3. Preposition workpiece
   rotate rotary axes manually until the inclined surface is almost perpendicular to the tool axis.
4. Manual swiveling
   Select "direct" swiveling, "Teach rotary axes", and press "Cycle start".
5. Manual swiveling
   Apply "Set zero plane" to store the resulting rotations in the work zero.
6. Measure workpiece
   Apply "Align plane" to correct the alignment of the workpiece.
7. Measure workpiece
   Apply "2 holes" to define the rotation and offset in the XY plane.
8. Measure workpiece
   Apply "Set edge Z" to define the offset in Z.
9. Start part program to remachine in AUTO.
   Start the program with swivel zero.

Example 2:
Measuring workpieces in swiveled position. The workpiece is to be probed in the X direction even though the probe cannot approach the workpiece in the X direction because of an obstructing edge. But with a swivel movement, the measurement in the X direction can be replaced by a measurement in the Z direction.

1. Clamp the workpiece
2. Insert probes T, S, M and activate the required work offset.

3. Swivel manually
   With "direct" swiveling enter the required rotary axis positions or with "axis by axis" the required rotations (e.g. Y=-90) and "cycle start".
4. Measure the workpiece
   Apply "Apply edge Z": The measured offset in Z is converted and entered as an X value in the chosen work offset.
5. Swivel manually
   Execute swivel to zero in order to rotate the coordinate system to its initial position.

2.6.1 Measuring an edge

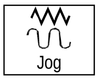


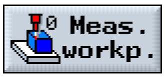
The following options are available to you when measuring an edge:

- Set the edge
  The workpiece lies parallel to the coordinate system on the work table. You measure one reference point in one of the axes (X, Y, Z).
- Align the edge
  The workpiece lies in any direction, i.e. not parallel to the coordinate system on the work table. By measuring two points on the workpiece edge you determine the angle with the coordinate system.
- Distance 2 edges
  The workpiece lies parallel to the coordinate system on the work table. You measure distance L of two parallel workpiece edges in one of the axes (X, Y, or Z) and determine its center.

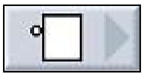
Setting an edge manually

1. Attach any tool for scratching to the spindle.

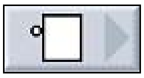




2. In "Machine Manual" mode, select the "Meas. workp." softkey.



3. Press the "Edge" softkey.
The "Edge" window opens with new vertical softkeys.



4. Press the "Set edge" softkey.

5. Select "Measurement only", if you only want to display the measured values.
- OR -
➢ Select "Work offset" and the work zero in which you want to store the zero point (e.g. base reference).

- OR -

Work offset

➢ Press the "Work offset" softkey.

The "work offset list" is displayed.

➢ Place the cursor on the chosen work offset (e.g. base reference).

In manual

➢ Press the "In manual" softkey.

X | Z

6. Use the softkeys to select in which axis direction you want to approach the workpiece first.

7. Select the measuring direction (+ or -) you want to approach the workpiece in.

8. Specify the setpoint position of the workpiece edge you are approaching.
The setpoint position corresponds, e.g. to the dimension specifications of the workpiece edge from the workpiece drawing.
9. Traverse the tool to the workpiece edge.

Set WO -or- Calculate

10. Press the "Set WO" or "Calculate" softkey.

The position of the workpiece edge is calculated and displayed.
The set position of the workpiece edge is stored as the new zero point when "Set WO" is pressed. The tool radius is automatically included in the calculation.

Example: Reference point workpiece edge X1 = -50
Approach direction: +
Tool radius = 3 mm
⇒ Work offset X = 53

11. Repeat the measurement procedure (steps 6 to 10) for the two other axes, if necessary.

Setting an edge automatically

1. Attach a 3D probe type tool to the spindle.
2. Prepare the measurement (as described under "Setting the edge manually", steps 2 to 8).
3. Move the tool up close to the workpiece edge you want to measure.
4. Press the "Cycle Start" key.

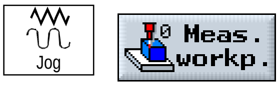
Cycle Start

This starts the automatic measuring process. The position of the workpiece edge is measured.
The position of the workpiece edge is calculated and displayed.
The set position of the workpiece edge is stored as the new zero point if you have selected "work offset". The tool radius is automatically included in the calculation.

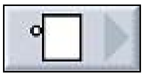
5. Repeat the measurement procedure (steps 3 to 4) for the two other axes, if applicable.

Aligning an edge manually

1. Attach any tool for scratching to the spindle.

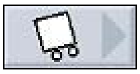


2. In "Machine Manual" mode, select the "Meas. workp." softkey.



3. Press the "Edge" softkey.

The "Edge" window opens with new vertical softkeys.



4. Press the "Align edge" softkey.
5. Specify whether you want "Measurement only" or in which work offset you want to store the zero point (as described under "Setting the edge manually", step 5).

6. Under "Measuring axis", select the axis in which you want to approach the workpiece.
7. Under "Reference axis", select the axis to which the angle to be measured refers.
8. Under "Angle offs.", select the "Coor. Rotation" entry.

- OR -

➢ Under "Angle offs.", select the "Rotary axis A, B, C" entry.

9. Enter the setpoint angle between the workpiece edge and the reference axis.
10. Traverse the tool to the workpiece edge.

Save P1

11. Press the "Save P1" softkey.

Save P2

12. Reposition the tool and repeat the measuring procedure (steps 6 to 11) to measure the second point, and then press the "Save P2" softkey.

Set WO -or- Calculate

13. Press the "Set WO" or "Calculate" softkey.

The angle between the workpiece edge and reference axis is calculated and displayed.
With "Set WO", the workpiece edge now corresponds to the setpoint angle. The calculated rotation is stored in the work offset.

Aligning an edge automatically

1. Attach a 3D probe type tool to the spindle.
2. Prepare the measurement (as described under "Aligning the edge manually", steps 2 to 9).
3. Move the tool up close to the workpiece edge along which you want to measure.

Cycle Start

4. Press the "Cycle Start" key.

This starts the automatic measuring process. The position of measuring point 1 is measured and stored.
The "P1 stored" softkey becomes active".

5. Repeat the measurement procedure (steps 3 to 4) to measure the second point.

The position of measuring point 2 is measured and stored.
The "P2 stored" softkey becomes active".

Set WO -or- Calculate

6. Press the "Set WO" or "Calculate" softkey.

The angle between the workpiece edge and reference axis is calculated and displayed.
With "Set WO", the workpiece edge now corresponds to the setpoint angle. The calculated rotation is stored in the work offset.

Measuring the distance between two edges manually

1. Attach any tool for scratching to the spindle.

Jog Meas. workp.

2. In "Machine Manual" mode, select the "Meas. workp." softkey.
3. Press the "Edge" softkey.
The "Edge" window opens with new vertical softkeys.
4. Press the "Distance between 2 edges" softkey.
5. Specify whether you want "Measurement only" or in which work offset you want to store the zero point (as described under "Setting the edge manually", step 5).

Alternat.

6. In "Measuring direction P1", select the measuring direction (+ or -) and the measuring axis in which you first want to approach the workpiece.
7. In "Measuring direction P2" select the measuring direction (+ or -) for the 2nd measuring point.
The axis selected in "Measuring direction P1" is displayed.
8. Specify the setpoint position of the center line between the two workpiece edges.
9. Traverse the tool to the first measuring point.

Save P1

10. Press the "Save P1" softkey.

Save P2

11. Reposition the tool, approach the second measuring point and store the second point.

Set WO -or- Calculate

12. Press the "Set WO" or "Calculate" softkey.

The distance between the two workpiece edges and the center line are calculated and displayed.
With "Set WO", the center line now corresponds to the position setpoint.
The calculated offset is stored in the work offset.

Measuring the distance between two edges automatically

1. Attach a 3D probe type tool to the spindle.
2. Prepare the measurement (as described under "Measuring the distance between two edges manually", steps 2 to 8).
3. Move the tool up close to the workpiece edge along which you want to measure.

Cycle Start

4. Press the "Cycle Start" key.

This starts the automatic measuring process. The position of measuring point 1 is measured and stored. The "P1 stored" softkey becomes active".

5. Repeat the measurement procedure (steps 3 to 4) to measure the second point.

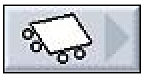
The position of measuring point 2 is measured and stored. The "P2 stored" softkey becomes active".

Set WO -or- Calculate

6. Press the "Set WO" or "Calculate" softkey.

The distance between the two workpiece edges and the center line are calculated and displayed.
With "Set WO", the center line now corresponds to the position setpoint.
The calculated offset is stored in the work offset.

2.6.2 Measuring a corner

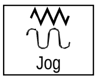


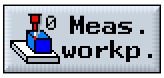
You can measure workpieces with a 90° angle or with any other angle.

- Measuring a right-angled corner
  The workpiece has a 90° corner and is in any orientation on the work table. By measuring three points you can determine the corner point in the working plane (X/Y plane) and angle α between the reference edge on the workpiece (line through P1 and P2) and the reference axis (always the 1st axis in the working plane).
- Measuring any corner
  The workpiece has any corner (not right-angled) and is in any orientation on the work table. By measuring four points you can determine the corner point in the working plane (X/Y plane), angle α between the reference edge on the workpiece (line through P1 and P2) and the reference axis (always the 1st axis in the working plane), and angle β in the corner.

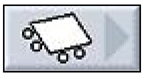
Measuring a right-angled/any corner manually

1. Attach any tool for scratching to the spindle.





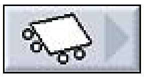
2. In "Machine Manual" mode, select the "Meas. workp." softkey.



3. Press the "Corner" softkey.
The "Corner" window opens with new vertical softkeys.

4. Press the "right-angled corner" softkey, if you want to measure a right-angled corner.

- OR -



➢ Press the "any corner" softkey, if you want to measure a corner not equal to 90°.

5. Specify whether you want "Measurement only" or in which work offset you want to store the zero point (as described under "Measuring an edge manually", step 5).

6. Under "Corner", select the type of corner you want to measure (e.g. outside corner) and its position (e.g. position 1).
7. Specify the setpoint of the workpiece corner (X0, Y0) you want to measure.
8. Traverse the tool to the first measuring point P1 as shown in the help display.

Save P1

9. Press the "Save P1" softkey.

The coordinates of the first measuring point are measured and stored.

Save P2 Save P3

10. Reposition the spindle holding the tool each time, approach measuring points P2 and P3 and press the "Save P2" and "Save P3" softkeys.

Save P4

11. Repeat the procedure to measure the fourth measuring point when you measure "any corner".

Set WO -or- Calculate

12. Press the "Set WO" or "Calculate" softkey.

The corner point and angles α and β are calculated and displayed. With "Set WO", the corner point now corresponds to the position setpoint. The calculated offset is stored in the work offset.

Measuring a right-angled/any corner automatically

1. Attach a 3D probe type tool to the spindle.
2. Prepare the measurement (as described under "Measuring a right-angled/any corner manually", steps 2 to 7).
3. Move the tool up close to measuring point P1.
4. Press the "Cycle Start" key.

Cycle Start

This starts the automatic measuring process. The position of measuring point 1 is measured and stored. The "P1 stored" softkey becomes active".

5. Repeat the measurement procedure (steps 3 to 4) to measure points P2 and P3.
6. If you are measuring a corner not equal to 90°, repeat the procedure to measure and store point P4.
7. Press the "Set WO" or "Calculate" softkey.

Set WO -or- Calculate

The corner point and angles α and α are calculated and displayed. With "Set WO", the corner point now corresponds to the position setpoint. The calculated offset is stored in the work offset.

2.6.3 Measuring a pocket and hole

You can measure rectangular pockets and one or more holes and then align the workpiece.

- Measuring a rectangular pocket
  The rectangular pocket must be aligned at right-angles to the coordinate system. By measuring four points inside the pocket you can determine the length, width, and center point of the pocket.
- Measuring 1 hole
  The workpiece is in any orientation on the work table and has one hole. You can determine the diameter and center point of the hole with four measuring points.
- Measuring 2 holes
  The workpiece is in any orientation on the work table and has two holes. 4 points are automatically measured in both holes and the hole centers are calculated from them. Angle α is calculated from the connecting line between both center points and the reference axis, and the new zero point that corresponds to the center point of the 1st hole is determined.
- Measuring 3 holes
  The workpiece is in any orientation on the work table and has three holes. 4 points are automatically measured in the three holes and the hole centers are calculated from them. A circle is placed through the three center points. The circle center point and diameter are determined from it. If an angle offset is selected, base angle of rotation α can also be found.
- Measuring 4 holes
  The workpiece is in any orientation on the work table and has four holes. 4 points are automatically measured in the four holes and the hole centers are calculated from them. Two hole center points are diagonally connected in each case. The point of intersection between the two lines is determined from this. If an angle offset is selected, base angle of rotation α can also be found.

You can only measure 2, 3, and 4 holes automatically.

Measuring a rectangular pocket manually

Jog

Meas. workp.

1. Attach any tool for scratching to the spindle.
2. In "Machine Manual" mode, select the "Meas. workp." softkey.
3. Press the "Pocket/Hole" softkey.
The "Pocket/Hole" window opens with new vertical softkeys.
4. Press the "Rectangular Pocket" softkey.
5. Specify whether you want "Measurement only" or in which work offset you want to store the zero point (as described under "Measuring an edge manually", step 5).
6. Specify the position setpoints (X0/Y0) of the pocket center point.
7. Traverse the tool to the first measuring point.

Save P1

8. Press the "Save P1" softkey.
The point is measured and stored.

Save P2 ... Save P4

9. Repeat steps 8 and 9 to measure and store measuring points P2, P3, and P4.

Set WO -or- Calculate

10. Press the "Set WO" or "Calculate" softkey.

The length, width, and center point of the rectangular pocket are calculated and displayed.
The set position of the center point is stored as a new zero point with "Set WO". The tool radius is automatically included in the calculation.

Measuring a rectangular pocket automatically

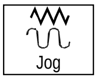
1. Attach a 3D probe type tool to the spindle.
2. Move the tool until it is approximately at the center of the pocket.
3. Prepare the measurement (as described under "Measuring a rectangular pocket manually", steps 2 to 6).
4. In field "L" enter the length (1st axis of the working plane) and in "W" (2nd axis of the working plane) enter the width of the pocket, if the measuring stroke would not reach the edges.

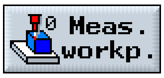
Cycle Start

5. Press the "Cycle Start" key.
This starts the automatic measuring process.

The length, width, and center point of the rectangular pocket are calculated and displayed.
The set position of the center point is stored as the new zero point if you have selected "work offset". The tool radius is automatically included in the calculation.

Measuring a hole manually





1. Attach any tool for scratching to the spindle.
2. In "Machine Manual" mode, select the "Meas. workp." softkey.

3. Press the "Pocket/Hole" softkey.

The "Pocket/Hole" window opens with new vertical softkeys.

4. Press the "1st hole" softkey.
5. Specify whether you want "Measurement only" or in which work offset you want to store the zero point (as described under "Measuring a rectangular pocket manually", step 5).
6. Specify the position setpoints (X0, Y0) of the hole center point.
7. Traverse the tool to the first measuring point.

Save P1

8. Press the "Save P1" softkey.

The point is measured and stored.

Save P2 ... Save P4

9. Repeat steps 8 and 9 to measure and store measuring points P2, P3, and P4.

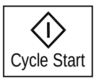
Set WO -or- Calculate

10. Press the "Set WO" or "Calculate" softkey.

The diameter and center point of the hole are calculated and displayed.
The set position of the center point is stored as a new zero point with "Set WO". The tool radius is automatically included in the calculation.

Measuring a hole automatically

1. Attach a 3D probe type tool to the spindle.
2. Move the tool until it is positioned approximately at the center of the hole.
3. Prepare the measurement (as described under "Measuring a hole manually", steps 2 to 6).
4. Enter a "Øhole" and the approximate diameter.
   This limits the area for rapid traverse. If no diameter is entered, travel starts from the starting point at measurement feedrate.
5. Enter an angle under "Probe angle".
   With the probe angle you can turn the travel direction of the probe any angle.



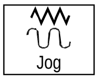
6. Press the "Cycle Start" key.

The tool automatically contacts 4 points in succession around the inside wall of the hole. When measurement has been successfully completed, the "P0 stored" softkey becomes active.

The diameter and center point of the hole are calculated and displayed.
The set position of the center point is stored as the new zero point if you have selected "work offset". The tool radius is automatically included in the calculation.

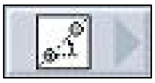
Measuring two holes automatically

1. Attach a 3D probe type tool to the spindle.
2. Move the tool until it is positioned approximately at the center of the first hole.



3. In "Machine Manual" mode, select the "Meas. workp." softkey.

4. Press the "Pocket/Hole" softkey.
The "Holes" window opens with new vertical softkeys.



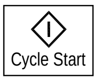
5. Press the "2 holes" softkey.
6. Specify whether you want "Measurement only" or in which work offset you want to store the zero point (as described under "Measuring an edge manually", step 5).
7. Enter the approximate diameter in "Øhole" (see step 4, "Measuring two holes manually".

8. Under "Angle offs.", select the "Coor. Rotation" entry.

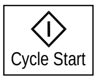
- OR -

➢ Under "Angle offs.", select the "Rotary axis A, B, C" entry.

9. Enter the setpoint angle.
10. Specify the position setpoints (X1/Y1) for the center point of the first hole.
X1 and Y1 are only active, if the "Coor. Rotation" entry is selected.



11. Press the "Cycle Start" key.
The tool automatically contacts 4 points in succession around the first inside wall of the hole. When measurement has been successfully completed, the "P1 stored" softkey becomes active.



12. Then move the tool approximately to the center of the hole and press the "Cycle Start" button.
The tool automatically contacts 4 points in succession around the second inside wall of the hole. When measurement has been successfully completed, the "P2 stored" softkey becomes active.

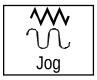
Set WO -or- Calculate

13. Press the "Set WO" or "Calculate" softkey.

The angle between the line connecting the center points and the reference axis is calculated and displayed.
With "Set WO", the center point of the first hole now corresponds to the position setpoint. The calculated rotation is stored in the work offset.

Measuring three holes automatically

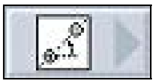
1. Attach a 3D probe type tool to the spindle.



2. Move the tool until it is approximately at the center of first the hole.
3. In "Machine Manual" mode, select the "Meas. workp." softkey.

4. Press the "Pocket/Hole" softkey.

The "Pocket/Hole" window opens with new vertical softkeys.



➢ Press the "3 holes" softkey.

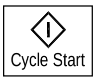
5. Specify whether you want "Measurement only" or in which work offset you want to store the zero point (as described under "Measuring an edge manually", step 5).
6. Enter the approximate diameter in "Øhole" (see step 4, "Measuring two holes manually").

7. Under "Angle offs.", select entry "No".

- OR -

➢ Under "Angle offs." select entry "Yes", if you want alignment to be performed with coordinate rotation.

8. Enter the setpoint angle.
   The angle entered here refers to the 1st axis of the working plane (X/Y plane). This input field only appears if you specified "Yes" for "Angle offs."
9. Specify setpoint positions X0 and Y0.
   These determine the center point of the circle on which the center points of the three holes are to lie.



10. Press the "Cycle Start" key.

The tool automatically contacts 4 points in succession around the first inside wall of the hole. When measurement has been successfully completed, the "P1 stored" softkey becomes active.

Cycle Start

11.Then move the tool approximately to the center of the second and third hole and press the "Cycle Start" key.
The tool automatically contacts 4 points in succession around the inside walls of the holes. When measurement of P2 and P3 has been successfully completed, the "P2 stored" and "P3 stored" softkeys become active.

Set WO -or- Calculate

12.Press the "Set WO" or "Calculate" softkey.

The center point and the diameter of the circle on which the three hole center points lie are calculated and displayed. If you selected entry "Yes" for "Angle offs.", angle α is additionally calculated and displayed.
With "Set WO", the center point of the first hole now corresponds to the position setpoint. The calculated rotation is stored in the work offset.

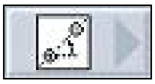
Measuring four holes automatically

1. Attach a 3D probe type tool to the spindle.
2. Move the tool until it is approximately at the center of first the hole.

Jog Meas. workp.

3. In "Machine Manual" mode, select the "Meas. workp." softkey.

4. Press the "Pocket/Hole" softkey.
The "Pocket/Hole" window opens with new vertical softkeys.



- Press the "4 holes" softkey.

5. Specify whether you want "Measurement only" or in which work offset you want to store the zero point (as described under "Measuring an edge manually", step 5).
6. Enter the approximate diameter in "Øhole" (see step 4, "Measuring two holes manually".

Alternat.

7. Under "Angle offs.", select entry "No".

- OR -

Alternat.

- Under "Angle offs." select entry "Yes", if you want alignment to be performed with coordinate rotation.

8. Enter the setpoint angle.
The angle entered here refers to the 1st axis of the working plane (X/Y plane). This input field only appears, if you specified "Yes" for "Angle offs."

Cycle Start

9. Specify setpoint positions X0 and Y0.
   These determine the point of intersection of the lines connecting the hole center points.
10. Press the "Cycle Start" key.

The tool automatically contacts 4 points in succession around the first inside wall of the hole. When measurement has been successfully completed, the "P1 stored" softkey becomes active.

Cycle Start

11. Then move the tool approximately to the center of the second, third, and fourth hole and press the "Cycle Start" key.

The tool automatically contacts 4 points in succession around the inside walls of the holes. When measurement of P2, P3, and P4 has been successfully completed, the "P2 stored", "P3 stored", and "P4 stored" softkeys become active.

Set WO -or- Calculate

12. Press the "Set WO" or "Calculate" softkey.

The hole center points are connected diagonally and the intersection point of the two connecting lines calculated and displayed If you selected entry "Yes" for "Angle offs.", angle α is additionally calculated and displayed.
With "Set WO", the intersection point now corresponds to the position setpoint. The calculated rotation is stored in the work offset.

2.6.4 Measuring a spigot

You can measure and align rectangular spigots, and one or more circular spigots.

- Measuring a rectangular spigot
  The rectangular spigot must be aligned at right-angles to the coordinate system. By measuring four points inside the spigot you can determine the length, width, and center point of the spigot.
- Measuring 1 circular spigot
  The workpiece is in any orientation on the work table and has one spigot. You can determine the diameter and center point of the spigot with four measuring points.
- Measuring 2 circular spigot
  The workpiece is in any orientation on the work table and has two spigots. 4 points are automatically measured at the two spigots and the spigot centers are calculated from them. Angle α is calculated from the connecting line between both center points and the reference axis, and the new zero point that corresponds to the center point of the first spigot is determined.

- Measuring 3 circular spigots
  The workpiece is in any orientation on the work table and has three spigots. 4 points are automatically measured at the three spigots and the spigot centers are calculated from them. A circle is placed through the three center points and the circle center and circle diameter are determined.
  If an angle offset is selected, base angle of rotation α can also be found.
- Measuring 4 circular spigot
  The workpiece is in any orientation on the work table and has two spigots. 4 points are automatically measured at the four spigots and the spigot centers are calculated from them. Two spigot center points are each connected diagonally and the intersection point of the two lines is then determined. If an angle offset is selected, base angle of rotation α can also be found.

You can only measure 2, 3, and 4 circular spigots automatically.

Measuring a rectangular spigot manually

1. Attach any tool for scratching to the spindle.

Jog | Meas. workp.

2. In "Machine Manual" mode, select the "Meas. workp." softkey.
3. Press the "Spigot" softkey.
4. Press the "Rectangular spigot" softkey.
5. Specify whether you want "Measurement only" or in which work offset you want to store the zero point (as described under "Measuring an edge manually", step 5).
6. Specify the position setpoints (X0/Y0) of the spigot center point P0.
7. Traverse the tool to the first measuring point.

Save P1

8. Press the "Save P1" softkey.

The point is measured and stored.

Save P2 ... Save P4

9. Repeat steps 7 and 8 to measure and store measuring points P2, P3, and P4.

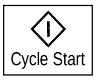
Set WO -or- Calculate

10. Press the "Set WO" or "Calculate" softkey.

The length, width, and center point of the rectangular spigot are calculated and displayed.
The set position of the center point is stored as a new zero point with "Set WO". The tool radius is automatically included in the calculation.

Measuring a rectangular spigot automatically

1. Attach a 3D probe type tool to the spindle.
2. Move the tool until it is approximately at the center of the spigot.
3. Prepare the measurement (as described under "Measuring a rectangular spigot manually", steps 2 to 6).
4. Enter the infeed value in "DZ" to determine the measuring depth.
5. In field "L" enter the length (1st axis of the working plane) and in "W" (2nd axis of the working plane) enter the width of the spigot, if the measuring stroke would not reach the edges.

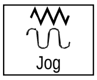


6. Press the "Cycle Start" key.

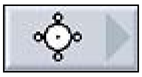
The tool automatically contacts 4 points in succession around the outside wall of the spigot.
The length, width, and center point of the rectangular spigot are calculated and displayed.
The set position of the center point is stored as the new zero point if you have selected "work offset". The tool radius is automatically included in the calculation.

Measuring a circular spigot manually

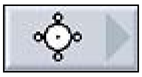
1. Attach any tool for scratching to the spindle.



2. In "Machine Manual" mode, select the "Meas. workp." softkey.



3. Press the "Spigot" softkey.



4. Press the "1 circular spigot" softkey.
5. Specify whether you want "Measurement only" or in which work offset you want to store the zero point (as described under "Measuring an edge manually", step 5).
6. Enter the infeed value in "DZ" to determine the measuring depth.
7. Specify the position setpoints (X0 and Y0) of the spigot center point P0.
8. Traverse the tool to the first measuring point on the spigot outside wall.

Save P1

9. Press the "Save P1" softkey.

Save P2 ... Save P4

10. Repeat steps 8 and 9 to measure and store measuring points P2, P3, and P4.

Set WO -or- Calculate

11.Press the "Set WO" or "Calculate" softkey.

The diameter and center point of the spigot are calculated and displayed.
The set position of the center point is stored as a new zero point with "Set WO". The tool radius is automatically included in the calculation.

Measuring a circular spigot automatically

1. Attach a 3D probe type tool to the spindle.
2. Move the tool until it is approximately at the center of the spigot.
3. Prepare the measurement (as described under "Measuring a circular spigot manually", steps 2 to 7).
4. In "Øspigot", enter the approximate diameter of the spigot.
   This limits the area for rapid traverse. If no diameter is entered, Travel starts from the starting point at measurement feedrate.
5. Enter an angle in "Probing angle" (see step 5, "Measuring one hole automatically").

Cycle Start

6. Press the "Cycle Start" key.

The tool automatically contacts 4 points in succession around the outside wall of the spigot. When measurement has been successfully completed, the "P0 stored" softkey becomes active.
The diameter and center point of the spigot are calculated and displayed.
The set position of the center point is stored as the new zero point if you have selected "work offset". The tool radius is automatically included in the calculation.

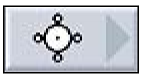
Measuring two circular spigots automatically

1. Attach a 3D probe type tool to the spindle.
2. Move the tool until it is approximately at the center of the first spigot.

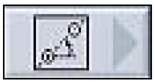
Jog

Meas. workp.

3. In "Machine Manual" mode, select the "Meas. workp." softkey.



4. Press the "Spigot" softkey.



5. Press the "2 circular spigots" softkey.
6. Specify whether you want "Measurement only" or in which work offset you want to store the zero point (as described under "Measuring an edge manually", step 5).

Alternat.

Alternat.

Cycle Start

Cycle Start

Set WO -or- Calculate

7. Enter the approximate diameter of the spigot in "Øspigot" (see step 4, "Measuring one spigot automatically").
8. Enter the infeed value in "DZ" to determine the measuring depth.
9. Under "Angle offs.", select the "Coor. Rotation" entry.

- OR -

➢ Under "Angle offs.", select entry "Rotary axis A, B, C".

10. Enter the setpoint angle.
    The angle entered here refers to the 1st axis of the working plane (X/Y plane).
11. Specify the position setpoints (X1/Y1) for the center point of the first hole.
    The input fields are only active if you have selected the angle offset via coordinate rotation.
12. Press the "Cycle Start" key.

The tool automatically contacts 4 points in succession around the first outside wall of the spigot. When measurement has been successfully completed, the center of the spigot is determined and the "P1 stored" softkey becomes active.

13. Then move the tool approximately to the center of the second spigot and press the "Cycle Start" key.

The tool automatically contacts 4 points in succession around the second outside wall of the spigot. When measurement has been successfully completed, the "P2 stored" softkey becomes active.

14. Press the "Set WO" or "Calculate" softkey.

The angle between the line connecting the center points and the reference axis is calculated and displayed.
With "Set WO", the center point of the first spigot now corresponds to the position setpoint. The calculated rotation is stored in the work offset.

Measuring two circular spigots automatically

Jog Meas. workp.

1. Attach a 3D probe type tool to the spindle.
2. Move the tool until it is approximately at the center of the first spigot.
3. In "Machine Manual" mode, select the "Meas. workp." softkey.
4. Press the "Spigot" softkey.

➢ Press the "3 circular spigots" softkey.

5. Specify whether you want "Measurement only" or in which work offset you want to store the zero point (as described under "Measuring an edge manually", step 5).
6. Enter the approximate diameter of the spigot in "Øspigot" (see step 4, "Measuring one spigot automatically").
7. Enter the infeed value in "DZ" to determine the measuring depth.

Alternat.

8. Under "Angle offs." select entry "No".

- OR -

Alternat.

➢ Under "Angle offs." select entry "Yes" if you want alignment to be performed with coordinate rotation.

9. Enter the setpoint angle.
The angle entered here refers to the 1st axis of the working plane (X/Y plane). This input field only appears if you specified "Yes" for "Angle offs."

10. Specify setpoint positions X0 and Y0.
These determine the center point of the circle on which the center points of the three spigots are to lie.

Cycle Start

11. Press the "Cycle Start" key.

The tool automatically contacts 4 points in succession around the first outside wall of the spigot. When measurement has been successfully completed, the center of the spigot is determined and the "P1 stored" softkey becomes active.

Cycle Start

12. Then move the tool approximately to the center of the second and third spigot and press the "Cycle Start" button.

The tool automatically contacts 4 points in succession around the spigot outside walls. After successful completion of measurement, measured values P2 and P3 are stored and the softkeys "P2 stored" and "P3 stored" become active.

Set WO -or- Calculate

13. Press the "Set WO" or "Calculate" softkey.

The center point and the diameter of the circle on which the three spigot center points lie are calculated and displayed. If you have selected "Yes" in "Coor. rot.", angle α is additionally calculated and displayed.
With "Set WO", the center point of the circle now corresponds to the position setpoint. The calculated rotation is stored in the work offset.

Measuring four circular spigots automatically

1. Attach a 3D probe type tool to the spindle.
2. Move the tool until it is approximately at the center of the first spigot.
3. In "Machine Manual" mode, select the "Meas. workp." softkey.
4. Press the "Spigot" softkey.

➢ Press the "4 circular spigots" softkey.

5. Specify whether you want "Measurement only" or in which work offset you want to store the zero point (as described under "Measuring an edge manually", step 5).
6. Enter the approximate diameter of the spigot in "Øspigot" (see step 4, "Measuring one circular spigot manually").
7. Enter the infeed value in "DZ" to determine the measuring depth.
8. Under "Angle offs." select entry "Yes" if you want alignment to be performed with coordinate rotation.

- OR -

➢ Under "Angle offs." select entry "No".

9. Enter the setpoint angle.
   The angle entered here refers to the 1st axis of the working plane (X/Y plane). This input field only appears if you specified "Yes" for "Angle offs."
10. Specify setpoint positions X0 and Y0.
    These determine the point of intersection of the lines connecting the spigot center points.
11. Press the "Cycle Start" key.

The tool automatically contacts 4 points in succession around the first outside wall of the spigot. When measurement has been successfully completed, the center of the spigot is determined and the "P1 stored" softkey becomes active.

12. Then move the tool approximately to the center of the second, third, and fourth spigot and press the "Cycle Start" button.

The tool automatically contacts 4 points in succession around the spigot outside walls. After successful completion of measurement, measured values P2, P3, and P4 are stored and the softkeys "P2 stored", "P3 stored", and "P4 stored" become active.

Set WO -or- Calculate

13.Press the "Set WO" or "Calculate" softkey.

The spigot center points are connected diagonally and the intersection point of the two connecting lines calculated and displayed If you have selected "Yes" in "Coor. rot.", angle α is additionally calculated and displayed.
With "Set WO", the intersection point now corresponds to the position setpoint. The calculated rotation is stored in the work offset.

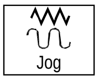
2.6.5 Aligning the plane

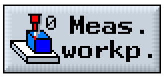
You can measure an inclined plane of a workpiece in space and determine rotations α and β. By subsequently performing coordinate rotation, you can align the tool axis perpendicular to the workpiece plane.
In order to determine the position of the plane in space, three different points are measured along the tool axis.
To align the tool axis in the perpendiculary you require a swiveling table or swivel head.

In order to be able to measure the plane, the surface must be flat.

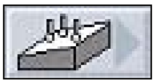
Aligning a plane manually

1. Attach any tool for scratching to the spindle.





2. In "Machine Manual" mode, select the "Meas. workp." softkey.



3. Press the "Align plane" softkey.
4. Specify whether you want "Measurement only" or in which work offset you want to store the zero point (as described under "Measuring an edge manually", step 5).
5. Traverse the tool to the first measuring point that you want to determine.

Save P1

6. Press the "Save P1" softkey.

Save P2 Save P3

7. Then move the tool to the second and third measuring point and press the "Save P2" and "Save P3" softkeys.

Set WO -or- Calculate

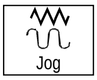
8. Press the "Set WO" or "Calculate" softkey.

Angles α and β are calculated and displayed.
With "Set WO" the angle offset is stored in the work offset memory.

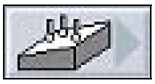
Aligning a plane automatically

1. Attach a 3D probe type tool to the spindle.

2. Traverse the tool to near the point you want to determine first.

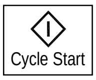


3. In "Machine Manual" mode, select the "Meas. workp." softkey.



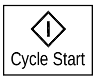
4. Press the "Align plane" softkey.

5. Specify whether you want "Measurement only" or in which work offset you want to store the zero point (as described under "Measuring an edge manually", step 5).



6. Press the "Cycle Start" key.

When measurement has been successfully completed, the measured value is stored and the "P1 stored" softkey becomes active.



7. Then move the tool so that it is approximately above the second and then the third measuring point and press the "Cycle Start" button.

When measurement is complete, the "P2 stored" and "P3 stored" softkeys becomes active.

Set WO -or- Calculate

8. Press the "Set WO" or "Calculate" softkey.

Angles α and β are calculated and displayed.
With "Set WO" the angle offset is stored in the work offset memory.

2.6.6 Corrections after measurement of the zero point

If you store the workpiece zero in a work offset, changes to the coordinate system or axis positions might be necessary in the following cases.

- Correcting the work offset causes the workpiece coordinate system to rotate, after which the tool can be aligned perpendicularly to the plane.
- Correcting the work offset necessitates positioning of the rotary axis in order to align the workpiece parallel with the coordinate system

Activation windows help you to adapt the coordinate system and the axis positions.

Activating work offset

You stored the workpiece zero in a work offset that was not active during measurement.

Set WO

When you press the "Set WO" softkey, the activation window opens asking whether you want to "Activate work offset xxx now?".

OK

➢ Press the "OK" softkey to activate the corrected work offset.

Aligning and retracting the tool

Rotating the workpiece coordinate system makes it necessary to realign the tool to the plane.
The activation window asking whether you want to "Position measuring probe perpendicular to plane?" is displayed.

Alternat.

➢ Select "Yes" if you want to swivel into the plane.

The query "Positioning by swiveling! Retract?" appears.

Alternat.

➢ Select the retract method you want to use.

Cycle Start

➢ Press the "Cycle Start" key.

When the axis has been retracted the tool is realigned with the help of the swivel cycle.
You can now measure again.

Positioning a rotary axis and entering a feedrate

Once you have measured the workpiece zero you must reposition the rotary axis.
The activation window asking whether you want to "Position rotary axis X to align?" is displayed.

Alternat.

➢ Select "Yes" if you want to position the rotary axis.

An input field for the feedrate and the softkey "Rapid traverse" are displayed.

Rapid traverse

➢ Press the "Rapid traverse" softkey to enter the feedrate in rapid traverse.

- OR -

➢ Enter the desired offset in input field "F".

Cycle Start

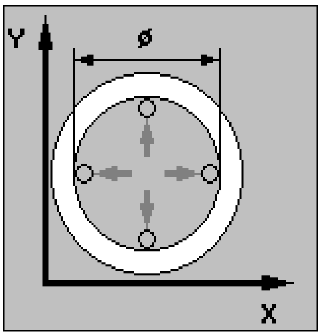
➢ Press the "Cycle Start" key.

The rotary axis is repositioned.

2.6.7 Calibrating an electronic measuring tool

When the electronic measuring tools are attached to the spindle, clamping tolerances often occur. This can lead to measurement errors.
In addition, you need to determine the trigger point of the measuring tool relative to the spindle center (trigger point).
Therefore, you need to calibrate the electronic measuring tool. The radius is calibrated in a hole, the length is calibrated on a surface. For the hole, you can use a hole in the workpiece or use a ring gauge.
The radius of the measuring tool must be contained in the tool list.

Calibrating a radius



1. Attach a 3D probe type tool to the spindle.
2. Move the tool into the hole and position it in the approximate center of the hole.

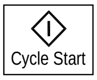
Jog | Meas. workp.

3. In "Machine Manual" mode, select the "Meas. workp." softkey.

Calibration probe | Radius

4. Press the "Calibration probe" and "Radius" softkeys.

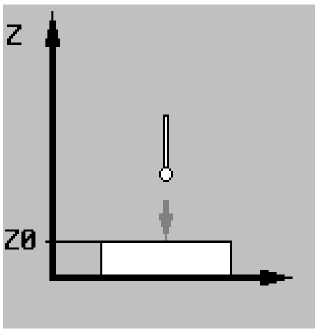
5. Enter the diameter of the hole



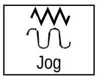
6. Press the "Cycle Start" key.

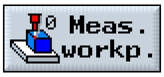
Calibration starts. First the exact hole center point is determined. Then the 4 trigger points on the inside wall of the hole are approached.

Calibrating a length



1. Attach a 3D probe type tool to the spindle.
2. Position the tool above the surface.

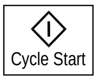




3. In "Machine Manual" mode, select the "Meas. workp." softkey.

Calibration probe | Length

4. Press the "Calibration probe" and "Length" softkeys.
5. Specify reference point Z0 of the surface, e.g. of the workpiece or the machine table.



6. Press the "Cycle Start" key.

Calibration starts. The length of the measuring tool is calculated and entered in the tool list.

2.7 Measuring a tool

The various tool geometry parameters must be referenced while the program is running. These are stored as so-called tool offset data in the tool list. Each time the tool is called, the control considers the tool offset data.
You can determine the tool offset data, i.e. the length and radius or diameter, either manually or automatically (per measuring probe).

2.7.1 Measuring a tool manually

For manual measurement, move the tool manually to a known reference point to determine the tool length and the radius and diameter. ShopMill then calculates the tool offsets from the position of the toolholder reference point and the reference point.

When measuring the tool length you can either use the workpiece or a fixed point in the machine coordinate system, e.g. a mechanical test socket or a fixed point in combination with a distance gauge as the reference point.

You can enter the position of the workpiece during measurement. The position of the fixed point on the other hand must be specified before you start measurement (see Section "Adjusting the fixed point").

When determining the radius/diameter, the workpiece is always the reference point.
Depending on the setting in a machine data, you can measure the radius or the diameter of the tool.

Please refer to the machine manufacturer's instructions.

Measuring length Workpiece reference point

- Attach the tool you want to measure to the spindle.

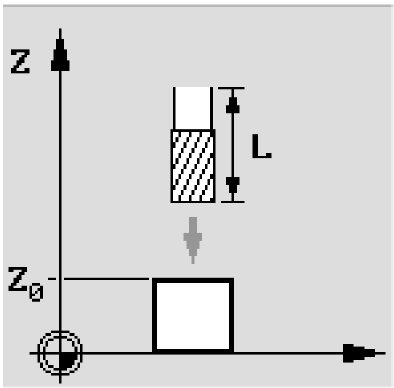
Jog | Meas. tool

- Select the "Measure tool" softkey in "Machine Manual" mode.

Length manual >

- Press the "Length manual" softkey.
- Select the tool cutting edge D and the duplo number DP for the tool.

➢ Select the "Workpiece" reference point.



Measuring the tool length on the edge of the workpiece

➢ Approach the workpiece in the Z direction and perform scratching with a rotating spindle (see Sec. "Traversing the Machine Axes").

➢ Specify the setpoint position Z0 of the workpiece edge.

Set length

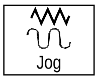
➢ Press the "Set length" softkey.

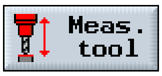
The tool length is calculated automatically and entered in the tool list.

If you want to determine the tool length not with a workpiece, but with a test socket instead, no work offset may be selected, or the basic work offset must be zero.

Measuring length Fixed point reference point

➢ Attach the tool you want to measure to the spindle.





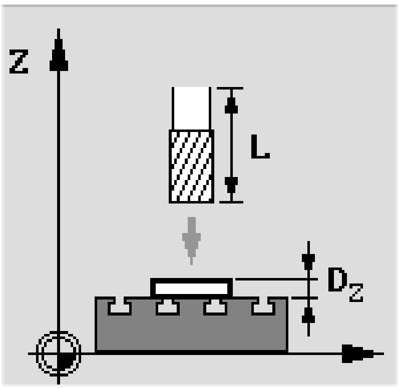
➢ Select the "Measure tool" softkey in "Machine Manual" mode.

Length manual >

➢ Press the "Length manual" softkey.

➢ Select the tool cutting edge D and the duplo number DP for the tool.

➢ Select the "fixed point" reference point.



Measuring the tool length on the measuring edge

➢ If you are measuring with a test socket, enter 0 for offset value "DZ" and approach the fixed point in the Z direction (see Sec."Traversing the machine axes").

Approach is performed with a rotating spindle in the opposite direction of rotation. The test socket automatically displays a reading when the precise position is reached.
- OR -

➢ If you are using a distance gauge, travel as close to the fixed point as possible, measure the gap with the distance gauge and enter the value in "DZ".

Approach to the distance gauge is performed with a stationary spindle.

Set length

➢ Press the "Set length" softkey.

The tool length is calculated automatically and entered in the tool list.

Measuring a radius/diameter

➢ Attach the tool you want to measure to the spindle.

Jog | Meas. tool

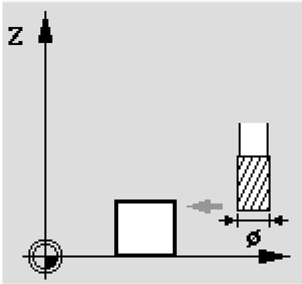
➢ Select the "Measure tool" softkey in "Machine Manual" mode.

Radius manual > or Dia. manual >

➢ Press the "Radius manual" or "Dia. manual" softkey.

➢ Select the tool cutting edge D and the duplo number DP for the tool.

➢ Approach the workpiece in the X or Y direction and perform scratching with the spindle rotating in the opposite direction (see Sec. "Traversing the machine axes").



Measuring radius/diameter

➢ Specify the setpoint position X0 or Y0 of the workpiece edge.

Set radius or Set diameter

➢ Press the "Set radius" or "Set diameter" softkey.

The tool radius or diameter is calculated automatically and entered in the tool list.

2.7.2 Calibrating a fixed point

If you want to use a fixed point as the reference point in manual measurement of the tool length, you must first determine the position of the fixed point relative to the machine zero.

You can use a mechanical test socket as the fixed point, for example. Mount the test socket on the machine table in the machining space of the machine. Enter zero as the distance.

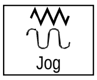
But you can also use any fixed point on the machine in combination with a distance gauge. Enter the thickness of the plate in "DZ".

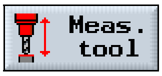
To calibrate the fixed point either use a tool of known length (i.e. the tool length must have been entered in the tool list) or the spindle directly.

The position of the fixed point may have already been determined by the machine manufacturer.

Please refer to the machine manufacturer's instructions.

- Traverse the tool or spindle to the fixed point.





- Select the "Measure tool" softkey in "Machine Manual" mode.

Calibrate fixed point

- Press the "Calibrate fixed point" softkey.
- Enter an offset value for "DZ".

If you have used a distance gauge, enter the thickness of the plate used.

Calibrate

- Press the "Calibrate" softkey.

The distance dimensions between machine zero and fixed point is calculated and entered in the machine data.

2.7.3 Measuring a tool with measuring probe

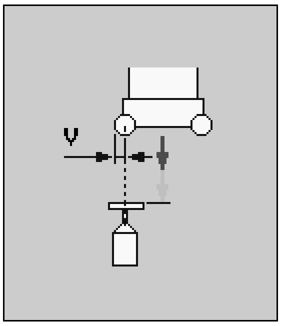
For automatic measurement, you determine the length and radius or diameter of the tool with the aid of a measuring probe (table probe system). ShopMill uses the known positions of the tool carrier reference point and measuring probe to calculate the tool offset data.

Before you measure a tool automatically, you must enter the approximate tool geometry data (length and radius or diameter) in the tool list and calibrate the probe.

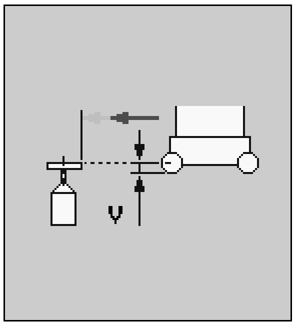
Depending on the setting in a machine data, you can measure the radius or the diameter of the tool.

Please refer to the machine manufacturer's instructions.

You can consider a lateral or longitudinal offset V when measuring. If the maximum length of the tool is not at the outer edge of the tool or the maximum width is not at bottom edge of the tool, you can store this difference in the offset.



Lateral offset

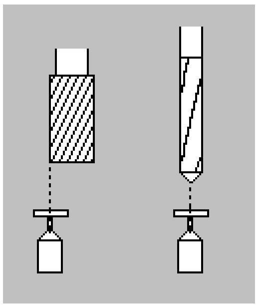


Longitudinal offset

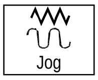
If measuring shows that the length of the tool diameter is greater than the probe diameter, measurement is automatically performed with a turning spindle rotating in the opposite direction. The tool is not traversed across the center of the probe but with the outside edge of the tool along the center point of the measuring probe.

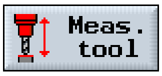
Measuring length

- Attach the tool you want to measure to the spindle.
- Position the tool near the measuring probe so that it can be approached without collision.



Measuring tool length

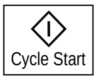




- Select the "Measure tool" softkey in "Machine Manual" mode.

Length Auto >

- Press the "Length Auto" softkey.
- Select the tool cutting edge D and the duplo number DP for the tool.
- If necessary, enter the lateral offset V.



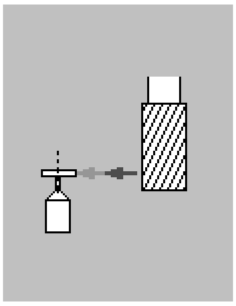
- Press the "Cycle Start" key.

This starts the automatic measuring process. The tool length is calculated automatically and entered in the tool list.
The measuring process depends on settings made by the machine manufacturer.

Please refer to the machine manufacturer's instructions.

Measuring radius/ diameter

- Attach the tool you want to measure to the spindle.
- Position the tool near the measuring probe so that it can be approached without collision.



Measuring radius/diameter

Jog | Meas. tool

- Select the "Measure tool" softkey in "Machine Manual" mode.

Radius Auto > or Dia. Auto >

- Then press the softkey "Radius Auto" or "Dia. Auto".
- Select the tool cutting edge D and the duplo number DP for the tool.
- Enter the longitudinal offset V, if necessary.

Cycle Start

- Press the "Cycle Start" key.

This starts the automatic measuring process. Measurement is performed with a spindle rotating in the opposite direction.
The tool radius or diameter is calculated automatically and entered in the tool list.
The measuring process depends on settings made by the machine manufacturer.

Please refer to the machine manufacturer's instructions.

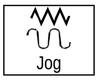
2.7.4 Calibrating a measuring probe

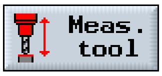
If you want to measure your tools automatically, you must first determine the position of the probe on the machine table with reference to the machine zero.

Mechanical tool measuring probes are typically shaped like a cube or a cylindrical disk. Install the probe in the working area of the machine (on the machine table) and align it relative to the machining axes.

You must use a mill-type calibration tool to calibrate the measuring probe. You must enter the length and radius/diameter of the tool in the tool list beforehand.

- Move the calibration tool until it is approximately over the center of the measuring surface of the probe.

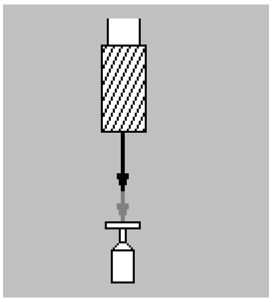




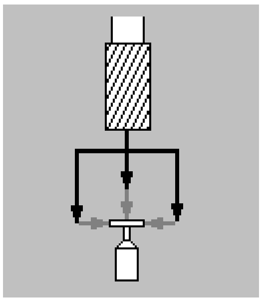
- Select the "Measure tool" softkey in "Machine Manual" mode.

Cal. meas. calipers >

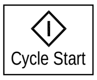
- Press the "Calibrate measuring calipers" softkey.
- Choose whether you want to calibrate the length or the length and the diameter.



Calibrating length only



Comparing length and diameter



- Press the "Cycle Start" key.

Calibration is automatically executed at the measuring feedrate. The distance measurements between the machine zero and measuring probe are calculated and stored in an internal data area.

2.8 Manual mode

Use "Machine Manual" mode to set up the machine for executing a program or when you want to perform simple traverse movements at the machine.

In manual mode you can:

1. Synchronize the measuring system of the control with the machine (reference point approach),
2. Set up the machine, i.e. activate manually controlled motions on the machine using the keys and handwheels provided on the machine control panel.
3. Activate manually controlled motions on the machine using the keys and handwheels provided on the machine control panel while a part program is interrupted.

2.8.1 Selecting a tool and attaching it to the spindle

For the preparatory work in manual mode, tool selection is performed centrally in a screenform.

Jog T,S,M

➢ Select the "T, S, M" softkey in "Machine Manual" mode.

The cursor is positioned on the input field of tool parameter "T":

T D1

➢ Enter the name or the number of the tool T.

-OR-

Tools or OFFSET

➢ Call up the tool list by pressing the "Tools" softkey or the "Offset" key.

➢ Place the cursor on the required tool in the tool list.

-AND-

In manual

➢ Press the "In manual" softkey.

The tool is transferred to the "T, S, M... window" and displayed in the field of tool parameter "T".

➢ Select tool edge D or enter the number directly in field "D".

Cycle Start

➢ Press the "Cycle Start" key.

The tool is attached to the spindle.

2.8.2 Entering a tool in the list and attaching it to the spindle

Preparing for loading

Jog T,S,M

- Select the "T, S, M" softkey in "Machine Manual" mode.

The cursor is positioned on the input field of tool parameter "T".

Entering tool in tool list

OFFSET or Tools

- Press the "Offset" or "Tools" softkey to open the tool list.

▲ ▼

- Enter a new tool (as described in Sec. "Tools and tool offsets").

In manual

- Press the "In manual" softkey.

You automatically return to function "T,S,M, …". The tool name is now entered in the input field of tool parameter "T".

Performing a tool change

Cycle Start

- Press the "Cycle Start" key.

Tool change is enabled.
The loaded tool is marked by a spindle symbol in the tool list.

Now load the tool manually into the spindle as described in the machine manufacturer's instruction manual.

2.8.3 Entering a new tool in the list and loading it in the magazine

Entering tool in tool list

OFFSET -or- Tools WOs

➢ Press the "Offset" key or the "Tool zero" softkey to open the tool list.

➢ Select a free slot in the tool list and enter a new tool (as described in Sec. "Tools and tool offsets").

New Tool

➢ Press the "New Tool" softkey.

Cutter ... 3D probe

➢ Select the desired tool type.

➢ Enter a name for the tool. Enter the tool offsets if applicable.

Loading tool to magazine

Load

➢ Press the "Load" softkey if you are using a magazine with variable location assignment.

If it is a magazine with fixed location assignment, load the tool in the required magazine location as described in the machine manufacturer's instruction manual.

2.8.4 Starting, stopping, and positioning a spindle manually

Setting the spindle speed

Jog T,S,M

➢ Select the "T, S, M" softkey in "Machine Manual" mode.

➢ Enter the spindle speed value of your choice in the "Spindle" input field.

Cycle Start

➢ Press the "Cycle Start" key.

If the spindle is already running the new speed is accepted. If the spindle is stationary, the value is stored as the setpoint speed. The spindle remains stationary.

Starting the spindle.

Spindle Left -or- Spindle Right

➢ Press the "Spindle Left" or "Spindle Right" key.

The spindle is started according to the preselected spindle speed and the current spindle override weighting.

Spindle Stop

You can stop the spindle again by pressing the "Spindle Stop" key.

- OR -

Jog T,S,M

➢ Select the "T, S, M" softkey in "Machine Manual" mode

SELECT

➢ In "Spindle M Fct.", select spindle direction of rotation

"right" or "left".

Cycle Start

➢ Press the "Cycle Start" key.

The spindle rotates.

Stopping the spindle

Jog T,S,M

➢ Select the "T, S, M" softkey in "Machine Manual" mode.

SELECT

➢ In "Spindle M Fct.", select "Off".

Cycle Start

➢ Press the "Cycle Start" key.

The spindle stops.

Positioning the spindle

The spindle position is specified in degrees.

Jog T,S,M

➢ Select the "T, S, M" softkey in "Machine Manual" mode.

SELECT

➢ Select "Spindle M Fct." "Stop Pos.".

Input field "Stop Pos." appears."

➢ Enter the desired spindle stop position.

The spindle position is specified in degrees.

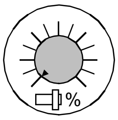
Cycle Start

The spindle is turned to the selected position when you press "Cycle Start".

You can use this function to position the spindle at a specific angle, e.g. during a tool change.

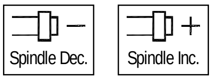
- A stationary spindle is positioned via the shortest possible route.
- A rotating spindle is positioned as it continues to turn in the same direction.

Changing the spindle speed



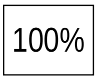
➢ Set the spindle speed with the spindle override switch.
You can select 50 to 120% of the value that last applied.

- OR - (on operator panel OP032S):



➢ Press the "Spindle Dec." or "Spindle Inc." key.

The programmed spindle speed (100%) is increased or decreased.



➢ Press the "100%" key.

The spindle speed is reset to the programmed spindle speed.

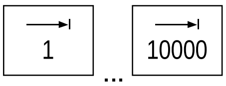
2.8.5 Traversing axes

You can traverse the axes in manual mode via the Increment and Axis keys or handwheels.
During a traverse initiated from the keyboard, the selected axis moves by a specified increment with the programmed setup feedrate.

Traversing the axes using the keyboard

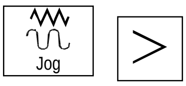
By pressing the increment keys, you can move the selected axis in defined increments in the appropriate direction every time you press an "Axis key" in manual mode.
The axes themselves traverse at the programmed setup feedrate.

Presetting increments



➢ Press keys [1], [10], ..., [10000] to move the axes through a defined increment.
The numbers on the keys indicate the traverse path in micrometers or micro-inches.
Example: For an increment of 100 μm (= 0.1 mm), press the "100" key.

-or-



➢ Open the extended horizontal softkey menu in "Machine Manual" mode.

➢ Press the "ShopM sett." softkey.

The settings menu opens.

➢ Enter the desired increment size in the "Variables increment" input field.
Example: For an increment of 500 μm (= 0.5 mm), enter 500.

[VAR]

➢ Press the "Inc Var" key.

The axes are traversed by the specified increments.

Setting the setup feedrate

➢ Enter a feedrate in the "Setup feedrate" field.

The axes are traversed in setup mode at the specified feedrate. A limitation for the maximum feed velocity is programmed in a machine data.

Traversing the axis

X ...

➢ Select the axis to be traversed.

− +

➢ Press the "+" or "-" key.

Each time you press the key the selected axis is traversed by the set increment.
Feedrate and rapid traverse override switches can be operative.

Depending on the PLC program, you may be able to select more than one axis at a time.

- When the control is switched on axes can be traversed to the limits of the machine because they have not yet been referenced. Emergency limit switches might be triggered as a result.
- The software limit switches and the working area limitation are not yet operative!
- The feed enable signal must be set.

Traversing axes using the handwheels

Please note the machine manufacturer's instructions with regard to the selection and mode of operation of handwheels.

2.8.6 Positioning axes

In "Machine Manual", you can move the axes to certain positions for the purpose of performing simple machining operations.

Jog | Posi- tion

- Select the "Position" softkey in "Machine Manual" mode.
- Select the axis or axes to be traversed with the "cursor up" and "cursor down" keys.
- Select the axis or axes to be traversed and enter the target position(s).
- Enter a feedrate in field "F".

\- OR -

Rapid traverse

- Press the "Rapid traverse" softkey if the axes are to be traversed in rapid traverse.

The rapid traverse is displayed in field "F".

Cycle Start

- Press the "Cycle Start" key.
  The axes are moved to the specified target position.

The axes are moved to the specified target position.

2.8.7 Swiveling

Manual swiveling provides functions that make setup, measuring, and machining of workpieces with inclined surfaces considerably easier.

If you want to create or correct an inclined position, the required rotations of the workpiece coordinate system around the geometry axes (X, Y, Z) are automatically converted to suitable positions of the swivel axes (A, B, C).
If you use manual swiveling, you can program the swivel axes of the machine directly and generate a matching coordinate system for those swivel axis positions.
If the swivel plane is active, the "Workpiece zero" function is operative but not the "Measure tool" function.
The swiveled coordinates are retained in reset state and even after power ON, i.e. you can still retract from an inclined hole by retracting in +Z direction.

Please refer to the machine manufacturer's instructions.

The following provides an explanation of the most important parameters for swiveling:

Moving clear

Before swiveling the axes you can move the tool to a safe retraction position. The retraction options available to you are defined when setting up the swivel data block in the "retraction position" parameter.

Please refer to the machine manufacturer's instructions.

Warning

You must select a retraction position that does not cause the tool and workpiece to collide in a swivel action.

Swivel methods

Swiveling can be axis by axis or direct.

- Axis-by-axis swiveling is based on the coordinate system of the workpiece (X, Y, Z). The coordinate axis sequence can be selected freely. The rotations are applied in the selected order. ShopMill calculates the rotation of the rotary axes (A, B, C) based on those values.
- In direct swiveling, the positions of the rotary axes are specified. ShopMill calculates a suitable new coordinate system based on those values. The tool axis is aligned in the Z direction. You can derive the resulting direction of the X and Y axis by traversing the axes.

The positive direction of each rotation for the different swivel methods is shown in the help displays.

Direction

In swivel systems with 2 rotary axes, a particular plane can be reached in two different ways. You can choose between these two different positions in the "Direction" parameter. +/- refers to the larger or smaller value of a rotary axis. This may affect the working area. When setting up the swivel data block, the entries you make in the "Direction" parameter determine for which rotary axis you can choose between the two settings.

Please refer to the machine manufacturer's instructions.

If one of the two positions cannot be reached for mechanical reasons, the alternative position is automatically selected irrespective of the setting in the "Direction" parameter.

Fixing the tool tip

To avoid collisions, you can use the 5-axis transformation (software option) to retain the position of the tool tip during swiveling. This function must be enabled when you set up "Swivel" in the "Correction T" parameter.

Please refer to the machine manufacturer's instructions.

Zero plane

You can use the "Manual swiveling" function both for machining and for setting up, to compensate for workpiece rotations when they are being clamped (base angle of rotation).
If you want to use the current swiveled plane as the reference plane for setting up your workpiece, you must define this plane as the zero plane.
With "Set zero plane" the current swivel plane in the active work offset is stored as the zero plane. As a result, the rotations in the active work offset are overwritten.
With "Delete zero plane", the active zero plane is deleted from the work offset. This sets the rotations in the active work offset to zero.
The overall coordinate system does not change with "Set zero plane" or "Delete zero plane".
You can also use manual swivel in conjunction with "Align plane" to measure a workpiece.

Jog | Swivel

➢ Select the "Swivel" softkey in "Machine Manual" mode.

➢ Enter values for the parameters.

Cycle Start

➢ Press the "Cycle Start" key.

The "Swivel" cycle is started.

Delete values

➢ Press the "Delete values" to restore the initial state, i.e. to reset the values to 0.
Perform this step, for example, to swivel the coordinate system back to its original position.

Set zero plane

➢ Press the "Set zero plane" softkey to set the current swivel plane to the new zero plane.

Delete zero plane

➢ Press the "Delete zero plane" softkey to delete the current swivel plane.

Teach rotary axis

➢ Select the "Teach rotary axis" softkey to accept the current positions of the rotary axes during direct swiveling.

<table>
<tr><th>Parameters</th><th colspan="2">Description</th><th>Unit</th></tr>
<tr><td>TC</td><td colspan="2">Name of the swivel data block
O: Remove swivel head, deselect swivel data block
No entry: No change to set swivel data block</td><td></td></tr>
<tr><td>Move clear</td><td colspan="2">No: Do not retract tool before swiveling
Z: Move tool axis to retraction position before swiveling
Z, X, Y: Move machining axes to retraction position before swiveling
Tool max: Retract tool in tool direction to software limit switch
Tool inc: Retract tool incrementally in tool direction by the entered value</td><td></td></tr>
<tr><td>Swivel plane</td><td colspan="2">Swivel new: Define new swivel plane
Swivel additive: Place swivel plane on last swivel plane</td><td></td></tr>
<tr><td>Swivel method</td><td colspan="2">Axial: Swivel coordinate system axially
Direct: Position rotary axes directly</td><td></td></tr>
<tr><td>X</td><td>Axis angle (swivel axially)</td><td rowspan="3">The sequence of axes can be changed to any order with "Alternat.".</td><td>Degr.</td></tr>
<tr><td>Y</td><td>Axis angle (swivel axially)</td><td>Degr.</td></tr>
<tr><td>Z</td><td>Axis angle (swivel axially)</td><td>Degr.</td></tr>
<tr><td>A</td><td colspan="2">Axis angle (swivel directly)</td><td>Degr.</td></tr>
<tr><td>B</td><td colspan="2">Axis angle (swivel directly)</td><td>Degr.</td></tr>
<tr><td>Direction</td><td colspan="2">Preferred direction of rotation given 2 alternatives
+: Larger angle of axis on scale for swivel head/table
-: Smaller angle of axis on scale for swivel head/table</td><td></td></tr>
<tr><td>Fix tool tip</td><td colspan="2">Follow-up: The position of the tool tip is maintained during swiveling.
No follow-up: The position of the tool tip changes during swiveling.</td><td></td></tr>
</table>

2.8.8 Face milling

You can use this cycle to face mill any workpiece. A rectangular surface is always machined.
For further information about the cycle, see Sec. "Programming - Face milling".

Jog | Face mill.

- Select the "Face mill." softkey in "Machine Manual" mode.
- Press the relevant softkey to specify the lateral limitations of the workpiece.
- Place the cursor in "Machine" and with the "Select" key choose a machining type (e.g. roughing).

...

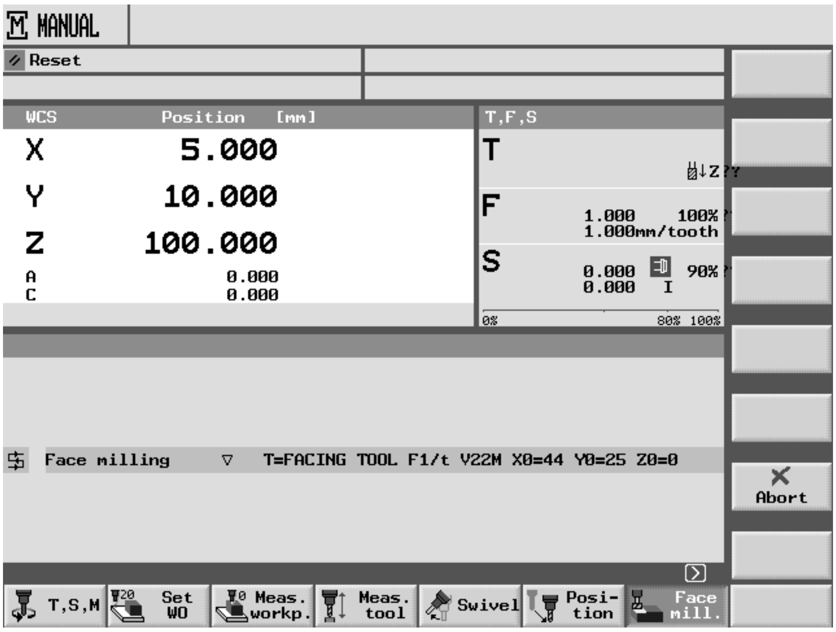
- Place the cursor in "Direction" and select the machining direction.
- Enter all other parameters in the input screen.

Please also note instructions regarding face milling in Sec. "Programming - Face Milling".

OK

- Press the "OK" softkey to confirm your entries.

Return to the program view in the Manual area.



Example of face milling in the program view

Cycle Start

Press the "Cycle Start" key to start the "Face milling" cycle.

You cannot use the "Repos" function while face milling.

2.8.9 Settings for manual mode

For manual mode, you can select the central machine functions and work offsets and set the unit of measurement.
Machine functions (M functions) are functions that are additionally provided by the machine manufacturer.

Please refer to the machine manufacturer's instructions.

In manual mode, you can display the axis positions and distance-defining parameters either in "mm" or "inches". However, tool offsets and work offsets remain in the original unit of measurement in which the machine was set (see Sec. "Switching over the unit of measurement millimeter/inch").

Selecting a gear stage

If your machine has a separate gear unit for spindles, you can select a gear stage.

Jog T,S,M

- Select the "T, S, M" softkey in "Machine Manual" mode.
- Position the cursor in the "Gear stage" field.

SELECT

- Select the gear stage you want to use (e.g. "auto").

This gear stage will be active the next time you press the "Cycle Start" key.

Selecting an M function

Jog T,S,M

- Select the "T, S, M" softkey in "Machine Manual" mode.
- Enter the number of the desired M function in the "Misc M fct." parameter field.
  Refer to the machine manufacturer's table for the correlation between the meaning and number of the function.

The M function will be active the next time you press the "Cycle Start" key.

Selecting a work offset

Jog T,S,M

- Select the "T, S, M" softkey in "Machine Manual" mode.

SELECT

- In the "Work offset" field, enter a work offset (e.g. base).

- OR -

Work offset

- Press the "Work offset" softkey to open the work offset list.

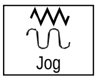
In manual

➢ Place the cursor on the required zero point and press the "In manual" softkey.

The work offset will be active the next time you press the "Cycle Start" key.

Setting the unit of measurement

The selected unit of measurement affects the actual value display and distance-defining parameters. The setting applies to the Manual area and remains valid until you switch to the other unit. In Automatic mode, the unit of measurement displayed in the program header is always activated.



➢ Select the "T, S, M" softkey in "Machine Manual" mode.

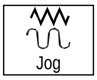
➢ Select a unit of measurement from the "Unit of measurement" box.

The unit of measurement will be active in manual mode the next time you press the "Cycle Start" key.

Selecting a tool axis

If your machine has a swivel-mounted work spindle, you can select the machining plane in the "Tool axis" selection field in menu "T, S, M".

This parameter is relevant for all screenforms in the Manual area, i.e. it influences the parameter displays for face milling or measurements. In addition, the plane setting determines how tool offsets are calculated in workpiece and tool measurements.



➢ Select the "T, S, M" softkey in "Machine Manual" mode.

➢ Select an axis from the "Tool axis" box.

That tool axis will be active in manual mode the next time you press the "Cycle Start" key.

For instructions on how to swivel the spindle, please refer to the machine manufacturer's instructions.

Changing the default settings

➢ Select the "Expand" softkey in "Machine Manual" mode to expand the softkey bar.

➢ Press the "ShopM. sett." softkey.

The ShopMill settings menu opens.

Retraction plane

➢ In the "Retraction plane" box enter the retraction position above the workpiece to be approached during face milling in rapid traverse in manual mode.

Safety clearance

➢ In the "Safety clearance" box, enter the position to which the axis is to traverse in rapid traverse.
The safety clearance is the distance between the tool tip and the workpiece surface. As soon as the safety clearance is reached, the programmed face milling cycle is executed at machining feedrate.

Setup feedrate

➢ In the "setup feedrate" box enter the feedrate with which you want to traverse the axes in manual mode.

Variable increment

➢ In the "variable increment" box enter an increment for traversing the axes in manual mode not at a fixed increment but at a variable increment.

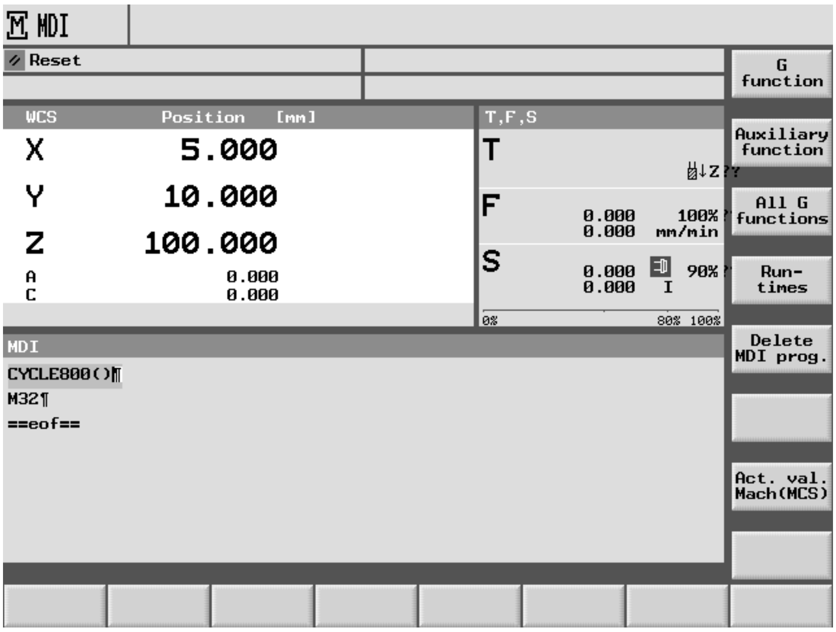
➢ Press the "Back" softkey.

The "ShopMill settings" menu box closes.
These settings remain valid until you change them.
These settings are made for the programs in the program header.

2.9 MDI mode

You can write and execute programs block by block in G code in "MDI" (Manual Data Input) mode. To do this, you enter specific movements as individual program blocks in the control via the keyboard.

The "MDI" program view displays position, feedrate, spindle and tool values as well as the contents of the MDI program.



Example of a program in the "MDI" program view

MDI

- Press the "MDI" key.
- Enter a G code in the working window.

Starting the program

Cycle Start

The control executes the blocks you have entered when you press the "Cycle Start" key.

Deleting the program

Delete MDI prog.

Programs written in MDI mode are automatically deleted as soon as they have finished running. Alternatively, you can delete them by selecting softkey "Delete MDI prog.".

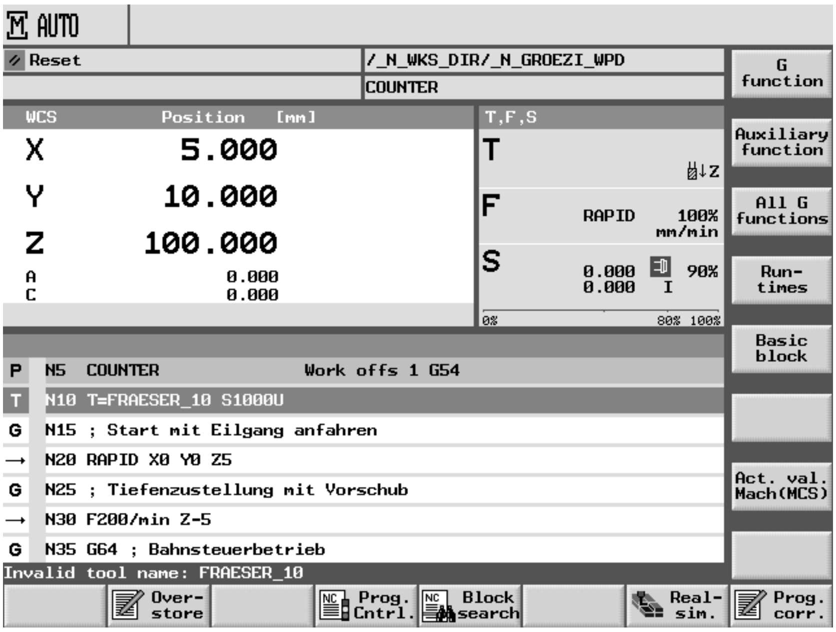
2.10 Automatic mode

Auto

In the "Machine Auto" operating mode, you can execute machining programs and monitor the progress of the current machining operation online on the screen.

Requirements for execution

- You have already synchronized the control measuring system with the machine (i.e. "approached" reference points).
- You have already written the relevant machining program.
- You have checked or entered the necessary offset values, such as work offsets or tool offsets.
- The required safety interlocks are already active.



Example of program view in "Machine Auto"

Sequential control programs produced with an earlier version of ShopMill can also be executed in the current sequential control version. If an older sequential control program is executed once in the current sequential control version, it is reclassified as being in the current sequential control version.

You can also execute a Version-6.3 sequential control program in ShopMill 6.2, provided that you consider the following points:

- If the machining type "Edge finishing" is programmed for a longitudinal groove in ShopMill 6.3, the parameter is replaced by "Roughing" in ShopMill 6.2.
- The functions "Deep drilling" and "Circumferential slot" programmed under ShopMill 6.3 only run in ShopMill 6.2 if you check the function parameters again in ShopMill 6.2 and confirm these.

When a ShopMill program with Version 6.3 is executed in ShopMill 6.2, it is deemed to be a Version-6.2 program.

2.10.1 Switchover between "T, F, S", "G functions" and "Auxiliary functions" displays

While the workpiece is being machined, if you want to know, for example, whether the tool tip radius compensation is currently active or which unit of measurement is being used, you can activate display of the G functions or auxiliary functions.

G functions

16 different G groups are displayed under "G function". Within a G group, only the G function currently active in the NC is displayed.

As an alternative, all G groups with all associated G functions are listed in "All G Func.".

Auxiliary functions

Auxiliary functions include M and H functions preprogrammed by the machine manufacturer, which pass parameters to the PLC to trigger reactions defined by the manufacturer.

Please refer to the machine manufacturer's instructions.
A maximum of five M functions and three H functions are displayed.

When executing a sequential control program, you can display the G functions currently active in the NC too, as the ShopMill functions are converted to G code internally.

G function

➢ Press the "G function" softkey in "Machine Manual" or "Machine Auto" mode.

Instead of parameters T, F and S, the currently active G functions within a G group will be displayed.
If you press the "G Function" softkey again, the "T, F, S" status display reappears.

- OR -

All
G func.

➢ Press the "All G func." softkey.

Instead of parameters T, F and S, all G groups with G functions are now listed. If you select the "All G func." softkey again, the "T, F, S" status display reappears.

- OR -

Auxiliary
function

➢ Press the "Auxiliary function" softkey.

Instead of parameters T, F and S, the currently active auxiliary functions will be displayed. If you press the "Auxiliary function" softkey again, the "T, F, S" status display reappears.

2.10.2 Selecting a program for execution

Pro- gram -or- PROGRAM MANAGER

➢ Press the "Program" softkey or the "Program Manager" key.

The directory overview is displayed.

➢ Place the cursor on the directory containing the program that you want to select.

INPUT -or- ▶

➢ Press the "Input" or "Cursor right" key.

The program overview is displayed.

➢ Place the cursor on the required program.

Execute

➢ Press the "Execute" softkey.

ShopMill automatically changes to "Machine Auto" operating mode and uploads the program.

- OR -

Pro- gram -or- PROGRAM MANAGER

➢ Press the "Program" softkey or the "Program Manager" key.

The directory overview is displayed.

➢ Place the cursor on the directory containing the program that you want to select.

INPUT -or- ▶

➢ Press the "Input" or "Cursor right" key.

The program overview is displayed.

➢ Place the cursor on the required program.

INPUT -or- ▶

➢ Press the "Input" or "Cursor right" key.

The selected program is opened in the "Program" operating area. The machining plan of the program is displayed.

➢ Place the cursor on the program block at which the program run must begin.

Ex- ecute

➢ Press the "Execute" softkey.

ShopMill automatically changes to "Machine Auto" manual mode, loads the program and conducts a block search until it reaches the selected program block (see Sec. "Entering a program at any selected point").

When you select a program for the first time for execution, that contains the cycles "Stock removal towards contour" or "Contour pocket", the individual stock removal steps or solid machining steps for the contour pocket are calculated automatically. This process may take several seconds depending on the complexity of the contour.

2.10.3 Starting/stopping/aborting a program

Shows how to start/stop programs that are loaded in "Machine Auto" operating mode and resume program execution after abnormal program termination.

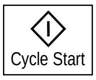
Once the program is loaded in "Machine Auto" mode, and "Automatic" mode is also activated on the machine control panel, you can start the program whatever your current operating area, even if you are not in "Machine Auto" mode.
This start option must be enabled in a machine data code.

Please refer to the machine manufacturer's instructions.

Precondition

No alarms are pending.
The program is selected.
Feedrate enable is active.
Spindle enable is set.

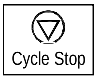
Starting execution



➢ Press the "Cycle Start" key.

The program is started and executed from the start or from the selected program block onwards.

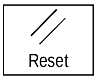
Stopping the program



➢ Press the "Cycle Stop" key.

Machining stops immediately, individual blocks do not finish execution. At the next start, execution is resumed at the same location where it stopped.

Aborting execution



➢ Press the "Reset" key.

Execution of the program is interrupted. When it is started again, it will execute from the beginning.

Starting program execution from an operating area

Cycle Start

The program is loaded in "Machine Auto" mode and "Automatic" mode is activated on the machine control panel.

➢ Press the "Cycle Start" key.

The program is started and executed from the beginning. However, the interface of the previously selected operating area remains on the screen.

2.10.4 Interrupting a program

Retracting from a contour

After you have interrupted a program ("NC Stop") in Automatic mode (e.g. in order to take a measurement on the workpiece and correct the tool wear values or after tool breakage), you can retract the tool from the contour in "Machine Manual" mode. In such cases, ShopMill stores the coordinates of the interruption point and displays the differences in distance traveled by the axes in "Machine Manual" mode as a "Repos" (= Reposition) offset in the actual value window.

For details of how to traverse machine axes, please refer to Sec. "Traversing machine axes".

Repositioning

The "Repos" function repositions the tool on the workpiece contour after traversal of the machine axes during a program interruption in Automatic mode.

Jog

➢ Select "Machine Manual" mode.

Move the axes away from the point of interruption.

Repos

➢ Press the "Repos" key.

X ...

➢ Select the axis to be traversed.

− +

➢ Press the "+" or "-" key.

It is not possible to overtravel the point of interruption.
The feedrate override is active.

Warning

The rapid traverse override switch is active.
Non-adjusted Repos offsets are adjusted with program advance and linear interpolation on switchover to Automatic mode followed by start with the "Cycle start" key.

2.10.5 Starting execution at a specific point in the program

If you only want to execute a particular section of a program on the machine, there is not obligation to start execution of the program from the beginning, you can also start processing from a specific program block or text string.

The point in the program at which you wish to start machining is called the "target".
ShopMill distinguishes between three different target types:

- ShopMill cycle
- Other ShopMill block or G code block
- Any text

For the "Other ShopMill block or G code block" target type, you can again define the target in three different ways:

- Position cursor on target block
  This is ideal for straightforward programs.
- Choose point of interruption
  Machining resumes at the point at which it was interrupted earlier. This is especially convenient in large programs with multiple program levels.
- Specify target directly
  This option is only possible if you know the precise data (program level, program name etc.) of the target.

Once the target has been specified, ShopMill calculates the exact starting point for program execution.
With "ShopMill cycle" and "Any text" target types, the calculation is always based on the end of the block. When calculating the starting point of all other ShopMill blocks and G code blocks, you can choose between four options.

1. Calculation **to end point:**
   During a block search, ShopMill makes the same calculations as when executing a program. The program is executed from the end of the target block or from the next programmed position of the target block.
2. **Without calculation**
   ShopMill performs no calculations during block search, i.e. the calculation is skipped up to the target block. The parameters within the control contain the values valid before the block search.
   This option is only available for programs that exclusively comprise G code blocks.

3. **External - without calculation**
   This method is performed in the same way as calculation to end point, except that subroutines called via EXTCALL are skipped in the calculation. In the same way, with G code programs that are executed entirely by external drives (disk drive/network drive), calculation is skipped until the target block is reached.
   This helps to speed up the calculation process.

Notice
Modal functions included in the part of the program that is not calculated are not taken into account for the part of the program to be executed. In other words, with the "Without calculation" and "External – without calculation" methods, you should choose a target block after which all the information needed for machining is included.

Specifying a target directly

With the "Other ShopMill block or G code block" target type, you can enter the target directly in the "Search pointer" screen form.
Each line of the screen form represents one program level. The actual number of levels in the program depends on the nesting depth of the program. Level 1 always corresponds to the main program and all other levels correspond to subroutines.
You must enter the target in the line of the screen form corresponding to the program level in which the target is located. For example, if the target is located in the subroutine called directly from the main program, you must enter the target in program level 2.
The specified target must always be unambiguous. This means for example that if the subroutine is called in the main program in two different places, you must also specify a target in program level 1 (main program).

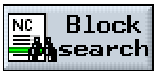
The parameters in the "Search pointer" screen form have the following meaning:

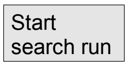
	Number of program level
Program:	Subroutine is in NC working memory Program name Example: subrt1 Subroutine is not in NC working memory Path + program level Example: c:\subrt1 or \\r1638\shopmill\subrt1 (the name of the main program is automatically entered.)
Ext:	File extension
P:	Continuous counter (if part of a program is repeated several times, you can specify the repetition number at which you wish machining to be resumed.)

Line:	Parameter is assigned by ShopMill	
Type:	" "	Search target is ignored on this level
	N no.	Block number
	Marker	Jump marker
	Text	Character string
	Sub-r.	Subprogram call
	Line	Line number
Search target:	Point in the program at which machining is to start	

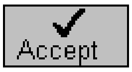
Selecting ShopMill cycle

- Load a program in "Machine Auto" operating mode (see Sec. "Selecting a program for execution").
- Position the cursor on the desired target block.

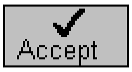




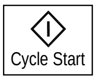
- Press the "Block search" and "Start search run" softkeys.
- Where chained program blocks have several technology blocks, select the desired technology block in the "Search run" window.
  The prompt does not appear in the case of single program blocks.



- Press the "Accept" softkey.
- For chained program blocks, enter the number for the desired starting position.
  The prompt does not appear in the case of single program blocks.

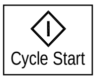


- Press the "Accept" softkey.



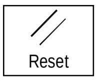
- Press the "Cycle Start" key.

ShopMill carries out all necessary default settings.



- Press the "Cycle Start" key again.

The new starting position is approached. The workpiece is then machined from the beginning of the target block.



You can abort the search by pressing the "Reset" key.

Select other ShopMill block or G code block

Position cursor on target block

- Load a program in "Machine Auto" operating mode (see Sec. "Selecting a program for execution").
- Position the cursor on the desired target block.

Block search

- Press the "Block search" softkey.

To endpoint ... External - without cal.

- Select a calculation technique.

Cycle Start

- Press the "Cycle Start" key.

ShopMill carries out all necessary default settings.

Cycle Start

- Press the "Cycle Start" key again.

The new starting position is approached. The program executes from the beginning or end of the target block, depending on the calculation technique.

Reset

You can abort the search by pressing the "Reset" key.

Select point of interruption
Program execution must have been interrupted by pressing the "Reset" key. (ShopMill automatically remembers this point of interruption.)

- Switch back to "Machine Auto" mode.

Block search Search pointer

- Press the "Block search" and "Search pointer" softkeys.

Interr. point

- Press the "Interr. point" softkey.

ShopMill inserts the saved point of interruption as the target.

To endpoint ... External - without cal.

- Select a calculation technique.

Cycle Start

- Press the "Cycle Start" key.

ShopMill carries out all necessary default settings.

Cycle Start

- Press the "Cycle Start" key again.

The new starting position is approached. The program executes from the beginning or end of the target block, depending on the calculation technique.

Reset

You can abort the search by pressing the "Reset" key.

Specify target directly

- Load a program in "Machine Auto" operating mode (see Sec. "Selecting a program for execution").

Block search | Search pointer

- Press the "Block search" and "Search pointer" softkeys.
- Specify the desired target.

To endpoint ... External - without cal.

- Select a calculation technique.

Cycle Start

- Press the "Cycle Start" key.

ShopMill carries out all necessary default settings.

Cycle Start

- Press the "Cycle Start" key again.

The new starting position is approached. The program executes from the beginning or end of the target block, depending on the calculation technique.

Reset

You can abort the search by pressing the "Reset" key.

Search for any text

- Load a program in "Machine Auto" operating mode (see Sec. "Selecting a program for execution").

Block search | Search

- Press the "Block search" and "Search" softkeys.
- Enter the text string that you want to locate.
- Select whether the search is to commence at the start of the program or the current cursor position.

Search

- Press the "Search" softkey.

The program block that contains the text string is marked.

Continue search

- Press the "Continue search" softkey, if you want to continue the search.

Abort | Start search run

- Press the "Abort" and "Start search run" softkeys.

Accept

- Where chained program blocks have several technology blocks, select the desired technology block in the "Search run" window and press the "Accept" softkey.
  The prompt does not appear in the case of single program blocks.

Accept

- For chained program blocks, enter the number for the desired starting position and press the "Accept" softkey.
  The prompt does not appear in the case of single program blocks.

Cycle Start

- Press the "Cycle Start" key.

ShopMill carries out all necessary default settings.

Cycle Start

- Press the "Cycle Start" key again.

The new starting position is approached. The workpiece is then machined from the beginning of the target block.

Reset

You can abort the search by pressing the "Reset" key.

2.10.6 Controlling the program run

During workpiece machining, if you want to check the result from time to time, you can stop machining at specified points (programmed stop). The program stops at "retraction plane" in the ShopMill program.

Conversely, if you do not want to execute certain machining steps programmed in G codes on every program run, you can mark these blocks accordingly (skip G code blocks). This is not possible with ShopMill blocks.

You can also choose to allow DRF offsets, i.e. offsets with the handwheel, during machining. This function must be set up by the machine manufacturer.

Please refer to the machine manufacturer's instructions.

Programmed stop

- Load a program in "Machine Auto" operating mode (see Sec. "Selecting a program for execution").

NC Prog. Cntrl.

- Press the "Prog. Cntrl." softkey.

Program. stop

- Press the "Program. stop" softkey.

Cycle Start

- Press the "Cycle Start" key.

Execution of the program starts. The program run stops at every block for which "programmed stop" was defined (see Sec. "Miscellaneous Functions").

Cycle Start

➢ Press the "Cycle Start" key again each time.

Execution of the program is continued.

Program. stop

➢ Press the "Program. stop" softkey again if you want the program to be executed without a programmed stop. (The softkey is deactivated again.)

Skipping G code blocks

➢ Load a program in "Machine Auto" operating mode (see Sec. "Selecting a program for execution").

Prog. Cntrl.

➢ Press the "Prog. Cntrl." softkey.

Skip

➢ Press the "Skip" softkey.

Cycle Start

➢ Press the "Cycle Start" key.

Execution of the program starts. G code blocks with the "/" character (slash) in front of the block number are not executed.

Skip

➢ Press the "Skip" softkey again if you want the marked G code blocks to be executed again during the next run. (The softkey is deactivated again.)

Allowing DRF offset

➢ Load a program in "Machine Auto" mode (see Sec. "Starting/ stopping program execution").

Prog. Cntrl.

➢ Press the "Prog. Cntrl." softkey.

DRF offset

➢ Press the "DRF offset" softkey.

Cycle Start

➢ Press the "Cycle Start" key.

Execution of the program starts. Offsets with the handwheel affect the machining process directly.

DRF offset

➢ Press the "DRF offset" softkey again if you no longer want to allow handwheel offsets during machining. (The softkey is deactivated again.)

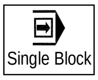
2.10.7 Overstore

In "Machine Auto" mode you can overstore technological parameters (auxiliary functions, programmable instructions, etc.) in the working memory of the NC. You can also enter and execute any NC block.

Overstore does not alter the programs stored in the part program memory.

Overstore with single block

➢ Load a program in "Machine Auto" operating mode (see Sec. "Selecting a program for execution").



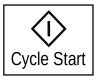
➢ Press the "Single Block" key to execute the program block by block.

The program automatically stops at the next block boundary.

➢ Press the "Overstore" softkey.

The "Overstore" window opens.

➢ Enter the NC blocks to be processed.

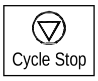


➢ Press the "Cycle Start" key.

The blocks you have entered are stored. You can observe execution in the "Overstore" window.
After "Overstore", a subroutine with the content REPOSA, which you can observe, is executed.

Overstore without single block

➢ Load a program in "Machine Auto" operating mode (see Sec. "Selecting a program for execution").

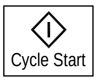


➢ Press the "Cycle Stop" key.

➢ Press the "Overstore" softkey.

The "Overstore" window opens.

➢ Enter the NC blocks to be processed.



➢ Press the "Cycle Start" key.

The blocks you have entered are stored. You can observe execution of the blocks in the "Overstore" window.
After the entered blocks have been executed, you can append blocks again.

Stopping overstore

Back

➢ Press the "Back" key to exit "Overstore".

The window closes.
You can switch modes now.
After you have pressed "Cycle Start" again, the selected program continues before overstore.

2.10.8 Testing a program

To prevent incorrect machining of the workpiece during the first pass of the program on the machine, first test the program without moving the machine axes.

ShopMill will then check the program for the following errors:

- Geometric incompatibility
- Missing data
- Non-executable instruction sequences and jumps
- Violation of working area

ShopMill automatically detects syntax errors when it loads a program in "Machine Auto" operating mode.

Whether ShopMill executes auxiliary functions (M functions and H functions) of not during the program test depends on the settings made by the machine manufacturer.

Please refer to the machine manufacturer's instructions.

The following functions can be used during the program test:

- Stop execution with "Programmed Stop" (see Sec. "Controlling the program run")
- Graphic display on screen (see Sec. "Simultaneous recording before machining").

➢ Load a program in "Machine Auto" operating mode (see Sec. "Selecting a program for execution").

Prog. Cntrl.

➢ Press the "Prog. Cntrl." softkey.

Program test

➢ Press the "Program test" softkey.

Cycle Start

➢ Press the "Cycle Start" key.

The program is tested without traversing of the machine axes.

Program test

➢ Press the "Program test" softkey again to deactivate test mode on completion of the program. (The softkey is deactivated again.)

2.10.9 Simultaneous recording before machining

In automatic mode you can display your program graphically in the "Program test" function before machining, without traversing the machine axes.
Simultaneous recording is a software option.

The graphic displays a workpiece as if it were being machined with a cylindrical tool.

Status displays

The status display in the graphic contains the following information:

- Current axis coordinates
- Block currently being processed.
- The processing time (in hours/minutes/seconds) indicates the approximate time that would actually be required to execute the machining program on the machine (incl. tool change). The timer is stopped if the program is interrupted.

➢ Select the program in "Machine Auto" mode.

Prog. Cntrl. Program test

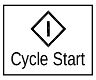
➢ Press the "Prog. Cntrl." and "Program test" softkeys.
➢ Also activate the "Dry run feedrate" softkey.

Dry run feedrate

The programmed feedrate is replaced by a dry run feedrate defined via machine data.

Real-sim

➢ Press the "Real-sim" softkey.



➢ Press "Cycle Start" to start the program.

You can still use the program control functions such as "Cycle Stop", "Single block", "Feedrate override", etc.

Program view

➢ Press the "Program view" softkey.

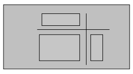
This changes the display from the "Simultaneous recording" graphic to the program view in Automatic mode. Recording of the graphical data continues in the background.

You can return to the graphic display by pressing one of the following softkeys:

Plan view

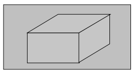
➢ Press the "Plan view" softkey.

- OR -



➢ Press the "3-Plane View" softkey.

- OR -



➢ Press the "3D Display" softkey (volume model).

You return from the program view to the graphical display.

Delete screen

➢ Press the "Delete screen" softkey.

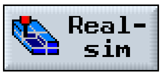
The graphical display of the previous machining is cleared. Recording of the machining operation will however continue.
For more information on the principles and operation, please refer to Sec. "Simulation".

2.10.10 Simultaneous recording during machining

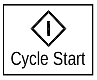
You can track the current machining operation on the machine tool simultaneously by monitoring the graphic display on the control screen.

Precondition

Program test and dry run feedrate must not be selected.
Simultaneous recording is a software option.



Press the "Real-sim" softkey and



start the program with "Cycle Start".

The "simultaneous recording" function can be switched on at any time during machining.

An explanation of the functions available under "Simultaneous recording" can be found in Secs. "Simultaneous recording before machining" and "Simulation".

2.11 Trial program run

2.11.1 Single block

If this function is active, execution is interrupted after every block that triggers a function on the machine (calculation blocks are not affected).

Standard setting

The following defaults apply:

- for drilling, the entire machining procedure and
- for pocket milling, the machining of a single plane

is combined in a single block.

Select with softkey

Single block fine

"Single block fine" active

When the "Single block fine" function is active, each individual drill infeed and pocket milling motion is executed as a separate block. Also, execution is stopped at the contour after every single contour element.

Select with softkey

Single block fine

Single block via machine control panel

Single Block

Activate the "Single block" key in "Machine Auto" mode. It will allow you to process a program block by block. If single block is activated, the associated LED on the machine control panel lights up.

If single block mode is active,

- the message "Stop: block in single block ended" is output in the channel mode message line (in the interrupt state).
- the current block of the program is not executed until you press the "Cycle Start" key,
- if machining stops after a block has been processed,
- you can start execution of the next block by pressing key "Cycle Start" again.

Deselecting a single block

Single Block

You can deselect the function by pressing the "Single block" key again.

2.11.2 Displaying the current program block

If you want precise information about axis positions and key G functions during a trial run or execution of the program, you can show the basic block display.

You can use the basic block display both in test mode and when machining the workpiece on the machine. All G code commands that initiate a function on the machine are displayed in the "Basic block" window for the currently active program block:

- Absolute axis positions
- G functions for the first G group
- Other modal G functions
- Other programmed addresses
- M functions

The basic block display function must be set up by the machine manufacturer.

Please refer to the machine manufacturer's instructions.

➢ Load a program in "Machine Auto" operating mode (see Sec. "Selecting a program for execution").

Basic block

➢ Press the "Basic block" softkey.

Single Block

➢ Press the "Single Block" key if you wish to execute the program block by block.

Cycle Start

➢ Start program execution.

The precise axis positions, modal G functions, etc., are displayed in the "Basic block" window for the currently active program block.

2.11.3 Correcting a program

As soon as the control detects a syntax error in the program, it interrupts the program and displays the syntax error in the alarm line. If an error occurs (Stop state), you can edit the program in the program editor.

➢ Select a program in "Machine Auto" mode.

The program status must be either "Stopped" or "Reset".

➢ Press the "Prog. corr." softkey.

The program editor opens.

If an error has occurred, the errored block is marked. Press the "Input" key and then correct the block.

➢ Press the "Accept" softkey to transfer the correction to the current program.

Continuing machining

➢ Press the "Execute" softkey followed by the "Cycle Start" key.

Execution of the program is continued.

- Cycle Stop status:
  You can only modify blocks that have not yet been executed or read in by the NC.
- Reset status:
  All blocks can be edited.

2.12 Run times

To provide you with an overview of the most important machine run times, ShopMill features a status window that displays the following operating times.

Program

Measurement of the program run time starts as soon as you press the "Cycle Start" key and stops on NC Stop or NC Reset.
If you start a new program, timing starts again from the beginning.

Timing continues if dwell time is active, or during program runs with program test or dry run feedrate. Timing stops with NC Stop or feedrate override = 0.

Workpiece

The current repetition and the programmed number of program repetitions (e.g. Workpiece: 15/100) are displayed. The number is only displayed in ShopMill programs and only when the number of programmed repetitions N is greater than 1. As from a programmed repetition of 100000 there is only enough room to display the current program repetition (e.g. Workpiece: 15).
If no information is yet available about the current program repetition, only two dashes are displayed (e.g. Workpiece: - -/100).

Time

The current time is displayed here.

Date

Today's date is displayed.

Machine

The machine run time displays how much time has elapsed since the control was last switched on.

Machining

The machining time displays the total run time of all programs executed since the control was last switched on.

Machine utilization

The system calculates the actual machine utilization from the timed machining time and the current machine run time.
The ratio of machining time to machine run time is displayed as a percentage.

A setting in the machine data determines which run times are displayed.

Please refer to the machine manufacturer's instructions.

Jog - or - Auto

- Select "Machine Manual" or "Machine Auto" mode.

Run times

- Press the "Run times" softkey.

The T,F,S display window turns into the "Run times" window.
Pressing the "Run times" softkey again, takes you back to the T,F,S display window.

2.13 Tools and tool offsets

You can manage tools with ShopMill. The following lists are available to you for this function

- Tool list
- Tool wear list
- Magazine list

Enter the tools, their offset data and wear monitoring data in the tool list or tool wear list. You will be able to identify in the magazine list which magazine locations are disabled or not.

Depending on individual requirements, a tool list might consist of the following:

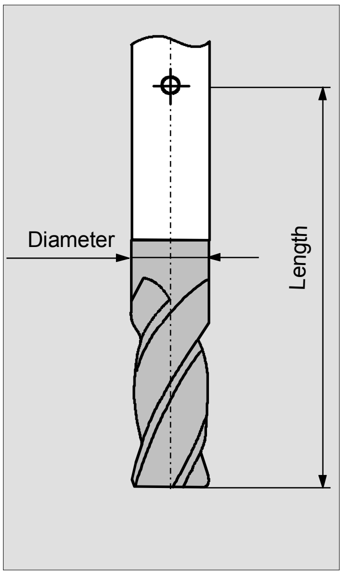
- A tool changer comprising
  - a spindle without dual gripper
  - or a spindle with dual gripper
- at least one tool magazine
- and tools that are not assigned to any tool magazine.

For details of the functionality of your tool management system, please refer to the machine manufacturer's instructions.
The different lists can be adapted by the machine manufacturer, if necessary.

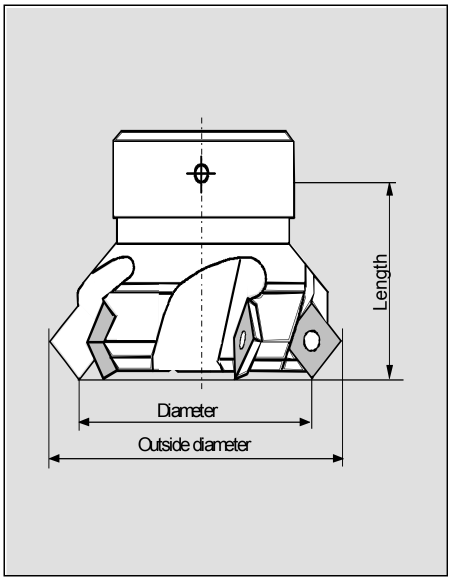
Tool list

The tool list displays all tools and their offset data stored as a tool data block in the NC, irrespective of whether or not they are assigned to a magazine location. The tool list offers you all commonly used tools. You can assign geometric and technological tool data to tool types. Various different examples of each tool might exist to which you can assign the various offset data of the tool being used.

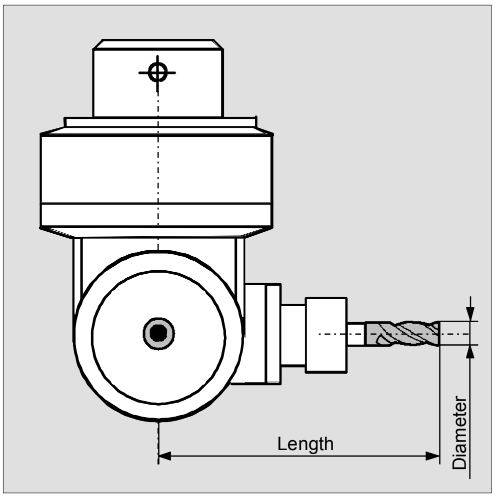
Depending on the tool type used, different tool offset data will be required.



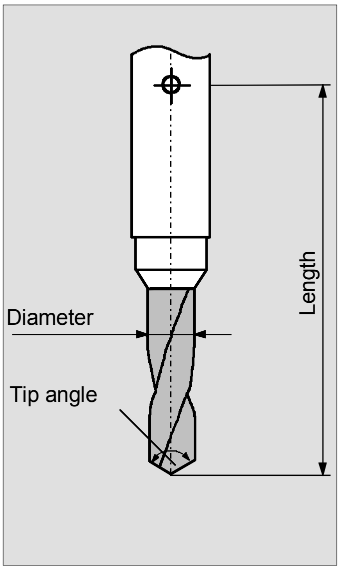
Mill



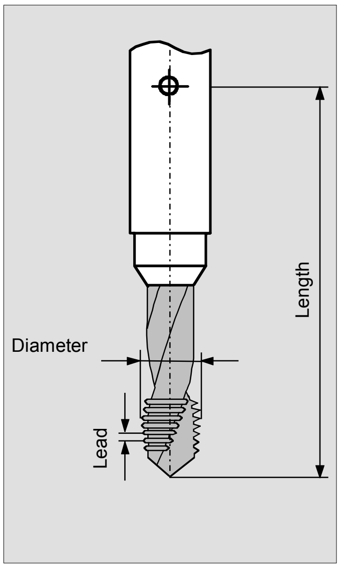
Face mill



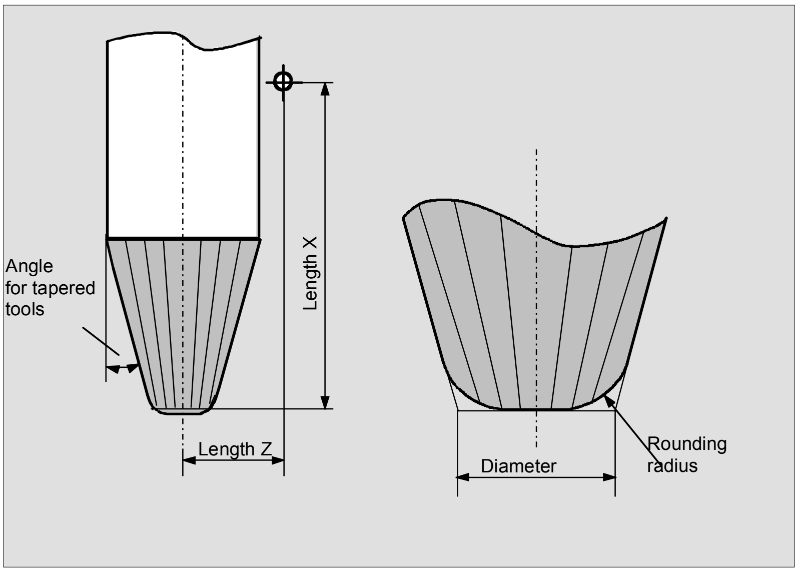
Angle head cutter



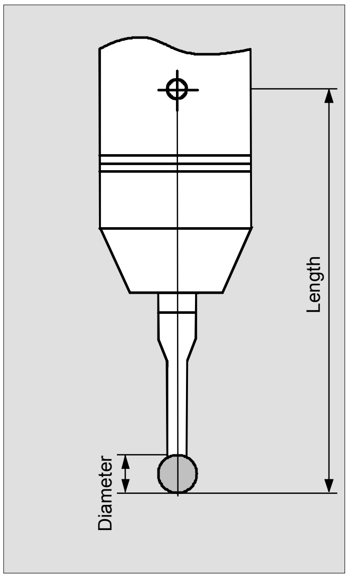
Drill



Threading tap



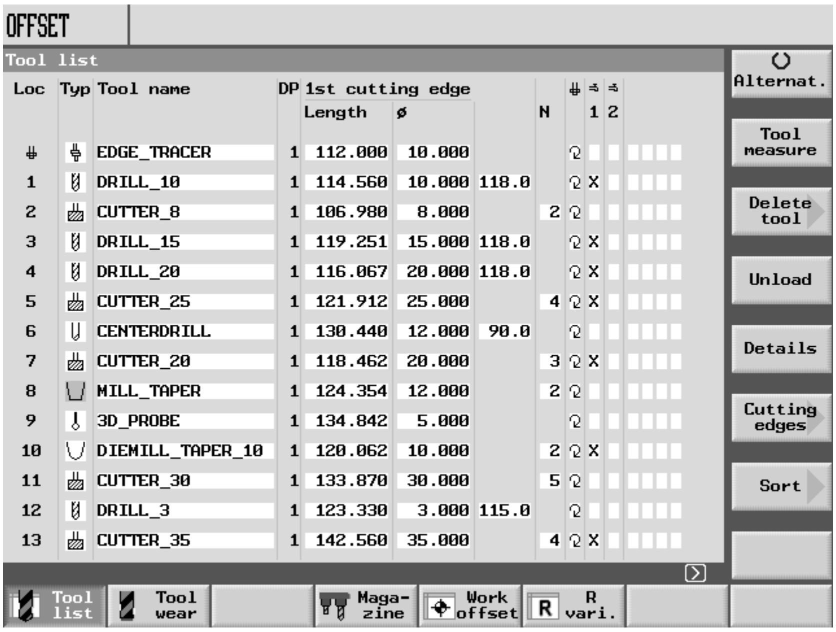
3D tool



3D probe

You can load and unload tools to and from a magazine via the tool list. When a tool is loaded, it is moved from its storage location to a magazine location. When it is unloaded, it is removed from the magazine and taken back to a storage location.
The loading and unloading of tool magazines is defined in a machine data.

Please refer to the machine manufacturer's instructions.



Example of a tool list with variable location assignment

The main display of the "Tools" operating area shows the current tool list with the following data:

Loc.

Location number
The following designations/symbols are used for:

- The spindle location
- The locations for gripper 1 and gripper 2 (applies only when a spindle with dual gripper is used)
- The magazine location numbers
  If the configuration includes more than one magazine, the location number is specified first followed by the magazine number in the magazine:
  e.g. 10/1 = Location number 10 in magazine 1
  5/2 = Location number 5 in magazine 2
- Tools that are not assigned to a magazine in the tool list are stored in a location without location number.
  This allows management of tools that are not actually contained in the tool magazine.

Type

Tool type
Depending on the tool type (represented by a symbol), only certain tool offset data are enabled.

Tool name

A tool is identified by its name and duplo number of the tool. You can enter the name as text or a number (see Sec. "Changing tool names").

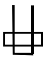
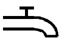
DP

Duplo number of twin tool (replacement tool)

Tool offset data

(D No.) edge	Tool offset data for the selected cutting edge of a tool (D No.)
Length	Tool length You can determine this value using the "Measure tool" function (see Sec. "Measuring the tool manually"). If the tool is measured externally, you can enter the value here.
Radius or ∅	Radius or diameter of the tool You can also enter the diameter for milling cutters and drills. A machine data code is used to switch from radius to diameter specification.
Angle	Angle of tool tip on a drill. If you want to insert a drill down to the shank, and not just to the tool tip, the control also takes the angle of the drill tip into account.
H	The "H" column is displayed only if ISO dialects are set up. Every H number of an ISO dialect program must be assigned to a tool offset data record.
N	Number of teeth for a milling cutter
Lead	Thread lead of a tap in mm/rev or turns/" if the inch system is set up on the machine.

Tool-specific functions

	Spindle rotation
	Coolant supply 1 and 2 can be activated/deactivated (e.g. internal and external cooling)
Tool-spec. fct 1...4	Other tool-specific functions such as additional coolant supply, monitoring functions for speed, tool breakage, etc.

Please refer to the machine manufacturer's instructions.

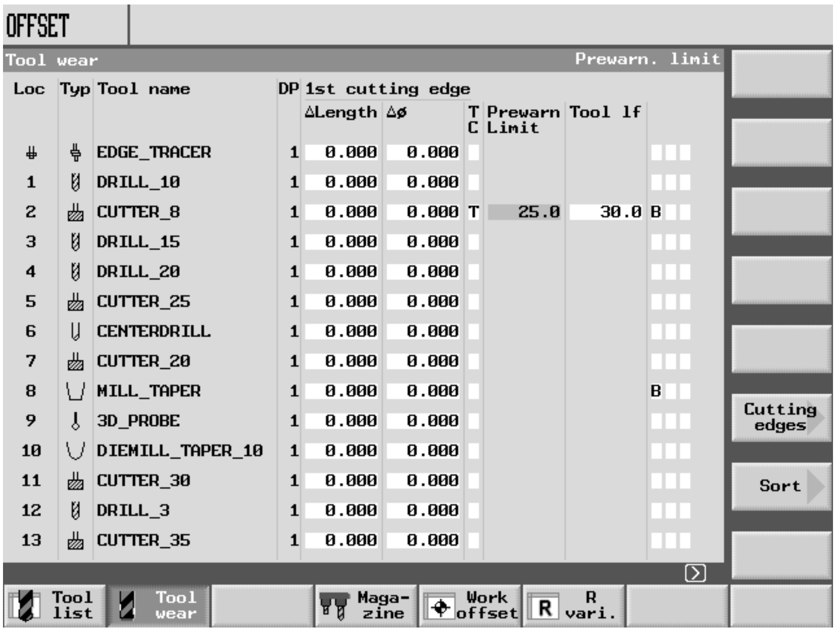
The "Details" softkey displays the additional parameters "Rounding radius" or "Angle" for 3D tapered milling tools.
For the facing tool, an additional outside radius and tool angle, and for angle head cutter, additional lengths and wear lengths are displayed under "Details.

Tool wear list

You can adapt the tool geometry (length and radius/diameter) to the wear-induced geometry in the tool wear list.

The following monitoring modes can also be selected for a tool.

- Monitoring of the effective operating time (tool life)
- Monitoring of number of tool load operations (quantity)
- Monitoring of wear
- Other tool status data (disable tool, tool in fixed location, oversized tool)



Loc	Typ	Tool name	DP	1st cutting edge ΔLength	Δø	T C	Prewarn Limit	Tool lf	
		EDGE_TRACER	1	0.000	0.000				
1		DRILL_10	1	0.000	0.000				
2		CUTTER_8	1	0.000	0.000	T	25.0	30.0	B
3		DRILL_15	1	0.000	0.000				
4		DRILL_20	1	0.000	0.000				
5		CUTTER_25	1	0.000	0.000				
6		CENTERDRILL	1	0.000	0.000				
7		CUTTER_20	1	0.000	0.000				
8		MILL_TAPER	1	0.000	0.000				B
9		3D_PROBE	1	0.000	0.000				
10		DIEMILL_TAPER_10	1	0.000	0.000				
11		CUTTER_30	1	0.000	0.000				
12		DRILL_3	1	0.000	0.000				
13		CUTTER_35	1	0.000	0.000				

Example of a tool wear list with variable location allocation

Tool magazine

The magazine locations with tools are specified in the magazine list. The list also indicates whether the magazine location is disabled (location disable) and the properties (tool status) assigned to the tools.

Fixed/variable location assignment

You can set a machine data to determine whether all tools must have a variable or fixed location assignment in the magazine.

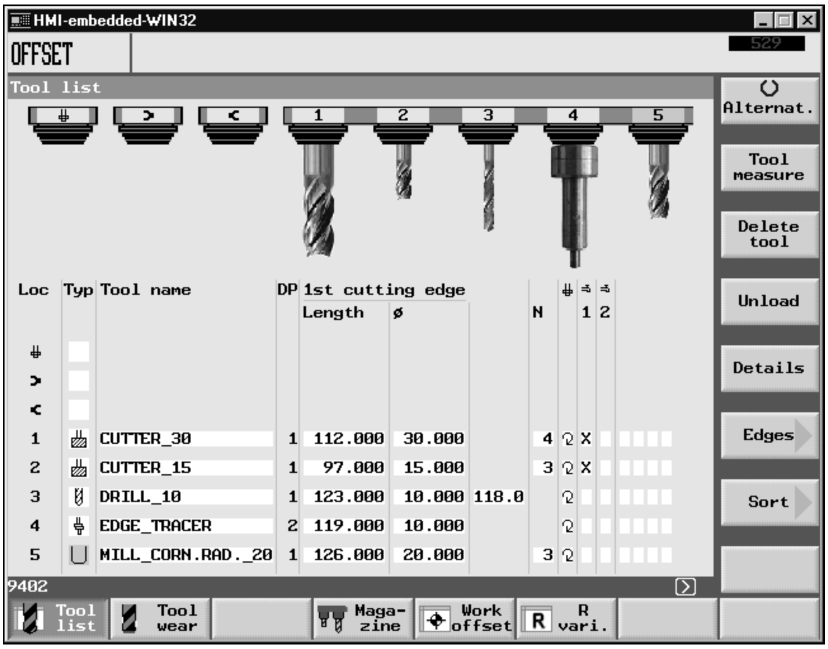
If you select a variable location assignment, the tools are taken to the next available space in the magazine after a tool change. With a fixed location assignment, the tools are always taken back to the location assigned specifically to them.

Please refer to the machine manufacturer's instructions for details about location assignments in the tool magazine.

Graphical display of tools and magazine locations

In addition to the list of tools, you can also display the tools and magazine locations in a dynamic graphic display. The tools are displayed in the order in the list with the correct proportions.
You can show or hide the graphical display with the "Help" key.
The graphical display must be set up by the machine manufacturer.

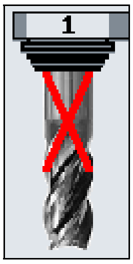
Please refer to the machine manufacturer's instructions.

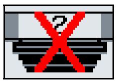


Graphical display of tools and magazine locations

The following applies for the graphical display:

- Small milling tools and 3-D tools are displayed as end mills, large ones as hobs.
- If a tool is too long for the display, the maximum possible length is shown.
- Oversized tools are truncated on the left and right sides.
- Tools that are not located in the magazine are displayed without toolholder.
- Disabled tools or magazine locations are marked as follows:

Disabled tool: 

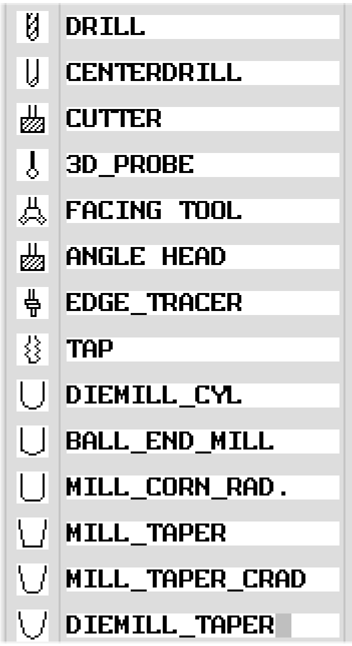
Disabled magazine location: 

- The data of the relevant tool nose selected are used for the display.
  If there is no tool nose for a tool in the selected view, the data of the first tool nose are used.

2.13.1 Creating a new tool

You can enter tools and the associated compensation data directly into the tool list or simply read existing tool data outside the tool management (see Section "Backup/restore tool/zero data").

If you want to enter a new tool directly into the tool list, ShopMill offers a range of conventional tool types. The tool type determines which geometry data are required and how they will be computed. The following common tool types are available:



➢ Attach the new tool to the spindle.

MENU SELECT ... Tools WOs

➢ Change operating area with "Menu Select" and press "Tools WO".

Tool list

The tool list opens.

▲ ▼

➢ Place the cursor on the location in the tool list that the tool occupies in the spindle. The location must still be vacant in the list.

New tool

➢ Press the "New tool" softkey.

Cutter ... 3D probe

➢ Select the tool type with the relevant softkey.
Additional tool types are available via the "More" softkey.

The new tool is created and automatically assumes the name of the selected tool type.

➢ Enter a unique tool name.

➢ Enter the offset data of the tool.

In the case of facing tools, angle head cutters, and 3D tools, you must define parameters in addition to the geometry data in the tool list.

Details

➢ Press the "Details" softkey and enter the additional parameters.
The "Details" softkey is only active when a tool is selected for which additional information is required.

Name	Additional parameters
Angle head cutter	Length2, Length3, ∆Length2, ∆Length3
Facing tool	Outside diameter, tool angle

3D tools

Type	Name	Additional parameters
110	Cylindrical die mill	-
111	Ball end mill	Smoothing radius
121	End mill with corner rounding	Smoothing radius
155	Bevel cutter	Angle for conical tools
156	Bevel cutter with corner rounding	Rounding radius, angle of conic. tools
157	Tapered die mill	Angle for conical tools

.

2.13.2 Setting up more than one edge per tool

In the case of tools with more than one cutting edge, a separate set of offset data is assigned to each cutting edge. You can set up a total of 9 edges for each tool.

There must be no gaps between the edges, i.e. if 3 edges are required for a tool, these must be edges 1 to 3.

In the case of ISO programs (e.g. ISO dialect 1) you must specify an H number. This corresponds to a particular tool offset set.

Follow the instructions given above to set up tools with more than one edge in the tool list and enter the offset data for the 1st edge.

Cutting edges > | New cutting edge

➢ Then select the "Cutting edges" and "New cutting edge" softkeys.

Instead of the input fields for the first cutting edge, the offset data input fields for the second cutting edge are displayed.

➢ Enter the offset data for the second cutting edge.

➢ Repeat this process if you wish to create more tool edge offset data.

Delete cutting edge

➢ Select the "Delete cutting edge" softkey if you want to delete the tool edge offset data for an edge.
You can only delete the data for the edge with the highest edge number.

D No. + | D No. -

By selecting softkey "D No. +" or "D No. –", you can display the offset data for the edge with the next highest or next lowest edge number respectively.

2.13.3 Changing the tool name

A tool that has just been created in the tool list is automatically assigned the name of the selected tool group. You can change this name as often as you want to

- a tool name, e.g. "Facing tool_120mm", or
- a tool number, e.g. "1".

The tool name must not exceed 17 characters in length. You can use letters, digits, the underscore symbol (_), periods (".") and slashes ("/").

2.13.4 Creating a replacement tool

A replacement tool is one that can be employed to perform the same machining operation as a tool that already exists (e.g. as a replacement after tool breakage).

When you create a tool as a replacement, you must give it the same name as an existing comparable tool.

INPUT

Confirm the name with the "Input" key. The duplo number of the replacement tool will automatically be incremented by 1.

The order in which replacement tools are attached to the spindle is determined by duplo number **DP**.

2.13.5 Manual tools

Manual tools are tools which are required during machining, but are only available in the tool list but not in the tool-holding magazine. These tools must be attached/detached manually to/from the spindle.

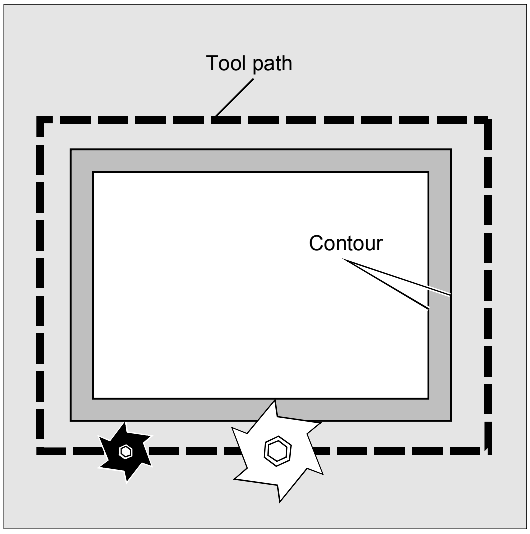
The "manual tool" function must be set up by the machine manufacturer.

Please refer to the machine manufacturer's instructions.

2.13.6 Tool offsets

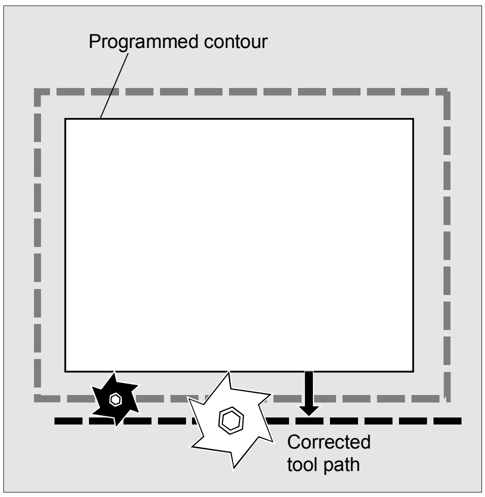
Why use tool offsets?

You do not have to take tool diameters and lengths into account when writing machining programs.
You can program workpiece dimensions directly, e.g. as specified in the production drawing.
When machining a workpiece, the tool paths are controlled according to the tool geometry such that the programmed contour can be machined using any tool.



The control corrects the travel path

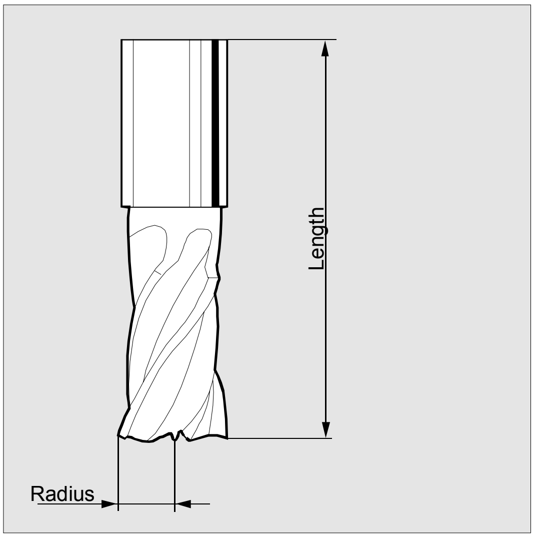
Enter the tool data separately in the "Tool list" and "Tool wear" tables.
When writing the program, you only need to call the tool you require.
While the program is being processed, the control fetches the offset data it requires from the tool table and corrects the tool path individually for different tools.



What type of tool offsets are available?

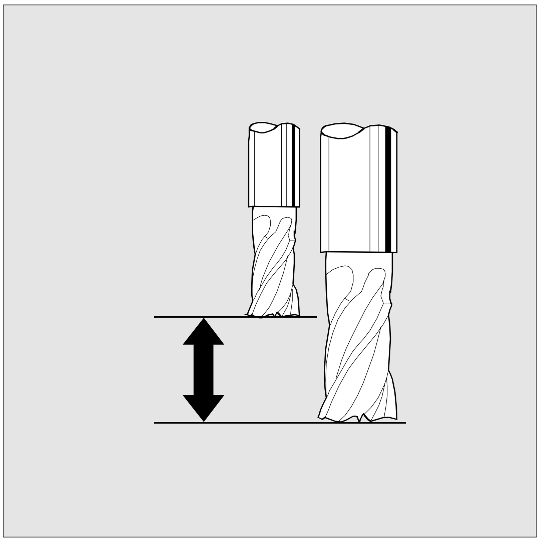
The offset memory of a tool includes the following:

- Tool type
  The tool type determines which tool data are required and how they must be calculated (e.g. drill, centering tool, mill).
- Total size: Length, radius, angle (drill)
  These comprise several components (geometry, wear). The control performs calculations on the components to obtain a final size (e.g. total length, total radius). The relevant total dimension becomes valid as soon as the offset memory is activated.



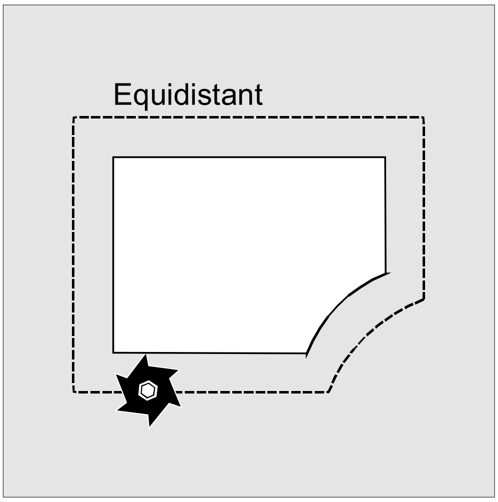
Tool length offset

This value compensates for the differences in length between the tools used.
The tool length is interpreted to be the distance between the toolholder reference point and the tool tip. This measured length is entered in the tool list.
The control uses this measurement and the wear values to calculate travel movements in the infeed direction.



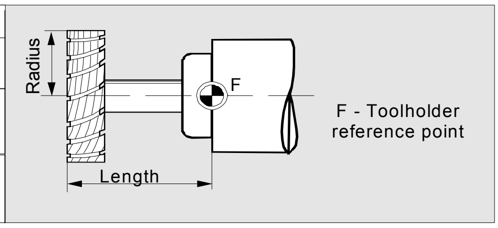
Tool radius compensation

The contour and tool path are not identical. The cutter or tool nose radius center must travel along a path that is equidistant from the contour.
For this purpose, the programmed tool center point path is automatically displaced by the control – as a function of radius and machining direction – in such a way that the tool edge travels exactly along the programmed contour.
The tool radius must be entered in the tool list. The control fetches the required radii during program execution and calculates the tool path from these values.



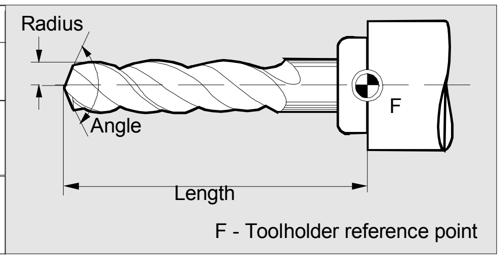
Offset values milling tool (example)

Infeed	Geometry in plane
Z	Length in Z Radius in X/Y
Y	Length in Y Radius in Z/X
X	Length in X Radius in Y/Z



Offset values drill (example)

Infeed	Geometry in plane
Z	Length in Z
Y	Length in Y
X	Length in X



Offset values are used in the simulation display and programming graphic for the following tools:

- Drill: Angle and radius/diameter
- Centering tool: Radius/diameter

2.13.7 Miscellaneous functions for a tool

You can assign other functions to tool types in the tool list.

Number of teeth N

Specify the number of teeth in this parameter. Parameter **N** is tool-dependent and can be applied only for milling tools. The control system calculates feedrate F internally if the feed is set in mm/tooth in the program.

Using the "Alternat." softkey, you can activate and deactivate the spindle direction of rotation (CCW/CW) in parameter "Spindle".

The spindle rotates clockwise.		Selection with softkey
The spindle rotates counterclockwise.		Alternat.
The spindle is stopped.		

Use parameters "coolant 1" and "coolant 2" if you want to supply coolant for the tool, e.g. for internal and external cooling.

Switch coolant ON:	☒	Selection with softkey
Do not switch on coolant:	☐	Alternat.

Tool-specific functions

You can also assign another four machine-specific actions to a tool. You can switch these tool-specific functions on and off with the "Alternat." softkey. Tool-specific functions might be, for example, 3rd coolant application or tool breakage monitoring.

Please refer to the machine manufacturer's instructions.

2.13.8 Entering tool wear data

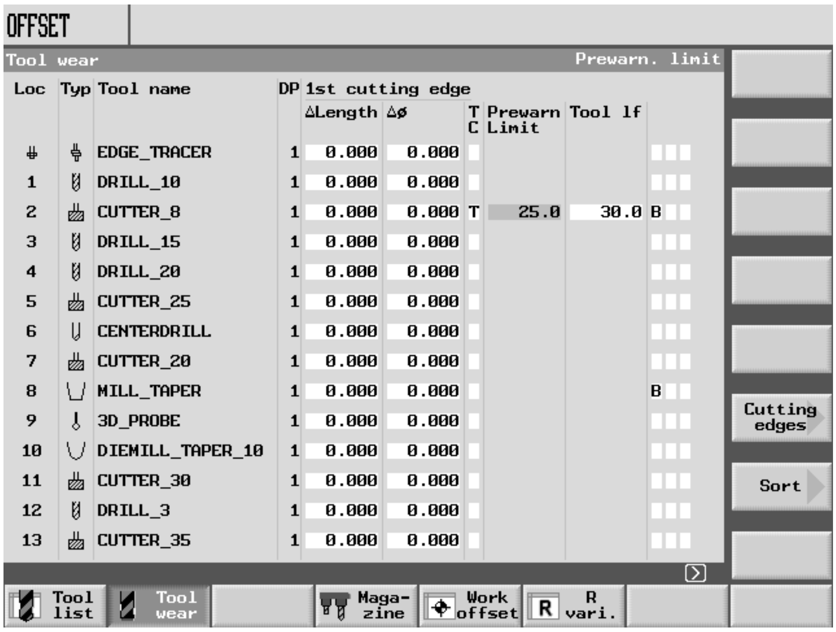
Tools that are in use for long periods are subject to wear. You can measure this wear and enter it in the tool wear list. ShopMill then takes this information into account when calculating the tool length or radius compensation. This ensures a consistent accuracy in workpiece machining.

When you enter the wear data, ShopMill checks that the values do not exceed an incremental or absolute upper limit. The incremental upper limit indicates the maximum difference between the previous and new wear value. The absolute upper limit indicates the maximum total value that you can enter.
The upper limits are set in a machine data code.

Please refer to the machine manufacturer's instructions.

Tools WOs | Tool wear

➢ Select the "Tool wear" softkey in the "Tools WOs" operating area.


OFFSET

Tool wear — Prewarn. limit

Loc	Typ	Tool name	DP	1st cutting edge ΔLength	Δø	T C	Prewarn Limit	Tool lf	
		EDGE_TRACER	1	0.000	0.000				
1		DRILL_10	1	0.000	0.000				
2		CUTTER_8	1	0.000	0.000	T	25.0	30.0	B
3		DRILL_15	1	0.000	0.000				
4		DRILL_20	1	0.000	0.000				
5		CUTTER_25	1	0.000	0.000				
6		CENTERDRILL	1	0.000	0.000				
7		CUTTER_20	1	0.000	0.000				
8		MILL_TAPER	1	0.000	0.000				B
9		3D_PROBE	1	0.000	0.000				
10		DIEMILL_TAPER_10	1	0.000	0.000				
11		CUTTER_30	1	0.000	0.000				
12		DRILL_3	1	0.000	0.000				
13		CUTTER_35	1	0.000	0.000				

Example of a tool wear list with variable location allocation

➢ Place the cursor on the tool whose wear data you want to enter.

➢ Enter the differences for length (ΔLength X, ΔLength Z) and radius/diameter (ΔRadius/Δ∅) in the appropriate columns.

The wear data entered is added to the radius but subtracted from the tool length. A positive differential value for the radius therefore corresponds to an oversize (e.g. for subsequent grinding).

2.13.9 Activating tool monitoring

ShopMill allows you to monitor the tool life of the tools automatically to ensure constant machining quality.

You can also disable tools that you no longer want to use, identify them as oversize or assign them permanently to a magazine location.

Tool monitoring can be activated via a machine data code.

Please refer to the machine manufacturer's instructions.

Tool life (T)

With the tool life T (Time), the service life for a tool with machining feedrate is monitored in minutes. When the remaining tool life is = 0, the tool is set to "disabled". The tool is not put into operation on the next tool change. If a replacement tool is available, it is inserted in its place.
Tool life is monitored on the basis of the selected tool cutting edge.

Count (C)

With the count C, on the other hand, the number of times a tool is attached to a spindle is counted. The tool is also disabled in this case, when the remainder reaches "0".

Wear (W)

With wear W, the greatest value in the wear parameters ∆ Length X, ∆ Length Z, or ∆ Radius or ∆ ∅ in the wear list is monitored. Here, too, the tool is disabled if one of the wear parameters reaches the value for wear W.

The wear monitoring function must be set up by the machine manufacturer.

Please refer to the machine manufacturer's instructions.

Prewarning limit

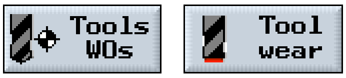
The prewarning limit specifies a tool life, workpiece count or wear at which the first warning is displayed.
The value for output of a warning due to the wear stage reached is calculated from the difference between the maximum wear and the warning limit entered.

Disabled (G)

Individual tools can also be disabled manually if you no longer want to use them for workpiece machining.

Oversize (U)	In the case of oversize tools, neighboring magazine locations (left and right adjacent location) are only reserved alternately, i.e. you can only insert the next tool in the next magazine location but one. (This can also contain an oversize tool.)
Coded for fixed location (P)	You can assign tools to a fixed location, i.e., the tool can only be used in its present magazine location. After machining, the tool always returns to its old magazine location.

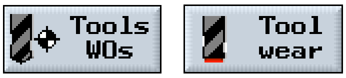
Monitoring tool use



- Select the "Tool wear" softkey in the "Tools WOs" operating area.
- Position the cursor on the tool that you want to monitor.
- In the column "T/C" select the parameter that you wish to monitor (T = Tool life, C = Count, W = Wear).
- Enter a prewarning limit for the tool life, count, or wear.
- Enter the scheduled service life for the tool, the scheduled number of workpieces to be machined or the maximum permissible wear.

The tool is disabled when the tool life, count, or wear is reached.

Entering tool statuses



- Select the "Tool wear" softkey in the "Tools WOs" operating area.
- Place the cursor on a tool.
- Select the option "G" in the first field of the last column if you want to disable the tool for machining.

-or-

- Select the option "U" in the second field of the last column if you want to mark the tool as oversize.

-or-

- Select the option "P" in the third field of the last column if you want assign the tool to a fixed magazine location.

The tool properties you have set become active immediately.

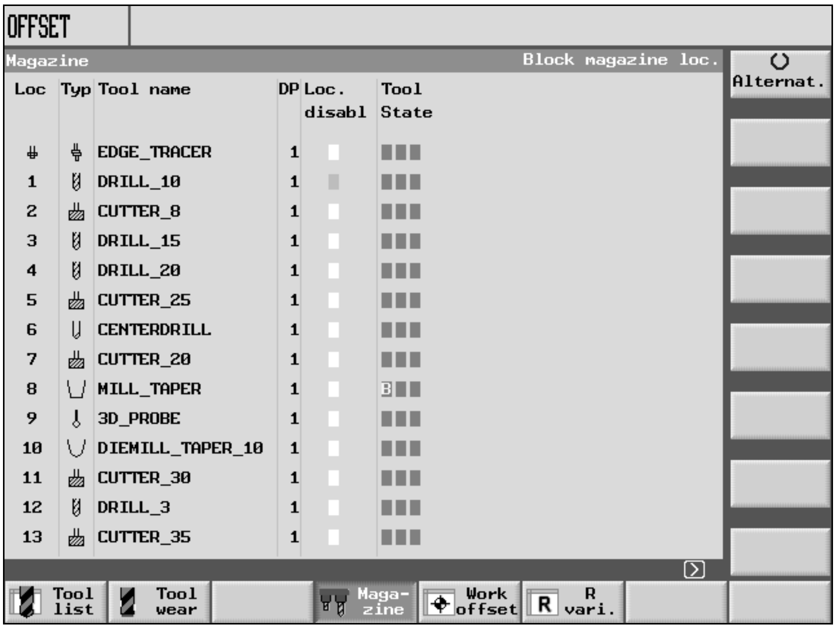
2.13.10 Magazine list

All magazine locations are listed in the magazine list. The list shows whether a magazine location is free, disabled, or occupied by a tool.

From the "Tool status" column, you can also see whether a tool is disabled (G) or oversize (U) or allocated to a fixed location.
You can change the tool status settings in the tool wear list (see Sec. "Activating tool monitoring")-
If a magazine location is defective, or an oversize tool requires more than half the adjacent location, you can disable the magazine location to code a tool for the fixed location. It is no longer possible to assign any tool data to a disabled magazine location.

Tools WOs | Maga- zine

➢ Select the "Magazine" softkey in the "Tools WOs" operating area.



Example of magazine with variable assignment

Disabling a machine location

Magazine locations can be reserved or disabled for tools, e.g. for oversized tools.

➢ Place the cursor on the relevant empty magazine location in the "Location disable" column.

Alternat.

➢ Press the "Alternat." softkey until a "G" (=disabled) appears in the field.

The location is now disabled. A tool can no longer be loaded into this magazine location.

Tool status

In the column "Tool status", you can see which properties have been assigned to the active tool:

- G: Tool is disabled
- U: Tool oversized
- P: Tool at a fixed location

2.13.11 Deleting a tool

Tools can be deleted from the tool list.

➢ Select the tool of your choice with the cursor keys.

Delete tool

➢ Press the "Delete tool" softkey.

➢ Confirm with "Delete".

The tool data for the selected tool are deleted. The magazine location in which the tool was stored is enabled.

2.13.12 Changing the tool type

In the tool list you can change a tool type into another tool type.

➢ Select a tool with the cursor keys and place the cursor on the "Type" input field.

Alternat.

➢ Press the "Alternat." softkey until the tool type you are looking for appears.

The input fields for the new tool type are displayed.

2.13.13 Loading/unloading a tool into/out of the magazine

You can unload tools in the magazine that you are not using at present. ShopMill then automatically saves the tool data in the tool list outside the magazine. Should you want to use the tool again later, simply load the tool with the tool data into the corresponding magazine location again. Then the same tool data does not have to be entered more than once.

Loading and unloading of tools into and out of magazine locations must be enabled in a machine data code.

Please refer to the machine manufacturer's instructions.

When you are loading a tool, ShopMill automatically suggests an empty location. The magazine in which ShopMill searches for an empty location first is stored in a machine data code.

Please also refer to the machine manufacturer's instructions.

You can also specify an empty magazine location directly when loading a tool, or define the magazine ShopMill should search for an empty location.

If your machine has only one magazine, you simply need to enter the location number you require when loading the tool, not the magazine number.

You can also attach or detach a tool to or from the spindle directly. You can disable loading and unloading with machine data.

Please refer to the machine manufacturer's instructions.

Loading a tool into the magazine

Tools WOs | Tool list

- Select the "Tool list" softkey in the "Tools WOs" operating area.
- Place the cursor on the tool that you want to load into the magazine (if the tools are sorted according to magazine location number you will find it at the end of the tool list).

Load

- Press the "Load" softkey.

The "Empty location" window appears. The "Location" field is initialized with the number of the first empty magazine location.

OK

- Press the "OK" softkey to load the tool into the suggested location.

-or-

OK

➢ Enter the location number you require and press the "OK" softkey.

-or-

Spindle OK

➢ Press the "Spindle" and "OK" softkeys to load a tool into the spindle.

The tool is loaded into the specified magazine location.

Finding an empty location in the magazine and loading the tool

Tools WOs Tool list

➢ Select the "Tool list" softkey in the "Tools WOs" operating area.

➢ Place the cursor on the tool that you want to load into the magazine.

Load

➢ Press the "Load" softkey.

The "Empty location" window opens. The "Location" field is initialized with the number of the first empty magazine location.

➢ Enter the magazine number and a "0" for the location number if you wish to search for an empty location in a particular magazine.

-or-

➢ Enter a "0" for the magazine number and location number if you wish to search for an empty location in all magazines.

OK

➢ Press the "OK" softkey.

An empty location is suggested.

OK

➢ Press the "OK" softkey.

The tool is loaded into the suggested magazine location.

Unloading an individual tool from the magazine

Tools WOs Tool list

➢ Select the "Tool list" softkey in the "Tools WOs" operating area.

➢ Position the cursor on the tool that you want to unload.

Unload

➢ Press the "Unload" softkey.

The tool is unloaded from the magazine.

Unloading all tools from the magazine

Tools WOs | Maga-zine

➢ Select the "Magazine" softkey in the "Tools WOs" operating area.

Unload all | Unload

➢ Press the "Unload all" and "Unload" softkeys.

All tools are unloaded from the magazine.

Abort

You can abort the unloading process at any time by pressing the "Abort" softkey. The current tool is unloaded and then the process is aborted.
The unloading process is also aborted if you exit the magazine list.

2.13.14 Relocating a tool

Tools can be relocated within magazines or between different magazines, which means that you do not have to unload tools from the magazine in order to load them into a different location.

ShopMill automatically suggests an empty location to which you can relocate the tool. The magazine in which ShopMill searches for an empty location first is stored in a machine data code.

Please refer to the machine manufacturer's instructions.

You can also specify an empty magazine location directly or define the magazine that ShopMill will search for an empty location.

If your machine has just one magazine, you only need to enter the location number you require, not the magazine number.

If a spindle location is shown in the tool list, you can also attach or detach a tool directly to or from the spindle.

Please refer to the machine manufacturer's instructions.

Specifying an empty location

Tools WOs | Maga-zine

➢ Select the "Magazine" softkey in the "Tools WOs" operating area.

➢ Place the cursor on the tool that you wish to relocate to a different magazine location.

Relocate

➢ Press the "Relocate" softkey.

The "Empty location" window appears. The "Location" field is initialized with the number of the first empty magazine location.

OK

➢ Press the "OK" softkey to relocate the tool to the suggested location.

-or-

OK

➢ Enter the location number you require and press the "OK" softkey.

-or-

Spindle OK

➢ Press the "Spindle" and "OK" softkeys to load a tool into the spindle.

The tool is relocated to the specified magazine location.

Finding an empty location

Tools WOs Maga-zine

➢ Select the "Magazine" softkey in the "Tools WOs" operating area.

➢ Place the cursor on the tool that you wish to relocate to a different magazine location.

Relocate

➢ Press the "Relocate" softkey.

The "Empty location" window appears. The "Location" field is initialized with the number of the first empty magazine location.

➢ Enter the magazine number and a "0" for the location number if you wish to search for an empty location in a particular magazine.

-or-

➢ Enter a "0" for the magazine number and location number if you wish to search for an empty location in all magazines.

OK

➢ Press the "OK" softkey.

An empty location is suggested.

OK

➢ Press the "OK" softkey.

The tool is relocated to the suggested magazine location.

2.13.15 Positioning a location

You can position magazine locations directly on the loading point.

Positioning a magazine location

Tools WOs | Maga-zine

➢ Select the "Magazine" softkey in the "Tools WOs" operating area.

➢ Place the cursor on the magazine location that you want to position on the loading point.

Position-ing

➢ Press the "Position" softkey.

The magazine location is positioned on the loading point.

2.13.16 Sorting tools

When you are working with large magazines or several magazines, it is useful to display the tools sorted according to different criteria. Then you will be able to find a specific tool more easily in the lists.

Tools can be sorted in the tool list or tool wear list according to magazine assignment, tool name (alphabetic), tool type, or numerically according to T number. When you sort according to magazine assignment, the empty locations in the magazine are also displayed.

Tools WOs | Tool list

-or-

Tools WOs | Tool wear

➢ "Select the "Tool List" or "Tool wear" softkey in the "Tool WOs" operating area.

The tool list or tool wear list opens.

➢ Press the "Sort" softkey.

A new vertical softkey menu is displayed.

to magazine or
to name or
to type or
to T number

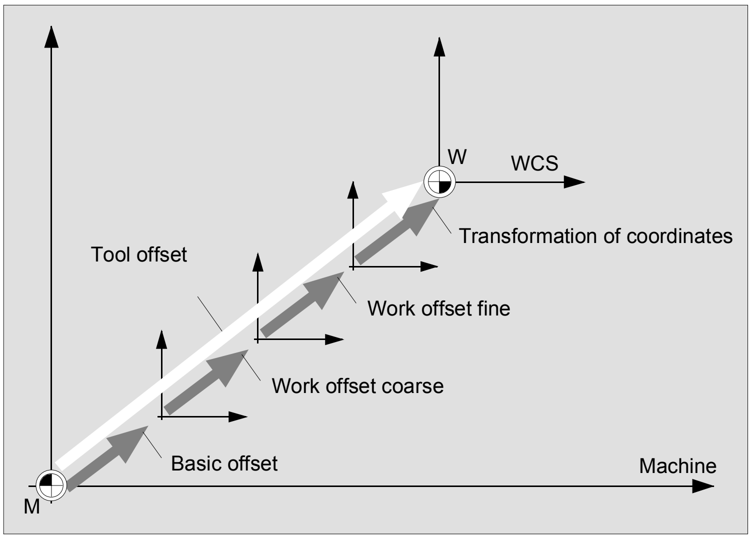
➢ Activate one of the softkeys to choose the sort criteria for the tools.

The tools are listed in the new order.

2.14 Work offsets

Following reference point approach, the actual value display for the axis coordinates is based on the machine zero (M) of the machine coordinate system (MCS = machine). The program for machining the workpiece, however, is based on the workpiece zero (W) of the workpiece coordinate system (WCS = work).
The machine zero and workpiece zero are not necessarily identical. The distance between the machine zero and workpiece vary in accordance with the type of tool and how it is clamped. This work offset is taken into account during execution of the program and can be a combination of different offsets.
In ShopMill, the position actual value display refers to the settable zero system. The position of the active tool relative to the workpiece zero is displayed.

The offsets are summated as follows:



Work offsets

When the machine zero is not identical to the workpiece zero, at least one offset (base offset or work offset) exists in which the position of the workpiece zero is saved.

Base offset

The base offset is a work offset that is always active. If you have not defined a base offset, its value will be zero. You determine the base offset via "Workpiece zero" (see Sec."Measurement workpiece zero") or "Set work offset" (see Sec. "Setting a new position value").

Work offsets

Every work offset (G54 to G57, G505 to G599) consists of a coarse offset and a fine offset. You can call the work offsets from any sequential control program (coarse and fine offsets are added together).
You can save the workpiece zero, for example, in the coarse offset, and then store the offset that occurs when a new workpiece is clamped between the old and the new workpiece zero in the fine offset.

Fine offsets must be set up by the machine manufacturer.

Please refer to the machine manufacturer's instructions.

For instructions on specifying and calling work offsets, see Secs. "Defining work offsets" and "Calling work offsets".

Coordinate transformations

You always program coordinate transformations for a specific sequential control program. They are defined by:

- Offset
- Rotation
- Scaling
- Mirroring

(See Sec. "Defining the coordinate transformations")

Total offset

The total offset is calculated from the sum of all offsets and coordinate transformations.

2.14.1 Defining work offsets

Enter work offsets (coarse and fine) directly in the work offset list.

Fine offsets must be set up by the machine manufacturer.
The number of possible work offsets is defined by a machine data.

Please refer to the machine manufacturer's instructions.

Tools WOs | Work offset

➢ Press the "Work offset" softkey in the "Tools WOs" operating area.

The work offset list appears.

➢ Position the cursor on the coarse or fine offset that you wish to define.

➢ Enter the desired coordinates for the axis in question You can use the cursor keys to switch between axes.

-or-

Set X ... Set Z

➢ Press the "Set X", "Set Y" or "Set Z" softkey to accept the position value of an axis from the position display for a coarse offset.

-or-

Set all

➢ Press the "Set all" softkey to accept the position values of all axes from the position display for a coarse offset.

The new coarse offset is set. The values from the fine offset are included in the calculation and then deleted.

Delete WO

➢ Press the "Delete WO" softkey to delete the coarse and fine offset values at the same time.

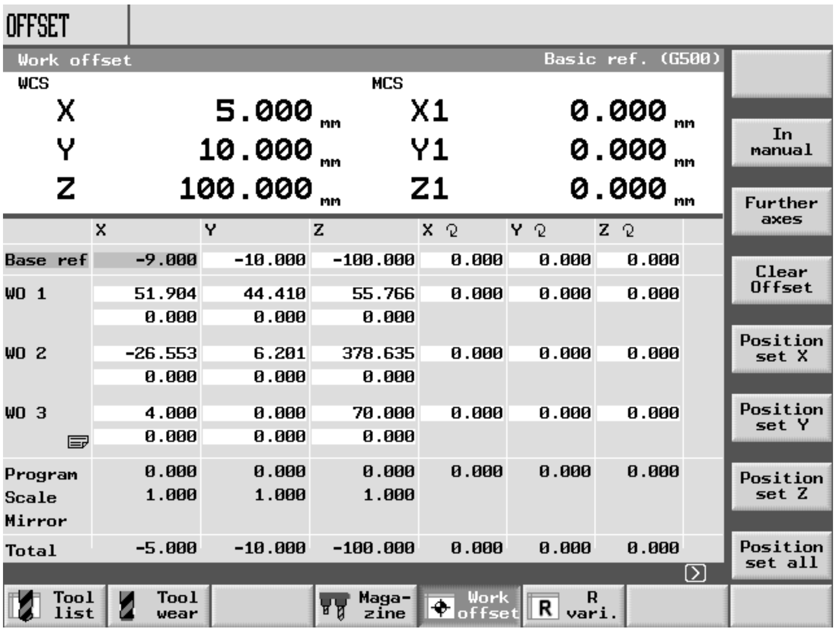
Additional axes

With the "Additional axes" softkey, you can display two additional rotary axes and determine their offset. These additional axes must be activated via machine data.

Please refer to the machine manufacturer's instructions.

2.14.2 Work offset list

The individual work offsets as well as the total offset are all displayed in the work offset list. The currently active work offset is displayed on a gray background. The work offset list also includes the current axis positions in the machine and workpiece coordinate systems.



Work offset list

Base offset

Basic reference — The coordinates of the base offset appear. You can change these here in the list.

Work offsets

NPV1 ... NPV3 — The coordinates of the individual work offsets (1st line Coarse offset, 2nd line Fine offset) and the angle with which the coordinate system may be rotated about an axis if necessary, are listed. You can change this data here in the list (see Sec. "Defining work offsets").
Fine offsets must be set up by the machine manufacturer.

Please refer to the machine manufacturer's instructions.

PAGE DOWN

You can display more work offsets with the "Page Down" key.

Coordinate transformations

Program	The active coordinates of the "Offset" transformation are displayed as well as the angle set in the "Rotation" transformation by which the coordinate system rotates. You cannot edit these values here.
Scale	The active scaling factor for the "Scaling" transformation is displayed for the respective axis. You cannot edit these values here.
Mirror	The mirror axis that was defined by means of the "Mirroring" transformation is displayed. You cannot edit these values here.

Total offset

Total	The total offset resulting from the base offset and all active work offsets and coordinate transformations appears.
Additional axes	With the "Additional axes" softkey, you can display two additional rotary axes and determine their offset. These additional axes must be activated via machine data. Please refer to the machine manufacturer's instructions.

➢ Press the "Work offset" softkey in the "Tools WOs" operating area.

The work offset list appears.

2.14.3 Selecting/deselecting the work offset in the Manual area

Jog T,S,M

➢ Select the "T, S, M" softkey in "Machine Manual" mode.

The "T,S,M" window opens.

Selecting a work offset

Alternat.

➢ Position the cursor on parameter "Work offset" and press the "Alternat." softkey until the work offset you require appears.

Cycle Start

➢ Press the "Cycle Start" key.

The set work offset is activated.
The active work offset is also displayed in the "WCS" window.
e.g. WO1

The offset values entered in the "Work offset" menu are also taken into account in the workpiece coordinate system display **WCS** (work).

Deselecting the work offset

Alternat.

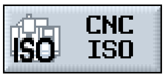
➢ Place the cursor on parameter "Work offset" and press the "Alternat." softkey until " - " appears in the field.

Cycle Start

➢ Press the "Cycle Start" key.

An active work offset is deselected.

2.15 Switching to CNC-ISO mode

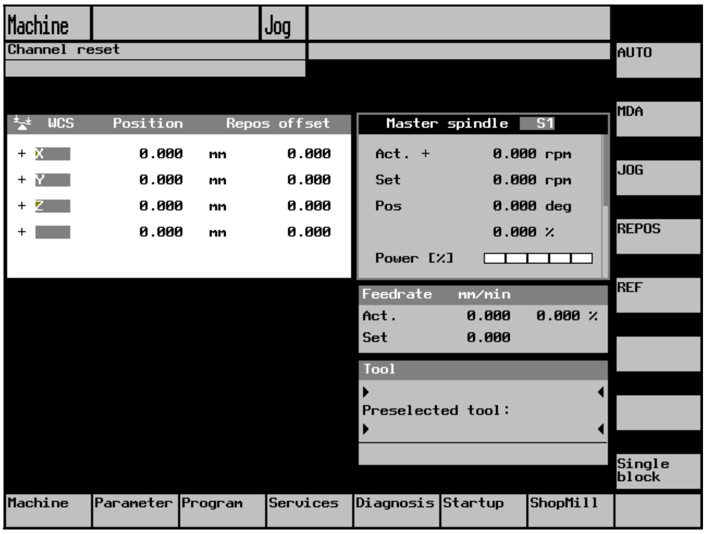


You can switch from the ShopMill interface to the CNC ISO standard operator interface of the SINUMERIK 840D or 840D/840Di/810D system by pressing the "CNC ISO" softkey.

Softkey "CNC ISO" will work only if the machine manufacturer has implemented the function internally via the PLC interface.

Please refer to the machine manufacturer's instructions.

If the "CNC ISO" softkey is active, the following basic display of the CNC ISO operator interface appears on your screen:



ShopMill

If you wish to return to the ShopMill operator interface, press the "ShopMill" softkey.

If you are working in the CNC ISO operator interface, please **read** the User Documentation for the SINUMERIK 840D/840Di/810D system (see Appendix, List of References).

2.16 ShopMill Open (PCU 50)

The ShopMill software is available for the PCU 50 in two versions, ShopMill Classic and ShopMill Open. ShopMill Classic is the software package previously marketed under the name ShopMill.

The difference between ShopMill Open and ShopMill Classic is that the Open variant has a basic menu bar or extended basic menu bar. ShopMill Open does not offer the option of switching to the CNC ISO user interface. Instead, the HMI Advanced operating areas "Services", "Diagnosis", "Start-up", and "Parameters" (without tool management and work offsets), are located directly on the extended horizontal softkey bar.

Services	Diagnosis	Start-Up	Parameters				

For a detailed description of the integrated HMI Advanced operating areas, please refer to:
References: /BAD/, Operator's Guide HMI Advanced SINUMERIK 840D/840Di/810D

Some of the softkeys in the basic menu or extended menu bars may be assigned to other operating areas by the machine manufacturer.

Please refer to the machine manufacturer's instructions.

2.17 Remote diagnostics

The control system can be operated from an external PC by means of a remote diagnostic function. You can use a modem to link the control system and the external PC.

Remote diagnosis

The remote diagnostics function is activated on the CNC ISO operator interface in the Diagnosis operating area.

Remote Diagnosis is a software option.

For further information about remote diagnosis, please refer to:
References: /FB/, Description of Functions Extended Functions, F3

■

Programming with ShopMill

3.1 Basics of programming

Important

Please pay particular attention to the following fundamental principles when writing programs for your machine tool!

Axes

The 3 main axes on milling machines are designated as X, Y and Z. Axis Z is normally the tool axis.

Dimensions in metric or inch

The control system can process both metric and inch dimensions. Depending on the basic setting you choose, the control interprets all geometric values as either metric or inch dimensions.
Irrespective of the basic setting, you can set metric or inch dimensions in the program header (define blank).
All dimensions stated in this section are metric.

Absolute dimensioning

With the absolute dimensioning method, dimensions refer to the zero point of the coordinate system of the total offset.

Incremental dimensioning

With the incremental dimensioning method, the programmed positional numerical value corresponds to the path to be traversed. The sign indicates the traversing direction.

Tool T

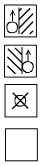
A tool must be programmed for every cutting operation. With the ShopMill machining cycles, a tool selection is already integrated in every parameterization screenform. Exception: You must select a tool before programming simple straight lines and circles.
Tool selection is modal with straight line/circle, i.e. if several machining steps with the same tool occur in succession, you only need to program the tool for the 1st straight line/circle.

Tool length compensation

Tool length compensations take effect immediately the tool is loaded into the spindle. Different tool offsets can be assigned to each tool with multiple cutting edges.
The tool length compensation of the spindle tool remains active even after the program has been executed (RESET).

Tool radius compensation

The tool radius compensation is automatically included in the cycles except for path milling. You can machine with or without radius compensation in conjunction with the "Path milling" and "Line" functions. In the case of the "Line" function, the tool radius compensation has a modal action, i.e. it is not automatically deactivated again.



Radius compensation left of contour

Radius compensation right of contour

Radius compensation off

Radius compensation is retained as set

Spindle speed

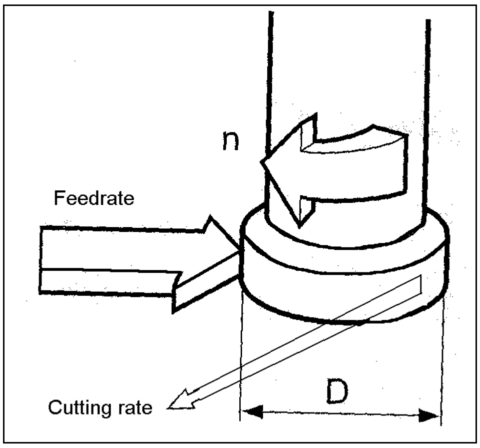
The spindle speed (S) determines the number of spindle rotations per minute. The CW/CCW setting is made in the tool list in ShopMill.

Programming:
The spindle speed is input when a new tool is loaded into the spindle. As an alternative to spindle speed, a cutting rate (V) can be specified in m/min.

Spindle start/spindle stop:
The spindle is started directly after a new tool has been loaded. It is stopped on Reset, end of program or tool change.

Cutting rate

Peripheral speed at which the tool cutting edge machines the workpiece. Cutting rates (V) are specified in m/min.



Cutting rate

Traversing at rapid traverse

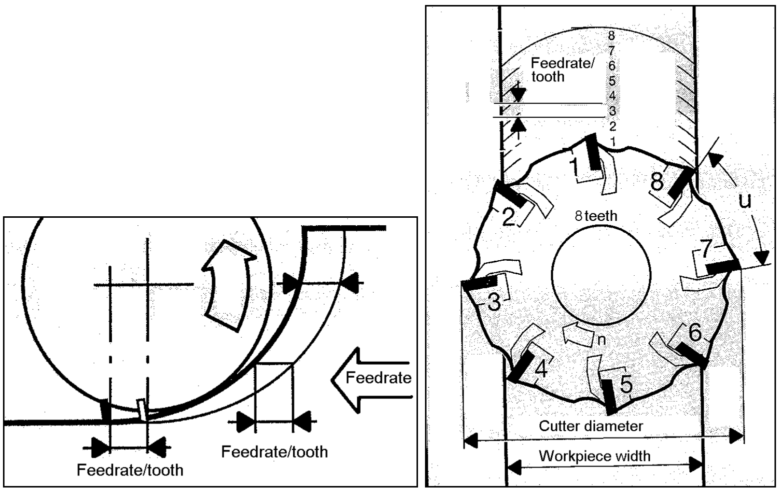
The programmed path is traversed along a straight line at the fastest possible velocity without the workpiece being machined. Rapid traverse is a non-modal command, i.e. if you want the axis to traverse rapidly in the next block, then you must enter "Rapid traverse" as feedrate (F) again.
If you do not program a feedrate or rapid traverse, the axis is automatically traversed at the last programmed feed value (machining feedrate).

Traversing at feedrate (machining feedrate)

The tool travels at the programmed feedrate F along a straight line or on a circle to the programmed end point and then machines the workpiece. Machining feedrates (F) are specified in mm/min, mm/rev or mm/tooth. The feedrate for milling cycles is automatically converted on switchover from mm/min to mm/rev and vice versa.

With milling cycles, the feedrate for rough cutting is relative to the milling tool center point. This also applies to finish cutting, with the exception of concave curves where the feedrate is relative to the cutting edge (contact point between milling tool and workpiece).

Feedrate in mm/tooth
Mills are multi-edged tools. For this reason, a value must be found which guarantees that each cutting edge can machine the workpiece under the best possible conditions. Feed per tooth corresponds to the linear path traversed by the mill when a tooth is engaged. Feed per tooth is also the effective distance covered by the table feed between the engagement of two successive cutting edges.

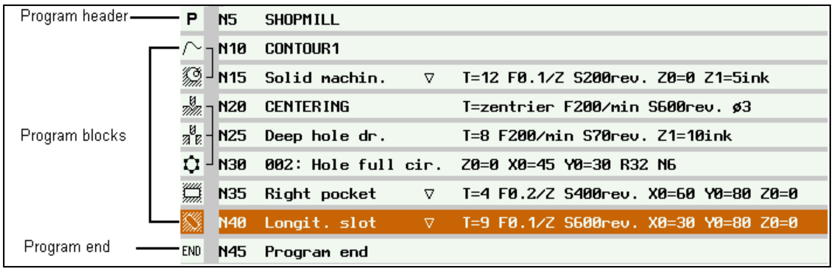


Feedrate in mm/tooth

The machining feedrate is modal, i.e. even if the machining process changes, you need not enter a new feedrate if the feedrate programmed in the preceding block is still appropriate. This applies even if you have programmed a rapid traverse command in between.

3.2 Program structure

The program is divided into 3 subsections:
Program header, program blocks and program end.
These sub-areas form a machining plan.



Program structure

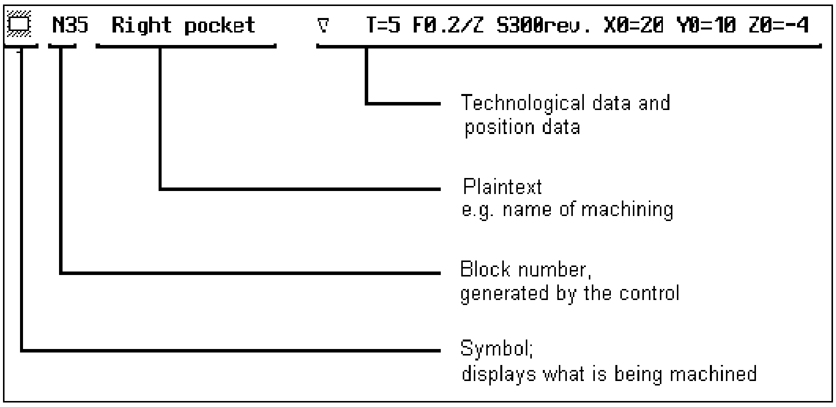
Program header

The program header contains the dimensions of the blank and parameters which are effective throughout the program, e.g.

- dimension in mm or inches
- tool axis X, Y or Z
- retraction plane, safety clearance, machining direction

Program blocks

To obtain a finished part, you must first program the various machining operations, travel motions, machine commands, etc. The programming is performed in the program blocks.



Chained machining

The control automatically chains the technology and position blocks. These blocks are identified by a square bracket immediately beside the machining symbol. The brackets are inserted from the beginning to the end of the sequence of chained blocks.

Blocks N10 to N25 are chained

3.3 Creating a sequential control program

For sequential control programs that you create directly at the machine, you require a software option.

3.3.1 Creating a new program; defining a blank

New programs are set up in the "Program Manager" area.

Select with softkey

Pro-gram | Programs | New > | ShopMill Program

Entering the program name

Enter a program name.
Program names may be a maximum of 24 characters in length. You can use any letters, digits or the underscore symbol (_). ShopMill automatically changes lower case to upper case.

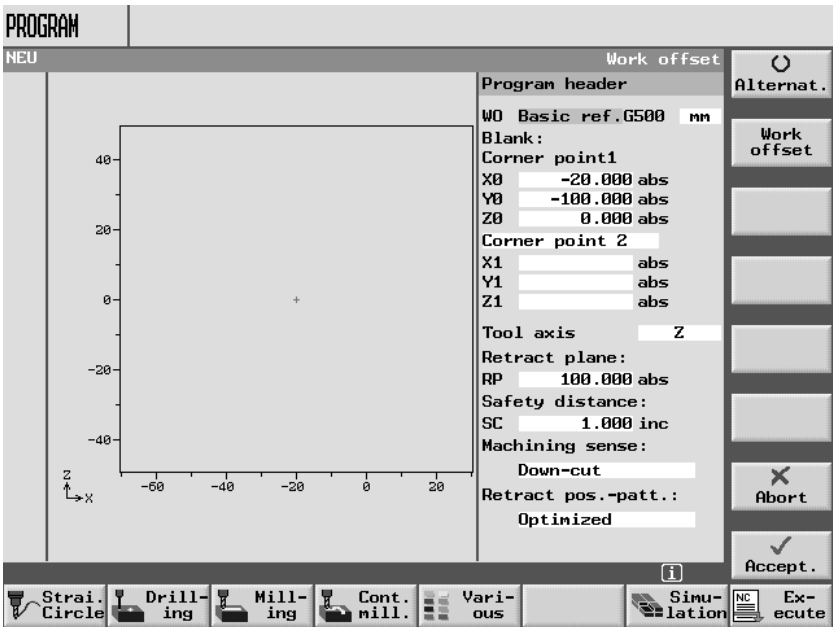
"Periods" are not permitted in program names because such programs cannot be deleted again.

Confirm the program name by pressing

the OK softkey or with the "Input" key INPUT .

The screenform for setting the "Program header" parameters then appears.

Parameterizing the program header

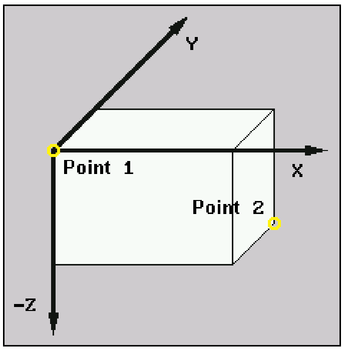


Set program header parameters

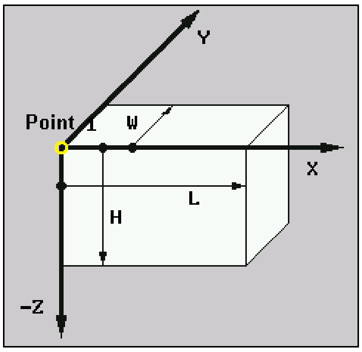
Parameter settings in the program header are valid throughout the entire program.

Parameters for input of a blank

- **Work offset (WO)** in which the workpiece zero is stored. You can select the work offset with the "Work Offset" softkey in the tool list or delete the default setting of the parameter if you do not want to state a work offset.
- Define the **unit of measurement** for the program [mm or inch].
- **Workpiece corner point 1** (X0, Y0, Z0):
  Workpiece corner point 1 is the reference point for the blank dimensions. It must be entered as an absolute value.
- **Workpiece corner point 2 or dimensions**
  (X1, Y1, Z1 or L, W, H):
  Workpiece corner point 2 is opposite workpiece corner point 1. It must be entered as an absolute value. The deviations are the length, width and height of the blank.

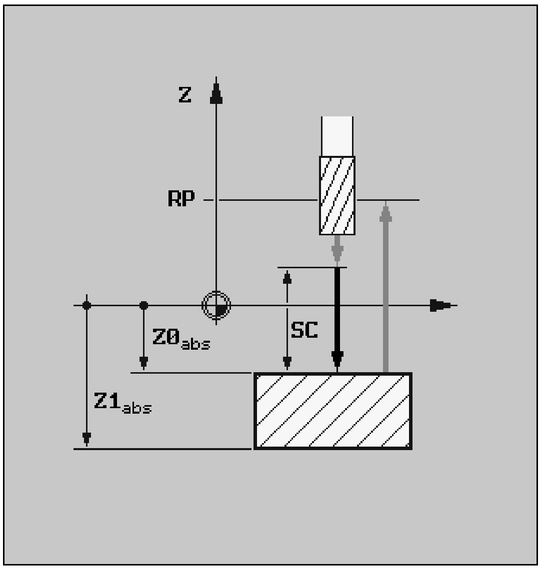


Workpiece corner points 1 and 2

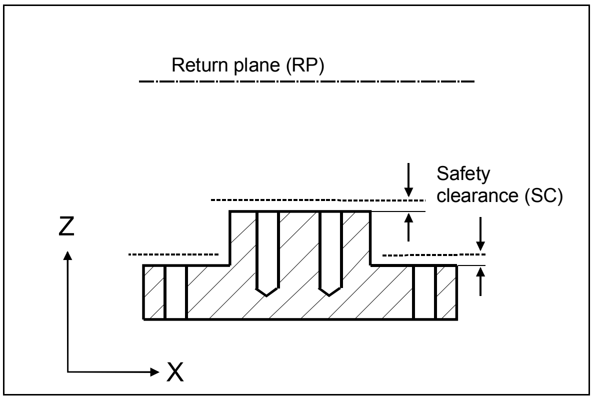


Workpiece corner point 1 and dimensions

- **Tool axis:** The tool length is calculated in the set axis.
- **Retraction plane (RP) and safety clearance (SC):**
  Planes above the workpiece.
  During machining the tool travels in rapid traverse from the tool change point to the return plan and then to the safety clearance. The machining feedrate is activated at this level. When the machining operation is finished, the tool travels at machining feedrate away from the workpiece onto the safety clearance level. It travels from the safety clearance to the retraction plane and then to the tool change point in rapid traverse.
  The retraction plane is entered as an absolute value.
  The safety clearance must be entered as an incremental value (without sign).

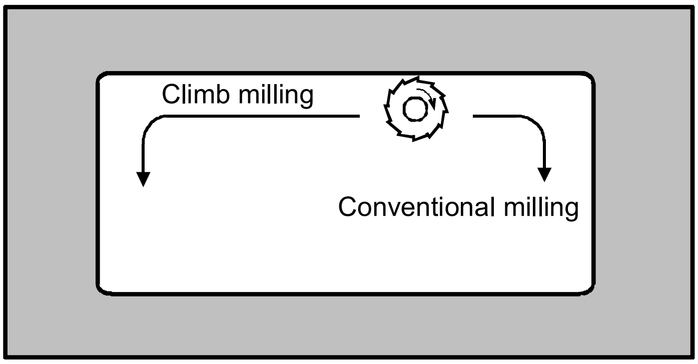


Retraction plane (RP) and safety clearance (SC)



Safety clearance for varying workpiece heights

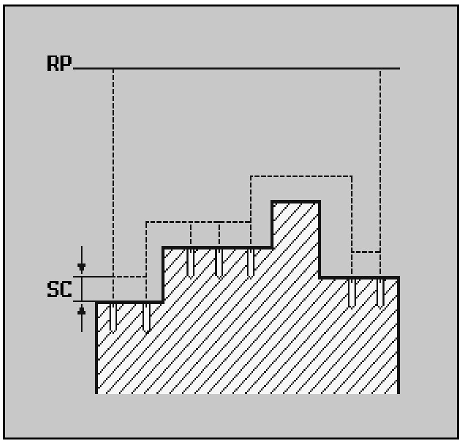
- **Machining direction**:
  When machining a pocket, a longitudinal slot, or a spigot, ShopMill takes the machining direction (down-cut or up-cut) and the spindle direction in the tool list into account. The pocket is then machined in a clockwise or counterclockwise direction.



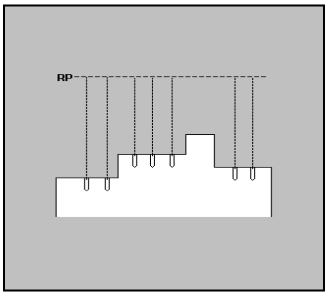
Machining pockets in climb or conventional milling operation with CW spindle rotation

During path milling, the programmed contour direction determines the machining direction.

- **Retraction with position patterns:**
  When working with **optimized retraction**, the tool travels in relation to the contour across the workpiece at machining feedrate and safety clearance (SC). On retraction **to RP**, the tool is retracted to the retraction plane when the machining step is complete and infeeds at the new position. Collisions with workpiece obstacles are thus prevented when the tool is retracted and fed in, e.g. when holes in pockets or grooves are machined at different levels and positions.

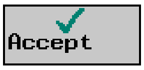


Optimized retraction



Retraction to retraction plane RP

Storing parameters

Select with the 

softkey.
The parameters you have entered are stored. The machining plan is then displayed.

Program end

ShopMill has automatically defined the program end.

3.3.2 Programming new blocks

Creating new program blocks

Once you have defined the blank, you can define machining operations, feedrates and positions in individual program blocks. You will be supported by "Help" displays for individual machining operations.

A large amount of memory is available for one program.

However, depending on the storage space required, you can only program a limited number of blocks.

- **PCU 20**
  You can program up to 1000 blocks with the "Straight" function or up to 600 blocks with the "Mill pocket" function.
- **PCU 50**
  You can program up to 3500 blocks with the "Straight" function or up to 2100 blocks with the "Mill pocket" function.

In the case of multiple clampings, a program can easily contain more than the permissible number of program blocks. If a message tells you that too many blocks are present, group together machining operations with the same tool in a subroutine. That way, you can open and execute the program.

Note

New programming blocks are always inserted **after the** selected block. You cannot program blocks before the program header or after program end.

Parameter input fields

Feedrate:
If you do not program a value for feedrate (F) (empty field), the system uses the last programmed feedrate.

Clear an input field:
Use the DEL key (or Backspace key) to clear an input field, i.e. to delete the programmed value.

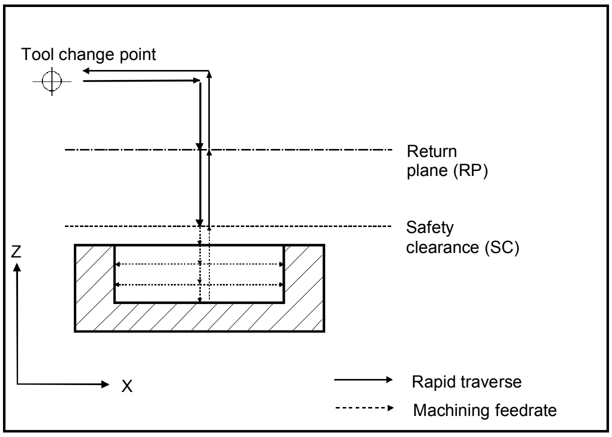
Preset (default) or empty parameter fields:
You must always enter a value in fields with a preset default. If you clear a default field, the "Accept" softkey disappears from the display!

"Alternat." softkey and toggle key:
If the cursor is positioned on an input field with various setting options, the "Alternat." softkey is automatically displayed on the vertical softkey bar (see "Alternat." softkey in Sec. "Important softkeys for operation and programming").

Roughing/finishing:
Every cycle can be programmed with roughing or finishing. If you wish to rough cut the workpiece first and then finish cut it, you must call the cycle a second time. The programmed values do not change if you call the cycle again.
Some cycles offer roughing and finishing as a **complete machining** operation, i.e. you need only call the cycle once.

Approaching a cycle

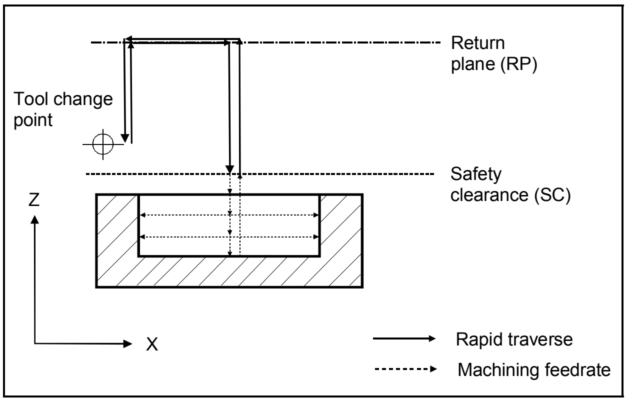
- Approach the programmed cycles with ShopMill
  - Tool is above the retraction plane (RP):
    Positioning of the tool is performed at rapid traverse in the X/Y plane and then in the Z direction to the retraction plane (RP)



Approach to cycle above the retraction plane

  - or tool is below the retraction plane (RP):

Positioning of the tool is performed at rapid traverse first in the Z direction to the retraction plane (RP) and then at rapid traverse in the X/Y plane



Approach to cycle below the retraction plane

- Tool axis travels in rapid traverse to safety clearance (SC)
- The cycle is then processed at the programmed machining feedrate
- On completion of machining, the tool travels to the cycle center in the X/Y plane at machining feedrate and then moves away from the workpiece with the tool axis until it reaches safety clearance
- The tool axis then retracts to the retraction plane in rapid traverse
- The tool change point is approached from the retraction plane in rapid traverse

3.3.3 Changing program blocks

You can optimize the parameters in the programmed ShopMill blocks later or adapt them to new situations, e.g. if you want to increase the feedrate or change a position. In this case, you can change all the parameters in every program block directly in the associated parameterization screen form.

Pro-
gram

➢ Press the "Program" softkey.

The directory overview is displayed.

➢ Place the cursor on the directory that contains the program that you want to open.

-or- INPUT

➢ Press the "Input" or "Cursor Right" key.

All the programs in this directory are displayed.

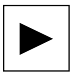
➢ Select the program that you want to change.

-or- INPUT

➢ Press the "Input" or "Cursor Right" key.

The machining plan of the program is displayed.

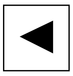
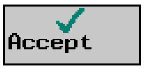
- Place the cursor on the desired program block in the machining plan.



- Press the "Cursor Right" key.

The parameter screen for the selected program block appears.

- Make the desired changes.


-or-


- Press the "Accept" softkey or the "Cursor left" key.

The changes are accepted in the program.

3.3.4 Program editor

You use the program editor when you want to change the sequence of program blocks within a program, delete program blocks or copy program blocks from one program to another.

The following functions are available in the program editor:

- Select
  You can select several program blocks simultaneously, for example, for cutting and pasting them subsequently.
- Copy/paste
  You can copy and paste program blocks within a program or between different programs.
- Cut
  You can cut and therefore delete program blocks. However, the program blocks remain in the buffer, so you can still paste them in somewhere else.
- Search
  You can search for a specific block number or any character string in a program.
- Rename
  You can rename a contour in the program editor, e.g. if you have copied the contour.
- Number
  If you insert a new or copied program block between two existing program blocks, ShopMill automatically generates a new block number. This block number may be higher than the one in the following block. You can use the "Numbering" function to number the program blocks in ascending order.

Opening the program editor

- Select a program.

>

- Press the "Expansion" key.

The softkeys for the program editor are displayed in the vertical softkey bar.

Selecting a program block

- Place the cursor in the machining plan on the first or last block you want to select.

Mark

- Press the "Mark" softkey.
- Use the cursor keys to select any further program blocks.

The program blocks are marked.

Copying a program block

- Select the program block(s) in the machining plan.

Copy

- Press the "Copy" softkey.

The program blocks are copied into buffer memory.

Cutting a program block

- Select the program block(s) in the machining plan.

Cut

- Press the "Cut" softkey.

The program blocks are removed from the machining plan and stored in buffer memory.

Pasting a program block

- Copy or cut the desired program blocks in the machining plan.
- Place the cursor on the line after which the program block(s) is (are) to be inserted.

Insert

- Press the "Insert" softkey.

The program blocks are inserted in the machining plan of the program.

Search

Search >

- Press the "Search" softkey.
- Enter a block number or text.
- Select whether the search is to commence at the start of the program or the current cursor position.

Search >

- Press the "Search" softkey.

ShopMill searches the program. The cursor highlights the search hit.

Continue search

- Press the "Continue search" softkey if you want to continue the search.

Renaming a contour

- Place the cursor on a contour in the machining plan.

Rename

- Press the "Rename" softkey.
- Enter a new name for the contour.
- Press the "OK" softkey.

OK

The name of the contour is changed and displayed in the machining plan.

Numbering program blocks

Renumber

- Press the "Renumber" softkey.

The program blocks are renumbered in ascending order.

Closing the program editor

Back

- Press the "Back" softkey to close the program editor.

3.4 Programming the tool, offset value and spindle speed

General information

When you program cycles, you will find the tool displayed in the screenform. When you program a line or a circular arc, you will have to select a tool beforehand.

Select with softkey: Strai. Circle | Tool

Programming a tool (T)

Select parameter field "T". ShopMill allows you to enter tools in several different ways:

Method 1: Enter the name or number of a tool via the keyboard.

Method 2: Press area the "Tool, offset" key, select a tool with the cursor keys and press the "to program" softkey.
The tool is copied into the parameter field.

Cutting edge (D)

You can select/specify for each programmed tool whether you want to apply cutting edge offset values D. The offsets are stored in the tool list.
You must program the correct tool edge number D for the different tools (counterbore with spigot, stepped drill, etc.) to avoid risk of collisions (see also Secs. "Programming examples for drilling" and "Tools and tool offsets").

Spindle speed (S) or Cutting rate (V)

In ShopMill you can program either the spindle speed (S) or the cutting rate (V). You can toggle between them using the "Alternat." key. In the milling cycles, the spindle speed is automatically converted to the cutting rate and vice versa.

- Spindle speed and cutting rate remain valid until you program a new tool.
- Spindle speeds are programmed in rev/min.
- Cutting rates are programmed in m/min
- You can set the direction of rotation of a tool in the tool list.

Allowance (DR)

You can program an allowance on the tool radius in this parameter input field. A finishing allowance is then left when the contour is machined (see also Sec. "Tools and tool offsets").
The DR parameter is active until a ShopMill cycle (drilling, missing, contour missing) is programmed.

Example

You want to leave a finishing allowance of 0.5 mm on a contour. DR must then be programmed with 0.5 mm.
With a setting of DR=0, the programmed contour is cut without a finishing allowance.

3.5 Contour milling

The "Contour milling" function is used when you want to mill simple and complex contours. You can define open contours or closed contours (pockets, islands, spigots) and machine them with path milling or milling cycles.

A contour comprises separate contour elements, whereby at least two and up to 250 elements result in a defined contour. You can also program chamfers, radii or tangential transitions between the contour elements.

The integrated contour calculator calculates the intersection points of the individual contour elements taking into account the geometrical relationships, which allows you to enter incompletely dimensioned elements.

With contour milling, you must always program the geometry of the contour before you program the technology.
You have the option of machining contours of any type by path milling, stock removal from pockets with or without islands, or clearing spigots.

Freely-definable contours

The machining of freely-definable open or closed contours is generally programmed as follows:

1. Enter contour
   You build up the contour gradually from a series of different contour elements.
2. Path milling (roughing)
   The contour is machined taking into account various approach and retract strategies.
3. Path milling (finishing)
   If you programmed a finishing allowance for roughing, the contour is machined again.
4. Path milling (chamfer)
   If you have planned edge breaking, chamfer the workpiece with a special tool.

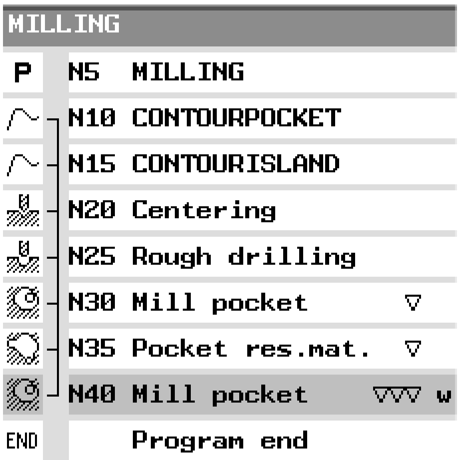
Contours for pockets or islands

Contours for pockets or islands must be closed, i.e. the start point and end point of the contour are identical. You can also mill pockets that contain one or more islands. The islands can also be located partially outside the pocket or overlap each other. ShopMill interprets the first contour specified as a pocket contour and all others as islands.

The machining of contour pockets with islands is generally programmed as follows:

1. Enter contour for the pocket
   You build up the contour pocket gradually from a series of different contour elements.
2. Enter contour for the island
   You enter the contour for the island after the contour for the pocket.
3. Centering predrilling of the contour pocket
   If you want to predrill the contour pocket, you can center the drill hole first to prevent the drill slipping.
4. Predrill contour pocket
   If you want the cutter to plunge into the material vertically and if a milling cutter with an end tooth is not available, you can predrill the pocket.
5. Remove stock from contour pocket with island (roughing)
   The stock is removed from the contour pocket complete with island taking into account various insertion strategies.
6. Remove residual material (roughing)
   During stock removal from the pocket, ShopMill automatically detects residual material that has been left. A suitable tool will allow you to remove this without having to machine the complete pocket again.
7. Finish contour pocket with island (finish edge/base)
   If you programmed a finishing allowance for the edge/base when you programmed roughing, the pocket edge/base will be machined again.

All machining steps involved in the contour milling operation are shown in the machining plan in square brackets.



Example: Removing stock from a contour pocket

Contours for spigots

Contours for spigots must be closed, i.e. the start point and end point of the contour are identical. You can define multiple spigots, which can also overlap. ShopMill interprets the first contour specified as a blank contour and all others as spigots.

The machining of contour spigots is generally programmed as follows:

1. Enter the blank contour
   i.e. the outer limits of the material. The tool moves at rapid traverse outside this area. Material is then removed between the blank contour and spigot contour.
2. Enter contour for the spigot
   You enter the contour for the spigot after the blank contour.
3. Clear contour spigot (roughing)
   The contour spigot is cleared.
4. Remove residual material (roughing)
   As it mills the spigot, ShopMill automatically detects residual material that has been left behind. A suitable tool will allow you to remove this without having to machine the complete spigot again.
5. Finish contour spigot (edge/base finishing)
   If you programmed a finishing allowance for roughing, the spigot edge/base is machined again.

3.5.1 Representation of the contour

ShopMill represents a contour as one program block in the machining plan. If you open this block, the individual contour elements are listed symbolically and displayed in broken-line graphics.

Symbolic representation

The individual contour elements are represented by symbols adjacent to the graphics window. They appear in the order in which they were entered.

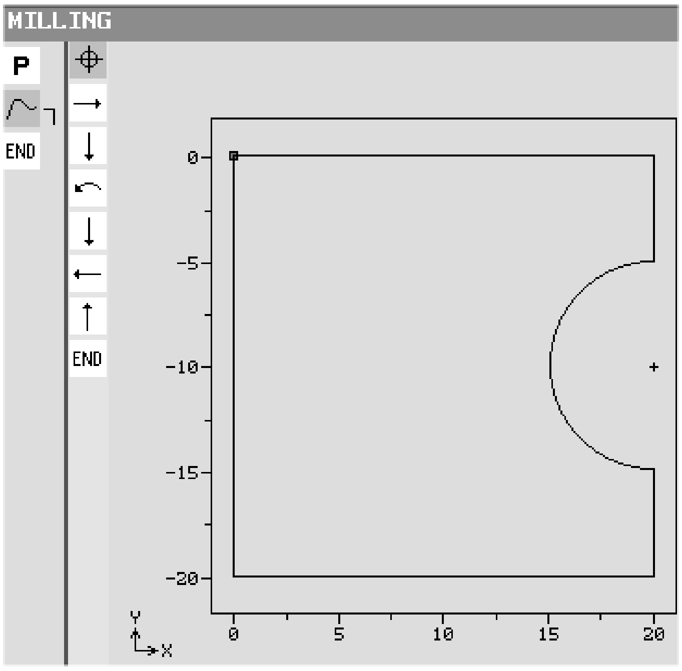
Contour element	Symbol	Meaning
Start point	⌖	Start point of contour
Straight line up	↑	Straight line in 90° matrix
Straight line down	↓	Straight line in 90° matrix
Straight line left	←	Straight line in 90° matrix
Straight line right	→	Straight line in 90° matrix
Straight line in any direction	↗	Straight line with any gradient
Arc Right	↷	Circle
Arc Left	↶	Circle
Finish contour	END	End of contour definition

The different color of the symbols indicates their status:

Foreground	Background	Meaning
-	red	Cursor on new element
black	red	Cursor on current element
black	white	Normal element
red	white	Element not currently evaluated (element will only be evaluated when it is selected with the cursor)

Graphical representation

The progress of contour programming is shown in broken-line graphics while the contour elements are being entered.



Graphical presentation of the contour during contour milling

When the contour element has been created, it can be displayed in different line styles and colors:

- Black: Programmed contour
- Orange: Current contour element
- Green dashed: Alternative element
- Blue dotted: Partially defined element

The scaling of the coordinate system is adjusted automatically to match the complete contour.

The position of the coordinate system is displayed in the graphics window.

3.5.2 Creating a new contour

For each contour that you want to mill, you must create a new contour.

The contours are stored at the end of the program.

The first step in creating a contour is to specify a starting point. ShopMill automatically defines the end of the contour.

If you alter the tool axis, ShopMill will automatically adjust the associated starting point axes.
You can enter any additional commands (up to 40 characters) in G code format for the start point.

If you want to create a contour that is similar to an existing contour, you can copy the existing one, rename it and just alter selected contour elements.
In contrast, if you want to use an identical contour at another place in the program, you must not rename the copy. Changes to the one contour will then automatically be applied to the other contour with the same name.

Cont. mill.

➢ Press the "Contour milling" and "New contour" softkeys.

New contour >

➢ Enter a name for the new contour.
The contour name must be unique.

OK

➢ Press the "OK" softkey.

The input form for the start point of the contour appears. You can enter Cartesian or polar coordinates.

Cartesian starting point

➢ Enter the starting point for the contour.

➢ Enter any additional commands in G code format, as required.

Accept

➢ Press the "Accept" softkey.

➢ Enter the individual contour elements (see Sec. "Creating contour elements").

Polar starting point

Pole

- Press the "Pole" softkey.
- Enter the pole position in Cartesian coordinates.
- Enter the starting point for the contour in polar coordinates.
- Enter any additional commands in G code format, as required.

Accept

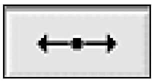
- Press the "Accept" softkey.
- Enter the individual contour elements (see Sec. "Creating contour elements").

Parameters	Description	Unit
Tool axis	Select Z as the tool axis, if the starting point/pole is in X / Y Select X as the tool axis, if the starting point/pole is in Y / Z Select Y as the tool axis, if the starting point/pole is in X / Z The coordinates also change for contour elements	
	Cartesian:	
X	Start point in X direction (abs.)	mm
Y	Start point in Y direction (abs.)	mm
	Polar:	
X	Pole position in X direction (abs.)	mm
Y	Pole position in Y direction (abs.)	mm
L1	Distance between pole and start point for contour (abs.)	mm
φ1	Polar angle between pole and start point for contour (abs.)	Degr.
Additional command	Any additional command in G code format	

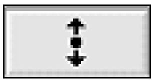
3.5.3 Creating contour elements

When you have created a new contour and specified the start point, you can define the individual elements that the contour comprises.

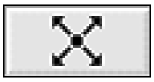
The following contour elements are available for the definition of a contour:



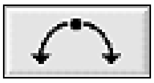
- Horizontal line



- Vertical line



- Diagonal line



- Circle / arc

For each contour element, you must parameterize a separate screen form. The coordinates for a horizontal or vertical line are entered in Cartesian format; however, for the contour elements Diagonal line and Circle/arc you can choose between Cartesian and polar coordinates. If you wish to enter polar coordinates you must first define a pole. If you have already defined a pole for the start point, you can also relate the polar coordinates to this pole. In this case there is therefore no need to define another pole.

Cylinder surface transformation

For contours (e.g. slots) on cylinders, the angle data for lengths are specified. If the 'cylinder surface transformation' function is activated via the "Alternat." softkey, you can also define the lengths of contours (in the circumferential direction of the cylinder envelope) with angle data. In that case, instead of X, Y and I, J, you will enter Xα, Yα and Iα, Jα (see also Sec. "Cylinder surface transformation").

Please refer to the machine manufacturer's instructions.

Parameter input

Parameter entry is supported by various "help displays" that explain the parameters.

If you leave certain fields blank, ShopMill assumes that the values are unknown and attempts to calculate them from other parameters.

Conflicts may result if you enter more parameters than are absolutely necessary for a contour. In such a case, try entering less parameters and allowing ShopMill to calculate as many parameters as possible.

Machining direction

In the case of path milling, the contour is always machined in the programmed direction. By programming the contour in the clockwise direction or counterclockwise direction, you can determine whether the contour is machined with down-cut milling or up-cut milling (see the following table).

Outside contour		
Required direction of rotation for machining	CW spindle rotation	CCW spindle rotation
Down-cut	Programming in clockwise direction CCW cutter radius compensation	Programming in counterclockwise direction, CW cutter radius compensation
Up-cut	Programming in counterclockwise direction, CW cutter radius compensation	Programming in clockwise direction CCW cutter radius compensation

Inside contour		
Required direction of rotation for machining	CW spindle rotation	CCW spindle rotation
Down-cut	Programming in counterclockwise direction, CCW cutter radius compensation	Programming in clockwise direction CW cutter radius compensation
Up-cut	Programming in clockwise direction CW cutter radius compensation	Programming in counterclockwise direction, CCW cutter radius compensation

Contour transition elements

As a transition between two contour elements, you can choose a radius or a chamfer. The transition is always appended to the end of a contour element. The contour transition is selected in the parameterization screen form of the contour element.

You can use a contour transition element whenever there is an intersection between two successive elements which can be calculated from input values. Otherwise you must use the "Straight/Circle" contour elements.

That means that for a closed counter, you can also program a transition element from the last to the first element of the contour. The contour starting point is outside the contour after you have programmed the transition.

Additional commands

For each contour element, you can enter any additional commands in G code format. For example, you can program "G9" deceleration, exact stop for the circle contour element.
You can enter the additional commands (max. 40 characters) in the extended parameterization screen form ("All parameters" softkey).

Additional functions

The following additional functions are available for programming a contour:

- Tangent to preceding element
  You can program the transition to the preceding element as a tangent.
- Dialog selection
  If two different possible contours result from the parameters entered thus far, one of the options must be selected.
- Close contour
  From the current position, you can close the contour with a straight line to the starting point.

Creating a contour element

➢ Select a contour element via softkey.

➢ Enter all the data available from the workpiece drawing in the input form (e.g. length of straight line, target position, transition to next element, angle of lead, etc.).

Accept

➢ Press the "Accept" softkey.

The contour element is added to the contour.

➢ Repeat the procedure until the contour is complete.

Accept

➢ Press the "Accept" softkey.

The programmed contour is transferred to the machining plan.

All parameters

If you want to display further parameters for certain contour elements, e.g. to enter additional commands, press the "All parameters" softkey.

Defining a pole

If you wish to enter the contour elements Diagonal line and Circle/arc in polar coordinates, you must first define a pole.

Continue Pole

➢ Press the "Continue" and "Pole" softkeys.

➢ Enter the coordinates of the pole.

Accept

➢ Press the "Accept" softkey.

The pole is defined. You can now choose between "Cartesian" and "Polar" in the input screen form for the Diagonal line and Circle/Arc contour elements.

Tangent to preceding element

Tangent to prec. elem.

When entering data for a contour element you can program the transition to the preceding element as a tangent.

➢ Press the "Tangent to prec. elem." softkey.

The angle to the preceding element $\alpha 2$ is set to 0°. The "tangential" selection appears in the parameter input field.

Selecting a dialog

When entering data for a contour element, there may be two different contour options, one of which you have to select.

Select dialog

➢ Press the "Select dialog" softkey to switch between the two different contour options.

The selected contour appears in the graphics window as a solid black line and the alternative contour appears as a dashed green line.

Accept dialog

➢ Press the "Accept dialog" softkey to accept the chosen alternative.

Closing the contour

A contour always has to be closed. If you do not wish to create all contour elements from starting point to starting point, you can close the contour from the current position to the starting point.

Continue | Close contour

➢ Press the "Continue" and "Close contour" softkeys.

ShopMill inserts a straight line between your current position and the starting point.

Parameters	Description for contour element "straight line"	Unit
	Cartesian:	
X	End point in the X direction (abs. or inc.) Incremental dimensions: The plus/minus sign is evaluated.	mm
Y	End point in the Y direction (abs. or inc.) Incremental dimensions: The plus/minus sign is evaluated.	mm
L	Length of line	mm
$\alpha 1$	Starting angle to X axis	Degr.
$\alpha 2$	Angle to preceding element Tangential transition: $\alpha 2=0$	Degr.
	Polar:	
L1	abs: distance between pole and end point inc: distance between final point and end point Incremental dimensions: The plus/minus sign is evaluated.	mm mm
$\varphi 1$	abs: polar angle between pole and end point inc: polar angle between final point and end point Incremental dimensions: The plus/minus sign is evaluated.	Degr. Degr.

L α1 α2	Length of line Starting angle to X axis Angle to preceding element Tangential transition: α2=0	mm Degr. Degr.
Transition to following element	FS: Chamfer as transition element to next contour element R: Radius as transition element to next contour element	mm mm
Additional command	Any additional command in G code format	

Parameters	Description for contour element "circle"	Unit
Direction of rotation	Clockwise rotation Counterclockwise rotation	
R	Radius of circle	mm
 X Y I J α1 α2 β1 β2	**Cartesian:** End point in the X direction (abs. or inc.) Incremental dimensions: The plus/minus sign is evaluated. End point in the Y direction (abs. or inc.) Incremental dimensions: The plus/minus sign is evaluated. Circle center point in X direction (abs. or inc.) Incremental dimensions: The plus/minus sign is evaluated. Circle center point in Y direction (abs. or inc.) Incremental dimensions: The plus/minus sign is evaluated. Starting angle to X axis Angle to preceding element Tangential transition: α2=0 End angle to X axis Angle of aperture of circle	 mm mm mm mm Degr. Degr. Degr. Degr.
 L1 φ1 L2 φ2 α1 α2 β1 β2	**Polar:** abs: distance between pole and end point inc: distance between final point and end point Incremental dimensions: The plus/minus sign is evaluated. abs: polar angle between pole and end point inc: polar angle between final point and end point Incremental dimensions: The plus/minus sign is evaluated. abs: distance between pole and center of circle inc: distance between final point and center of circle Incremental dimensions: The plus/minus sign is evaluated. abs: polar angle between pole and center of circle inc: polar angle between final point and center of circle Incremental dimensions: The plus/minus sign is evaluated. Starting angle to X axis Angle to preceding element Tangential transition: α2=0 End angle to X axis Angle of aperture of circle	 mm mm Degr. Degr. mm mm Degr. Degr. Degr. Degr. Degr. Degr.
Transition to following element	FS: Chamfer as transition element to next contour element R: Radius as transition element to next contour element	mm mm
Additional command	Any additional command in G code format	

3.5.4 Changing a contour

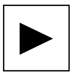
You can change a previously created contour later. Individual contour elements can be

- appended,
- modified,
- inserted or
- deleted.

If your program contains two contours of the same name, changes to the one contour are automatically applied to the second contour with the same name.

Appending a contour element

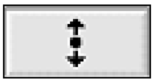
➢ Select the contour in the machining plan.

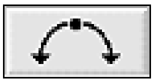


➢ Press the "Cursor Right" key.

The individual contour elements are listed.

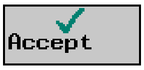
➢ Place the cursor on the last element before the end of the contour.





➢ Select the required contour element via softkey.

➢ Enter the parameters in the input screen.

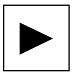


➢ Press the "Accept" softkey.

The required contour element is appended to the contour.

Modifying a contour element

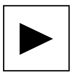
➢ Select the contour in the machining plan.



➢ Press the "Cursor Right" key.

The individual contour elements are listed.

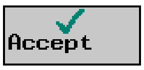
➢ Place the cursor on the contour element that you want to modify.



➢ Press the "Cursor Right" key.

The associated input form is opened and an enlarged view of the selected element appears in the programming graphics.

➢ Enter the desired changes.



➢ Press the "Accept" softkey.

The current values for the contour element are accepted and the change is immediately visible in the programming graphics.

Changing the selected dialog

If when you entered the data for a contour element there were two different contour options and you chose the wrong one, you can alter your choice afterwards. If the contour is unique as a result of other parameters, the system will not prompt you to make a selection.

- Open the input screen form for the contour element.

Change selection

- Press the "Change selection" softkey.

The two selection options appear again.

Select dialog

- Press the "Select dialog" softkey to switch between the two different contour options.

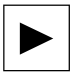
Accept dialog

- Press the "Accept dialog" softkey.

The chosen alternative is accepted.

Inserting a contour element

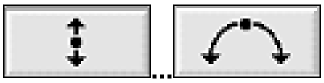
- Select the contour in the machining plan.



- Press the "Cursor Right" key.

The individual contour elements are listed.

- Position the cursor on the contour element after which the new element is to be inserted.



- Select a new contour element via softkey.
- Enter the parameters in the input screen.

Accept

- Press the "Accept" softkey.

The contour element is inserted in the contour. Subsequent contour elements are updated automatically according to the new contour status.

When you insert a new element into a contour, the remaining contour elements are not interpreted until you select the symbol for the first subsequent element alongside the graphics window using the cursor. The end point of the inserted element may not correspond to the start point of the subsequent element. In this case, ShopMill outputs the error message "Geometrical data contradictory". To rectify the problem, insert an incline without entering parameter values.

Deleting a contour element

➢ Select the contour in the machining plan.

►

➢ Press the "Cursor Right" key.

The individual contour elements are listed.

➢ Place the cursor on the contour element that you want to delete.

Delete element

➢ Press the "Delete element" softkey.

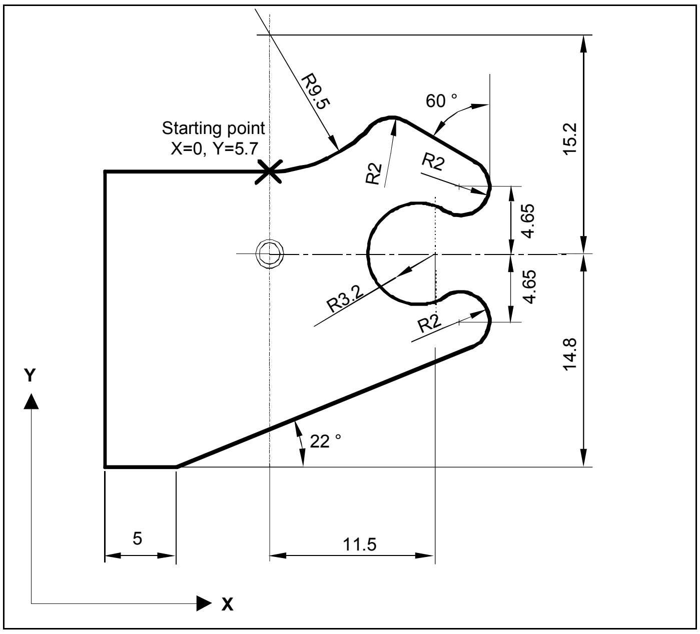
➢ Press the "Delete" softkey.

The selected contour element is deleted.

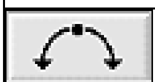
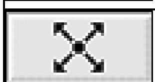
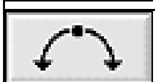
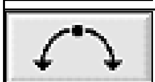
3.5.5 Programming examples for freely defined contours

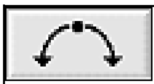
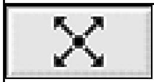
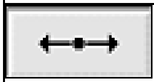
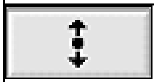
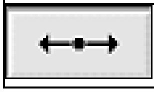
Example 1

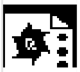
Starting point: X=0 abs., Y=5.7 abs.
The contour is programmed in the clockwise direction with dialog selection.



Workpiece drawing of contour

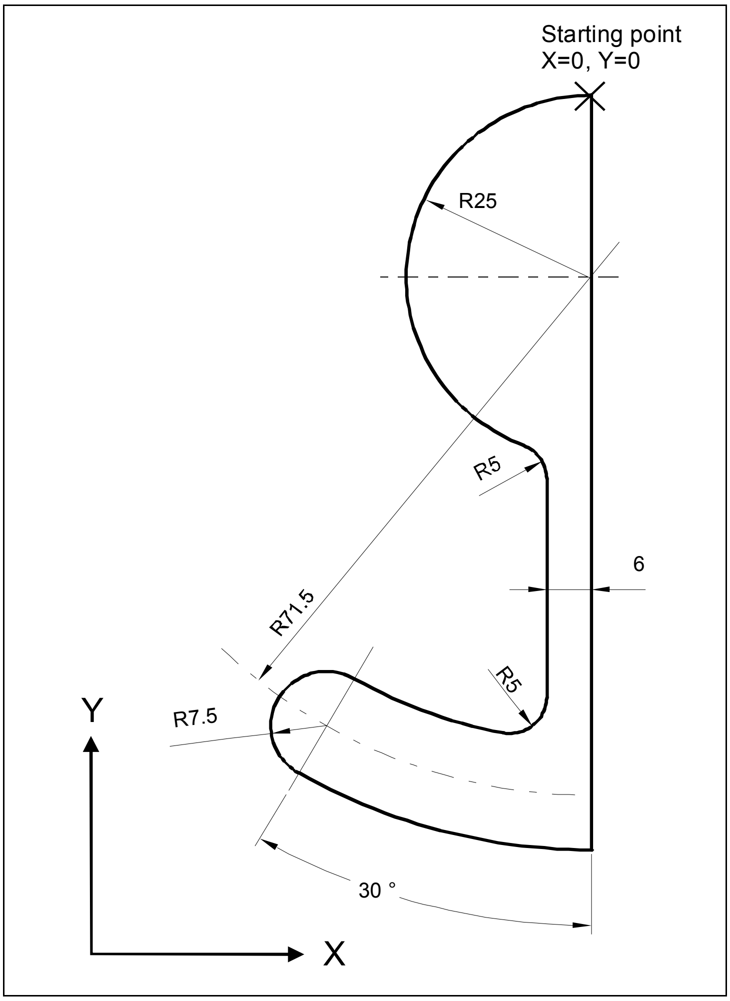
Element	Input	Remarks
	CCW rotation, R=9.5, I=0 abs., make dialog selection, transition to following element: R=2	
	α1=−30 degrees	Observe angles in help screen!
	CW rotation, tangent prev. elem., R=2, J=4.65 abs.	
	CCW rotation, tangent prev. elem. R=3.2, I=11.5 abs., J=0 abs., make dialog selection, Make dialog selection	

	CW direction of rotation, tangent to preced. R=2, J=–4.65 abs., select dialog	
	Tangent to previous element Y=–14.8 abs., –1=–158 degrees	Observe angles in help screen!
	All parameters, L=5, select dialog	
	Y=5.7 abs.	
	X=0 abs.	



Example 2

Starting point: X=0 abs., Y=0 abs.
The contour is programmed in the clockwise direction with dialog selection. It is advisable to display all parameters for this contour by selecting the "All param." softkey.

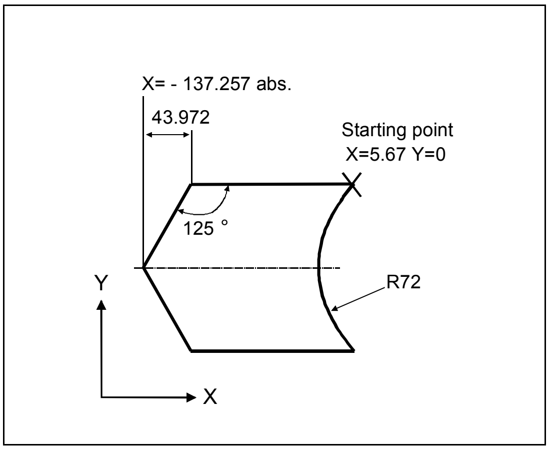


Workpiece drawing of contour

Element	Input	Remarks
	Y=−104 abs.	
	CW rotation, R=79, I=0 abs., make dialog selection, all parameters, β2=30 degrees	
	CW rotation, tangent prev. elem. R=7.5, all parameters, β2=180 degrees	
	CCW rotation, R=64, X=−6 abs., I=0 abs., make dialog selection, make dialog selection Transition to following element: R=5	
	All parameters, α1=90 degrees, Transition to following element: R=5	Observe angles in help screen!
	Direction of rotation right, R=25, X=0 abs., Y=0 abs. I=0 abs., make dialog selection, make dialog selection	

Example 3

Starting point: X=5.67 abs., Y=0 abs.
The contour is programmed in the counterclockwise direction.



Workpiece drawing of contour

Element	Input	Remarks
	All parameters, α1=180 degrees	Observe angles in help screen!
	X=−43.972 inc, all parameters X=−137.257 abs, −1=−125 degrees	Coordinate X in "abs" and in "inc" Observe angles in help screen!
	X=43.972 inc α1=−55 degrees	Coordinate X in "abs" and in "inc" Observe angles in help screen!
	X=5.67 abs	
	CW rotation, R=72, X=5.67 abs., Y=0 abs., make dialog selection	

3.5.6 Path milling

You can mill along any contour you have programmed with the "Path milling" function. The function operates with cutter radius compensation. Machining can be performed in either direction, i.e. in the direction of the programmed contour or in the opposite direction. The contour does not have to be closed. You can perform any of the following operations:

- Inside or outside machining (on left or right of the contour).
- Machining along center-point path

For machining in the opposite direction, contours must not consist of more than 170 contour elements (incl. chamfers/radii).
Special aspects (except for feed values) of free G code input are ignored during path milling in the opposite direction to the contour.

Cont. mill. | Path milling >

➢ Press the "Cont. mill." and "Path milling" softkeys.

Path milling on right or left of the contour

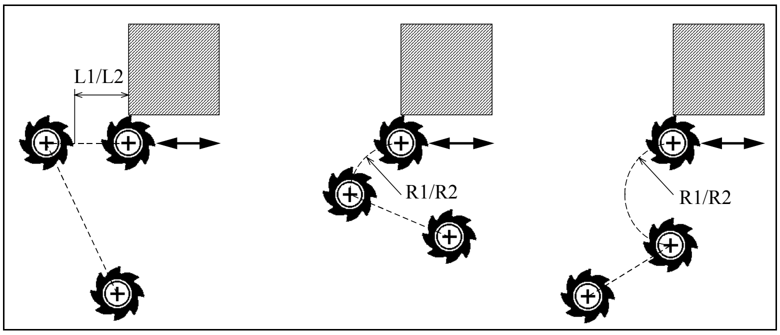
A programmed contour can be machined with the cutter radius on the right or left. You can also select various modes and strategies of approach and retraction from the contour.

Approach/retraction mode

The tool can approach or retract from the contour along a quadrant, semi-circle or straight line.

- With a quadrant or semi-circle approach path, you must enter the current center point path.
- With a straight line, you must specify the distance between the cutter outer edge and the contour start or end point.

You can also program a mixture of modes, e.g. approach along quadrant, retract along semi-circle.



Approach and retraction along straight line, quadrant and semi-circle; (L1=approach length, L2=retract length, R1=approach radius, R2=retract radius)

Approach/retraction strategy

You can choose between planar approach/retraction and spatial approach/retraction:

- Planar approach: First approach depth in the Z direction then in the XY plane.
- Spatial approach: Approach in depth and plane simultaneous.

- Retraction is performed in reverse order.
  Mixed programming is possible, for example, approach in the plane, retract spatially.

Path milling along the center-point path

A programmed contour can also be machined along the center-point path if the operation has been activated under radius compensation (no radius compensation). In this case, approaching and retraction is only possible along a straight line or vertical. Vertical approach/retraction can be used for closed contours, for example.

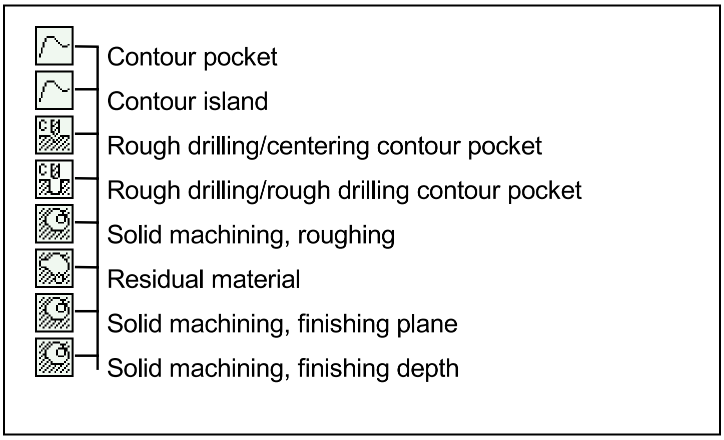
Parameters	Description	Unit
T, D, F, S, V	See Sec. "Programming the tool, offset value and spindle speed".	
Radius compensation	Machining to the left of the contour Machining to the right of the contour Machining along the center path	
Machining type	Roughing Finishing Chamfer	
Machining direction	Forward: machining is performed in the programmed contour direction Backward: machining is performed in the opposite direction to the programmed contour	
Z0	Reference plane (abs. or inc.)	
Z1	End depth (abs. or inc.) (not for chamfer)	mm
DZ	Infeed depth (not for chamfer)	mm
FS	Chamfer width (for chamfer only), inc.	mm
ZFS	Insertion depth tool tip (for chamfer only), abs. or inc.	mm
UZ	Finishing allowance base (not for chamfer)	mm

UXY	Finishing allowance on edge (not applicable to center-point path machining operations) (not for chamfer)	mm
Approach mode	Quadrant: Part of a spiral (only with path milling left and right of the contour) Semicircle: Part of a spiral (only with path milling left and right of the contour) Linear: Slope in space Perpendicular: Perpendicular to the path (only with path milling on the center path)	
Approach strategy	planar three-dimensionally (not with perpendicular approach mode)	
R1 or L1	Approach radius (only for path milling on left and right of contour), approach length	mm
Retract mode	Quadrant: Part of a spiral (only with path milling left and right of the contour) Semicircle: Part of a spiral (only with path milling left and right of the contour) Linear: Slope in space Perpendicular: Perpendicular to the path (only with path milling on the center path)	
Retract strategy	planar spatial (not with perpendicular approach mode)	
R2 or L2	Retract radius (only for path milling on left and right of contour), retract length	mm
Retraction mode	If more than one depth infeed is necessary, specify the retraction height to which the tool retracts between the separate infeeds (in the transition from the end of the contour to the beginning). Z0 + safety clearance Safety clearance To retraction plane No retraction	

3.5.7 Predrilling a contour pocket

If a milling tool cannot be inserted in the center to remove stock from contour pockets, it is necessary to rough-drill first.
The number and positions of the required rough drill holes depends on certain conditions, e.g. type of contour, tool, plane infeed, finishing allowances.

The rough-drilling cycle comprises a centering cycle and the actual rough-drilling cycle.
The drilling positions in the contour pocket cycle are determined when the contour pocket is calculated. This calculation generates a special drilling program that is called in the rough drilling cycles (centering and rough drilling).



Example of a chain containing rough drilling (centering and rough drilling) and solid machining

If you mill several pockets and want to avoid unnecessary tool changeover, predrill all the pockets first and then remove the stock. In this case, for centering/predrilling, you also have to enter the parameters that appear when you press the "All parameters" softkey. Then program as follows:

1. Contour pocket 1
2. Centering
3. Contour pocket 2
4. Centering
5. Contour pocket 1
6. Predrilling
7. Contour pocket 2
8. Predrilling
9. Contour pocket 1
10. Remove stock
11. Contour pocket 2
12. Remove stock

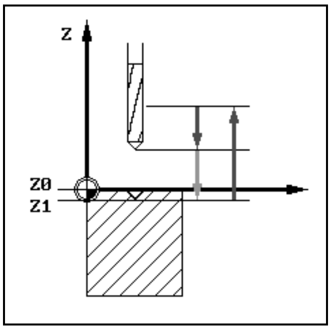
If you are doing all the machining for the pocket at once, i.e. centering, rough-drilling and removing stock directly in sequence, and do not set the additional parameters for centering/rough-drilling, ShopMill will take these parameter values from the stock removal (roughing) machining step.

Centering

Cont. mill. | Predrilling > | Centering

➢ Press the "Cont. Mill.", "Predrilling", and "Centering" softkeys.

Call the help display with the HELP key



Centering for a contour pocket

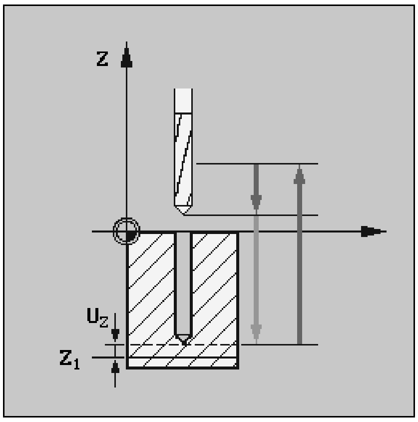
Parameters	Description	Unit
T, F, S	See Sec. "Programming the tool, offset value and spindle speed".	
TR	Reference tool for centering	
Z0	Workpiece height (abs.)	mm
Z1	Depth with reference to Z0 (inc.)	mm
DXY	Max. infeed plane Alternatively, you can specify the plane infeed as a %, as the ratio --> plane infeed (mm) to milling cutter diameter (mm).	mm %
UXY	Finishing allowance, plane	mm
Retraction mode	Retraction mode before new infeed If a machining operation requires several insertion points, you can program the retraction height: • To retraction plane • Z0 + safety clearance On making the transition to the next insertion point, the tool returns to this height. If there are no elements larger than Z0 in the pocket area, "Z0 + safety clearance" can be selected as the retraction mode.	 mm mm

Predrilling

Cont. mill. | Predrilling > | Predrilling

➢ Press the "Cont. mill.", "Predrilling", and "Predrilling" softkeys.

Call the help display with the HELP key



Predrilling a contour pocket

Parameters	Description	Unit
T, F, S	See Sec. "Programming the tool, offset value and spindle speed".	
TR	Reference tool for predrilling	
Z0	Workpiece height (abs.)	mm
Z1	Depth with reference to Z0 (inc.)	mm
DXY	Max. infeed plane Alternately, you can specify the plane infeed as a %, as the ratio --> plane infeed (mm) to milling cutter diameter (mm).	mm %
UXY	Finishing allowance, plane	mm
UZ	Finishing allowance, depth	mm
Retraction mode	Retraction mode before new infeed If a machining operation requires several insertion points, you can program the retraction height: • To retraction plane • Z0 + safety clearance On making the transition to the next insertion point, the tool returns to this height. If there are no elements larger than Z0 in the pocket area, "Z0 + safety clearance" can be selected as the retraction mode.	 mm mm

3.5.8 Milling a contour pocket (roughing)

Before you can machine a pocket with islands, you must enter the contour of the pocket and islands (see Sec. "Freely defined contours"). The first contour you specify is interpreted as the pocket contour and all the others as islands.

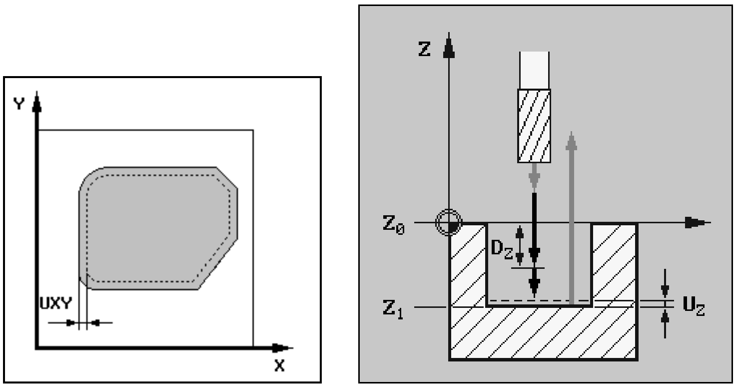
Using the programmed contours and the input screen form for solid machining, ShopMill generates a program which removes the pockets with islands from inside to outside in parallel to the contour. The direction is determined by the direction of rotation specified in the program header for machining (reverse or synchronous).

The islands can also be located partially outside the pocket or overlap each other.

Cont. mill. | Mill pocket >

- Press the "Cont. mill." and "Mill pocket" softkeys.

Call help display with the HELP **key**



Help displays for solid machining

Parameters	Description	Unit
T, F, V	See Sec. "Programming the tool, offset value and spindle speed".	
Machining mode	Roughing	
Z0	Workpiece height (abs.)	mm
Z1	Depth with reference to Z0 (abs. or inc.)	mm
DXY	Max. infeed in X/Y plane. Alternately, you can specify the plane infeed as a %, as a ratio → plane infeed (mm) to milling cutter diameter (mm).	mm %
DZ	Max. infeed depth (abs. or inc.)	mm
UXY	Finishing allowance, plane	mm
ZU	Finishing allowance, depth	mm
Start point	The starting point can be determined **automatically** or entered **manually**.	
X Y	Starting point X (abs.), manual input only Starting point Y (abs.), manual input only	mm mm

Insertion	Oscillation: The tool is inserted oscillating with the program. angle (EW). Helical: The tool is inserted along a helical path with the programmed radius (ER) and programmed pitch (EP). Center: For this insertion strategy, a milling cutter is required that cuts in the center. The programmed feed (FZ) is used for insertion.	
EW	Insertion angle (for oscillation only)	Degr.
FZ	Feedrate FZ (for center only)	mm/min
EP	Insertion gradient (for helical only) The gradient of the helix may be smaller in some geometric conditions.	mm/rev
ER	Insertion radius (for helical only) The radius must not be larger than the cutter radius, otherwise material will remain. Also make sure the pocket is not violated.	mm
Retraction mode	If the machining operation requires several points of insertion, the retraction height must be programmed: • To retraction plane • Z0 + safety clearance (SC) On making the transition to the next insertion point, the tool returns to this height. If no elements greater than Z0 are in the pocket area, Z0 + safety clearance (SC) can be programmed as the retraction mode.	 mm mm

When input manually, the starting point can also be located outside the pocket. This can be useful, for example, when machining a pocket which is open on one side. The machining operation then begins without insertion with a linear movement into the open side of the pocket.

3.5.9 Removing residual material from a contour pocket

If you have removed stock in a pocket (with/without islands) and residual material still remains, ShopMill will detect this automatically. You can use a suitable tool to remove this residual material without having to machine the whole pocket again, i.e. avoiding unnecessary idle motions.
Material that remains as part of the finishing allowance is not residual material.

The residual material is calculated on the basis of the milling cutter used for stock removal.

If you mill several pockets and want to avoid unnecessary tool changeover, remove stock from all the pockets first and then remove the residual material. In this case, for removing the residual material, you also have to enter a value for the "Reference tool TR" parameter that appears when you press the "All parameters" softkey.

Then program as follows:

1. Contour pocket 1
2. Remove stock
3. Contour pocket 2
4. Remove stock
5. Contour pocket 1
6. Remove residual material
7. Contour pocket 2
8. Remove residual material

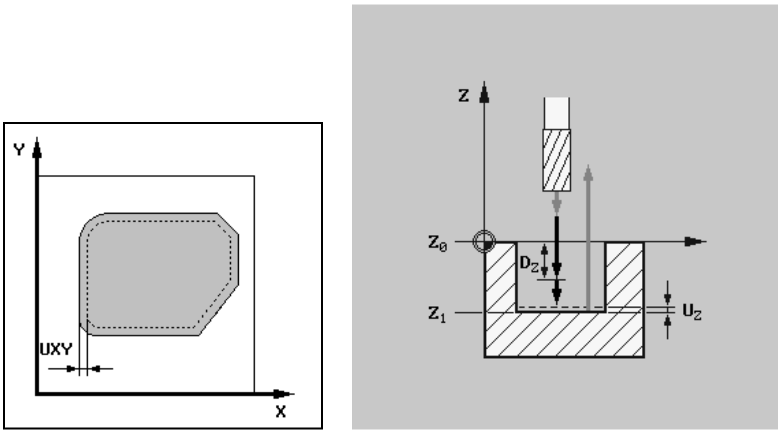
The "Residual material" function is a software option.

Pocket Res. Mat. >

➢ Press the "Cont. mill." and "Pocket Res. Mat." softkeys.

Call help display with the

key



Help display for residual material

Parameters	Description	Unit
T, F, V	See Sec. "Programming the tool, offset value and spindle speed".	
Machining type	Roughing	
TR	Reference tool for residual material	
Z0	Workpiece height (abs.)	mm
Z1	Depth with reference to Z0 (abs. or inc.)	mm
DXY	Max. infeed, plane Alternatively, you can specify the plane infeed as a %, as a ratio --> plane infeed (mm) to milling cutter diameter (mm).	mm %
DZ	Max. infeed, depth	mm
UXY	Finishing allowance, plane	mm
UZ	Finishing allowance, depth	mm

Retraction mode	If the machining operation requires several points of insertion, the retraction height can be programmed: • To retraction plane • Z0 + safety clearance (SC) On making the transition to the next insertion point, the tool returns to this height. If no elements greater than Z0 are in the pocket area, Z0 + safety clearance (SC) can be programmed as the retraction mode.	 mm mm

3.5.10 Finishing the contour pocket

If you programmed stock removal from the pocket with a finishing allowance for the base or edge of the pocket, you still have to finish the pocket.
Separate blocks must be programmed for finishing the base and/or for finishing the edge. In each case, the pocket will only be machined once.
When finish cutting, ShopMill takes any existing island(s) into account as is the case for rough cutting.

Mill pocket >

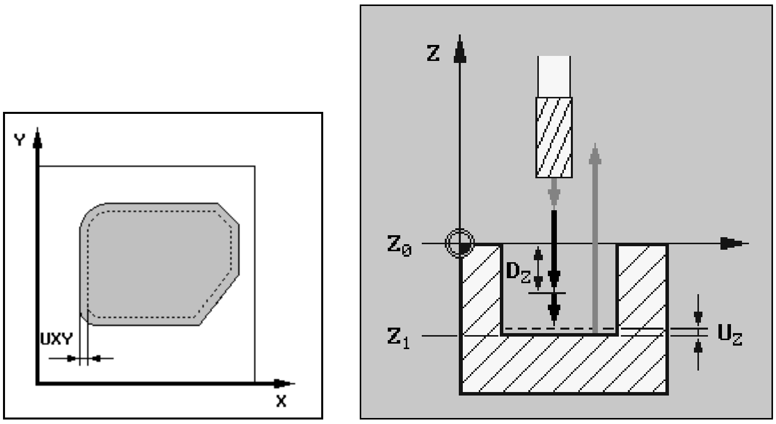
➢ Press the "Cont. mill." and "Mill pocket" softkeys".

➢ Select "Finish base" or "Finish edge" in machining mode.

Call help display

with the 

key



Help display for "Finish pocket with islands"

Parameters	Description of finish cut along base:	Unit
T, F, V	See Sec. "Programming the tool, offset value and spindle speed".	
Machining type	Finish base	
Z0	Workpiece height (abs.)	mm
Z1	Depth with reference to Z0 (abs. or inc.)	mm
DXY	Max. infeed, plane Alternately, you can specify the plane infeed as a %, as a ratio --> plane infeed (mm) to milling cutter diameter (mm).	mm %
UXY	Finishing allowance, plane	mm
UZ	Finishing allowance, depth	mm
Start point	The starting point can be determined automatically or entered manually. Manual entry allows for a start point outside the pocket, whereby straight line machining into the pocket is performed first, e.g. for a pocket with a side opening without any insertion.	
X Y	Coordinate of starting point (abs.), manual input only Coordinate of starting point (abs.), manual input only	mm mm
Insertion	Oscillation: The tool is inserted at the programmed angle (EW). Helical: The tool is inserted along a helical path with the programmed radius (ER) and programmed pitch (EP). Center: For this insertion strategy, a milling cutter is required that cuts in the center. The programmed feed (FZ) is used for insertion.	
EW	Insertion angle (for oscillation only)	Degr.
EP	Insertion gradient (only for helical) The gradient of the helix may be smaller in some geometric conditions.	mm/rev
ER	Insertion radius (only for helical) The radius must not be larger than the cutter radius, otherwise material will remain. Also make sure the pocket is not violated.	mm
FZ	Feedrate FZ (for Center only)	mm/min
Retraction mode	If the machining operation requires several points of insertion, the retraction height can be programmed: • To retraction plane • Z0 + safety clearance (SC) On making the transition to the next insertion point, the tool returns to this height. If no elements greater than Z0 are in the pocket area, Z0 + safety clearance (SC) can be programmed as the retraction mode.	 mm mm

Parameters	Description of finish cut along edge:	Unit
T, F, V	See Sec. "Programming the tool, offset value and spindle speed".	
Machining type	Edge finishing	
Z0	Workpiece height (abs.)	mm
Z1	Depth with reference to Z0 (abs. or inc.)	mm
DZ	Max. infeed, depth	mm
UXY	Finishing allowance, plane	mm
Retraction mode	If the machining operation requires several points of insertion, the retraction height can be programmed: • To retraction plane • Z0 + safety clearance (SC) On making the transition to the next insertion point, the tool returns to this height. If no elements greater than Z0 are in the pocket area, Z0 + safety clearance (SC) can be programmed as the retraction mode.	 mm mm
	Note: An alternative to the "Edge finish cut" option is the "Path milling" function which offers greater optimization potential (approach and retract strategies and modes).	

3.5.11 Chamfering a contour pocket

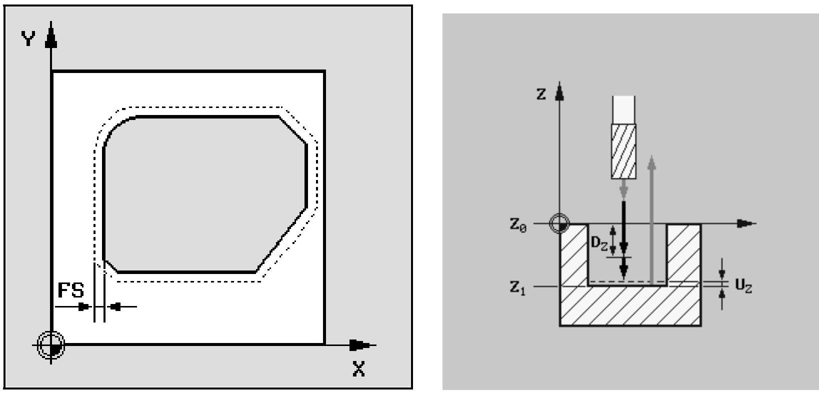
If you have planned edge breaking, mill a chamfer after that.

Mill pocket >

➢ Press the "Cont. mill." and "Mill pocket" softkeys.

➢ Select "Chamfer" in machining mode.

Call help display with the HELP key



Help display "Chamfer pocket"

If you want to mill a chamfer and have programmed inside corners without filleting during rounding, you must specify the radius of the finishing tool as the rounding in the contour.

Parameters	Description for chamfer:	Unit
T, F, V	See Sec. "Programming the tool, offset value and spindle speed".	
Machining type	Chamfer	
Z0	Workpiece height (abs.)	mm
FS	Chamfer width (for chamfer only), inc.	mm
ZFS	Insertion depth tool tip (for chamfer only), abs. or inc.	mm

3.5.12 Milling contour spigots (roughing)

Use the "Mill spigot" function if you want to mill any kind of spigot.

Before you mill the spigot, you must first enter a blank contour and then one or more spigot contours. The blank contour defines the outer limits of the material. The tool moves at rapid traverse outside this area. Material is then removed between the blank contour and spigot contour.

You can select the machining mode (roughing or finishing) for milling. If you want to rough and then finish, you have to call the machining cycle twice (Block 1 = roughing, Block 2 = finishing). The programmed parameters are retained on the second call. For more on finishing, see Sec. "Finishing the contour spigot".

If you only program a blank contour without a second contour for a spigot, you can face mill the blank contour.

Approach/retraction

1. The tool approaches the starting point at rapid traverse at the height of the retraction plane and adjusts to the safety clearance. The start point is calculated by ShopMill.
2. The tool first infeeds to the machining depth and then approaches the spigot contour from the side in a quadrant at machining feedrate.
3. The spigot is cleared in parallel with the contours from the outside in. The direction is determined by the machining direction (climb/conventional) (see "Creating a new program; defining a blank").
4. When the first plane of the spigot has been cleared, the tool retracts from the contour in a quadrant and then infeeds to the next machining depth.
5. The spigot is again approached in a quadrant and cleared in parallel with the contours from outside in.
6. Steps 4 and 5 are repeated until the programmed spigot depth is reached.
7. The tool moves back to the safety clearance at rapid traverse.

Cont. mill. | Mill spigot >

➢ Press the "Cont. mill." and "Mill spigot" softkeys.

➢ Select "Roughing" machining mode.

Parameters	Description for roughing	Unit
T, D, F, S, V	See Sec. "Creating program blocks".	
Machining type	▽ Roughing	
Z0 Z1 DXY DZ UXY UZ	Reference point in Z direction (abs.) Depth with reference to Z0 (abs. or inc.) Maximum infeed in the XY plane Plane infeed in %: Ratio of plane infeed (mm) to milling cutter diameter (mm) Maximum depth infeed (Z direction) Finishing allowance in plane Finishing allowance in depth	mm mm mm % mm mm mm
Retraction mode	If more than one approach point is necessary, specify the retraction height to which the tool retracts between approach points. • To retraction plane • Z0 + safety clearance If there are no spigots or other elements larger than Z0 in the machining area, "Z0 + safety clearance" can be selected as the retraction mode.	

3.5.13 Removing residual material from a contour spigot

When you have milled a contour spigot and residual material remains in place, this is automatically detected by ShopMill. You can use a suitable tool to remove this residual material without having to machine the whole spigot again, i.e. avoiding unnecessary idle motions.
Material that remains as part of the finishing allowance is not residual material.

The residual material is calculated on the basis of the milling cutter used for clearing.

If you mill several spigots and want to avoid unnecessary tool changeover, clear all the spigots first and then remove the residual material. In this case, for removing the residual material, you also have to enter a value for the "Reference tool TR" parameter that appears when you press the "All parameters" softkey. Then program as follows:

1. Contour blank 1
2. Contour spigot 1
3. Clear spigot 1
4. Contour blank 2
5. Contour spigot 2
6. Clear spigot 2
7. Contour blank 1

8. Contour spigot 1
9. Clear residual material spigot 1
10. Contour blank 2
11. Contour spigot 2
12. Clear residual material spigot 2

The "Residual material" function is a software option.

Cont. mill. | Spigot Res. Mat. >

➢ Press the "Cont. mill." and "Spigot Res. Mat." softkeys.

All parameters

➢ Press the "All parameters" softkey if you want to enter additional parameters.

Parameters	Description	Unit
T, D, F, S, V	See Sec. "Creating program blocks".	
Machining type	▽ Roughing	
TR	Reference tool for residual material	
D	Cutting edge of reference tool (1 or 2)	
Z0 Z1 DXY DZ UXY UZ	Reference point in Z direction (abs.) Depth with reference to Z0 (abs. or inc.) Maximum infeed in the XY plane Plane infeed in %: Ratio of plane infeed (mm) to milling cutter diameter (mm) Maximum depth infeed (Z direction) Finishing allowance in plane Finishing allowance in depth	mm mm mm % mm mm mm
Retraction mode	If more than one approach point is necessary, specify the retraction height to which the tool retracts between approach points. • To retraction plane • Z0 + safety clearance If there are no spigots or other elements larger than Z0 in the machining area, "Z0 + safety clearance" can be selected as the retraction mode.	

3.5.14 Finishing the contour spigot

If you programmed a finishing allowance for the base or edge of the spigot in spigot milling, you still have to finish the spigot.

Separate blocks must be programmed for finishing the base and/or for finishing the edge. In each case, the spigot will only be machined once.

You can program "Path milling" as an alternative to "Edge finishing". Optimization possibilities are also offered for the approach/retract strategy and the approach/retract mode. Then program as follows:

1. Contour blank
2. Contour spigot
3. Mill spigot (roughing)
4. Contour blank
5. Path milling (finishing)
6. Contour spigot
7. Path milling (finishing)

Cont. mill. | Mill spigot >

- Press the "Cont. mill." and "Mill spigot" softkeys.
- Select "Finish base" or "Finish edge" machining mode.

Parameters	Description	Unit
T, D, F, S, V	See Sec. "Creating program blocks".	
Machining type	Finishing the base Finishing the edge	
Z0 Z1 DXY DZ UXY UZ	Reference point in Z direction (abs.) Depth with reference to Z0 (abs. or inc.) Maximum infeed in the XY plane (base finishing only) Plane infeed in %: Ratio of plane infeed (mm) to milling cutter diameter (mm) Maximum depth infeed (Z direction) – (edge finishing only) Finishing allowance in plane Finishing allowance in depth – (edge finishing only)	mm mm mm % mm mm mm
Retraction mode	If more than one approach point is necessary, specify the retraction height to which the tool retracts between approach points. • To retraction plane • Z0 + safety clearance If there are no spigots or other elements larger than Z0 in the machining area, "Z0 + safety clearance" can be selected as the retraction mode.	

3.5.15 Chamfering a contour spigot

If you have planned edge breaking, mill a chamfer after that.

Parameters	Description	Unit
T, D, F, S, V	See Sec. "Creating program blocks".	
Machining type	Chamfer	
Z0 FS ZF	Reference point in Z direction (abs.) Chamfer width; abs Insertion depth tool tip; abs or inc	mm mm mm

3.6 Linear or circular path motions

This function is intended for the implementation of very simple machining operations as path movements.
More complex operations such as contours with chamfers, radii, approach strategies, tangential transitions, etc. should be implemented using the "Mill contour" and "Path milling" functions.

You must program a tool before you program simple lines or circles. A tool with spindle speed is selected by means of softkeys "Straight circle" and "Tool".
You can only program rapid traverse for linear travel motions.

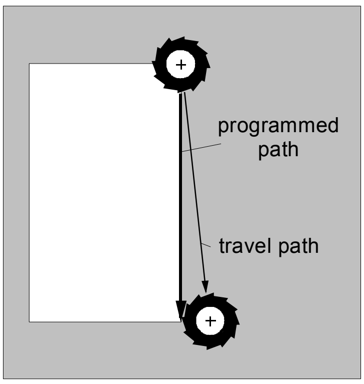
3.6.1 Straight

The tool moves at the programmed feedrate or in rapid traverse from its current position to the programmed end position.

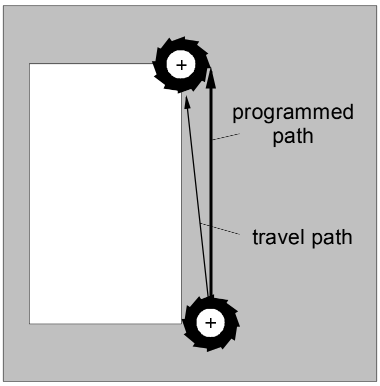
Radius compensation

Alternately you can implement the straight line with radius compensation. The radius compensation acts modally, which means you must deactivate the radius compensation again if you want to traverse without radius compensation. Where several straight line blocks with radius compensation are programmed sequentially, you may select radius compensation only in the first program block.

When executing the first path motion with radius compensation, the tool traverses without compensation at the start point and with compensation at the end point. This means that if a vertical path is programmed, the tool traverses an oblique path. The compensation is not applied over the entire traversing path until the second programmed path motion with radius compensation is executed. The reverse occurs when radius compensation is deactivated.



First path motion with radius compensation



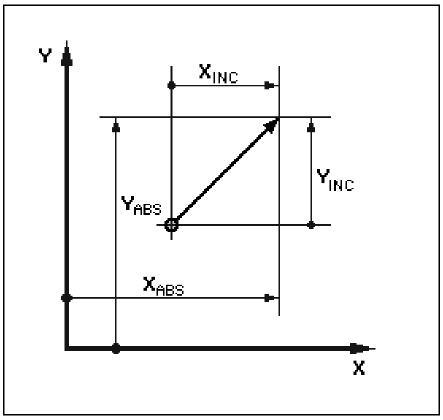
First path motion with deselected radius compensation

To avoid deviation between the programmed and actually traversed path, you can program the first path motion with radius compensation or deselected radius compensation outside the workpiece. Programming without coordinate data is not possible.

Strai. Circle   Straight

➢ Press the "Straight/Circle" and "Straight" softkeys.

Call help display with the HELP key



Help display for a line

Parameters	Description	Unit
X Y Z	Coordinate of end point in X direction (abs. or inc.) Coordinate of end point in Y direction (abs. or inc.) Coordinate of end point in Z direction (abs. or inc.)	mm mm mm
Radius compensation	Input defining which side of the contour the cutter travels in the programmed direction: Radius compensation, left of contour — Radius compensation off Radius compensation, right of contour — Radius compensation is retained as set	

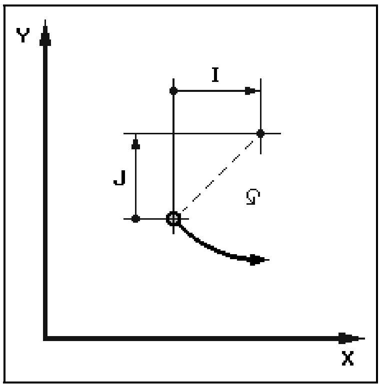
3.6.2 Circle with known center point

The tool travels along a circular path from its current position to the programmed circle end point. You must know the position of the circle center point. The control calculates the radius of the circle/arc on the basis of your interpolation parameter settings.
The circle can only be traversed at machining feedrate. You must program a tool before the circle can be traversed.

Strai. Circle | Circle center point

➢ Press the "Straight/Circle" and "Circle center point" softkeys.

Call help display with the HELP key



Help display for circle with known center point

Parameters	Description	Unit
Direction of rotation	The tool travels in the programmed direction from the circle starting point to its end point. You can program this direction as clockwise or counterclockwise.	
X Y	X position circle end point (abs. or inc.) Y position circle end point (abs. or inc.)	mm mm
I J	Distance between circle start and center point in X direction (inc.) Distance between circle start and center point in Y direction (inc.)	mm mm
Plane	The circle is traversed in the set plane with the relevant interpolation parameters: XYIJ: XY plane with interpolation parameters I and J XZIK: XZ plane with interpolation parameters I and K YZJK: YZ plane with interpolation parameters J and K	 mm mm mm

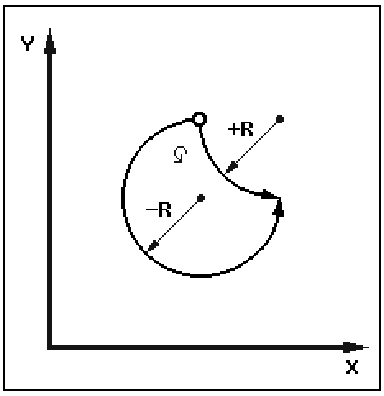
3.6.3 Circle with known radius

The tool traverses a circular path with the programmed radius from its current position to the programmed circle end point. The control system works out the circle center point. You do not need to program interpolation parameters.
The circle can only be traversed at machining feedrate.

Strai. Circle | Circle radius

➢ Press the "Straight/Circle" and "Circle radius" softkeys.

Call help display with the HELP key



Help display for circle with known radius

Parameters	Description	Unit
Direction of rotation	The tool travels in the programmed direction from the circle starting point to its end point. You can program this direction as clockwise or counterclockwise.	
X Y	X position circle end point (abs. or inc.) Y position circle end point (abs. or inc.)	mm mm
R	Radius of arc; You can select the arc of your choice by entering a positive or a negative sign.	mm

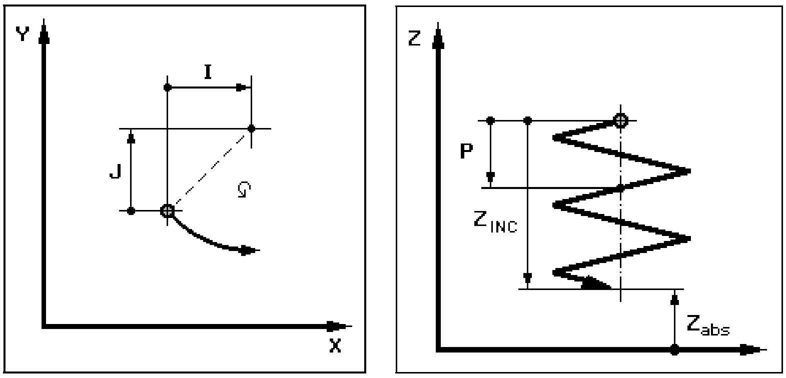
3.6.4 Helix

With helical interpolation, a circular movement is overlaid in the plane with a linear motion in the tool axis, i.e. a spiral is created.

Strai. Circle | Helix >

➢ Press the "Straight Circle" and "Helix" softkeys.

Call help display with the HELP key



Help display for a helix

Parameters	Description	Unit
Direction of rotation	The tool travels in the programmed direction from the circle starting point to its end point. You can program this direction as clockwise or counterclockwise.	
I, J	Incremental: Distance between helix start and center point X and Y direction Absolute: Center point of helix in X and Y directions	mm
P	Pitch of helix; The pitch is programmed in mm per revolution.	mm/360 °
Z	Z position of helix end point (abs. or inc.)	mm

3.6.5 Polar coordinates

If a workpiece has been dimensioned from a central point (pole) with radius and angles, you will find it helpful to program these as polar coordinates.

You can program straight lines and circles as polar coordinates.

Defining a pole

You must define the pole before you can program a line or circle in polar coordinates. This pole acts as the reference point of the polar coordinate system.

The angle for the first line or circle then needs to be programmed in absolute coordinates. You can program the angles for any further lines and circles as either absolute or incremental coordinates.

Strai. Circle | Polar > | Pole >

➢ Press the "Straight/Circle", "Polar" and "Pole" softkeys.

Parameters	Description	Unit
X	X position of the pole (abs. or inc.)	mm
Y	Y position of the pole (abs. or inc.)	mm

3.6.6 Straight polar

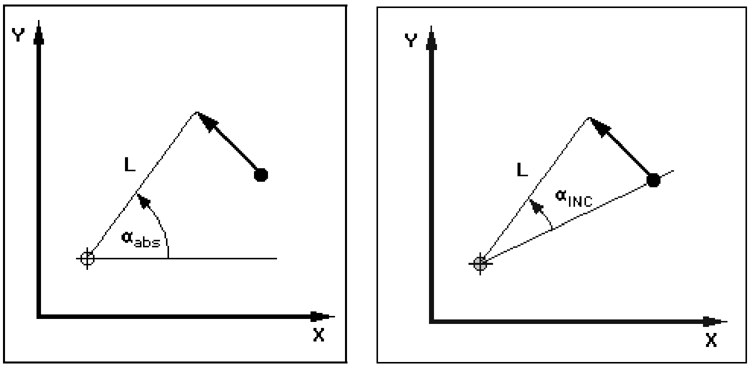
A straight line in the polar coordinate system is defined by a radius (L) and an angle (α). The angle refers to the X axis.
The tool moves from its current position on a straight line to the programmed end point at the machining feedrate or in rapid traverse.

The 1st line in polar coordinates entered after the pole must be programmed with an absolute angle. You can program any further lines or circles with incremental coordinates.

Strai. Circle | Polar > | Straight polar >

➢ Press the "Straight/Circle", "Polar" and "Straight polar" softkeys.

Call help display with the HELP **key**



Help display for polar line with absolute and incremental angle

Parameters	Description	Unit
L	Radius from pole to end point of line	mm
α	Polar angle (abs. or inc., positive or negative)	Degr.
Radius compensation	Input defining which side of the contour the cutter travels in the programmed direction: Radius compensation, left of contour Radius compensation, right of contour Radius compensation off Radius compensation is retained as set	

3.6.7 Circle polar

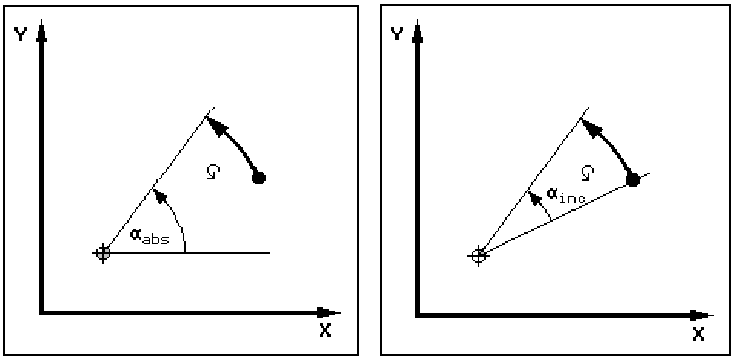
A circle in the polar coordinate system is defined by an angle (α). The angle refers to the X axis.
The tool moves from its current position on a circular path to the programmed end point (angle) at the machining feedrate.
The radius corresponds to the distance from the current tool position to the defined pole, i.e. the circle start and end point positions are at the same distance from the defined pole.

The 1st circle in polar coordinates entered after the pole must be programmed with an absolute angle. You can program any further lines or circles with incremental coordinates.

Strai. Circle | Polar > | Circle polar >

➢ Press the "Straight/Circle", "Polar" and "Circle polar" softkeys.

Call help display with the HELP **key**



Help display for polar circle with absolute and incremental angle

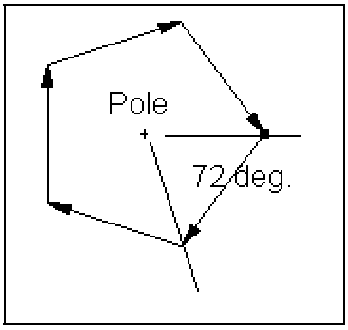
Parameters	Description	Unit
Direction of rotation	The tool travels in the programmed direction from the circle starting point to its end point. You can program this direction as clockwise (right) or counterclockwise (left).	
α	Polar angle (abs. or inc., positive or negative)	Degr.

3.6.8 Programming examples for polar coordinates

Programming a pentagon

You want to machine the outside contour of a pentagon.
Make sure that you enter the correct workpiece dimensions!
Approach starting point in rapid traverse: X70, Y50, radius compensation off.
Pole: X=50, Y=50

1. polar line: L=20, α= –72 **absolute**, radius compensation right
2. to 5th polar line: L=20, α= –72 degrees **incremental**, radius compensation right

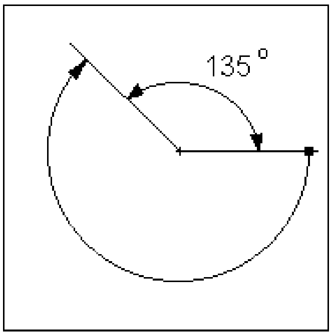


→	N10 RAPID X70 Y50 Z2
⌖	N15 X50 Y50
→	N20 L20 α-72
→	N25 L20 α-72inc
→	N30 L20 α-72inc
→	N35 L20 α-72inc
→	N40 L20 α-72inc
→	N45 L20 α-72inc
END	N50 Program end

Programming graphics and extract from machining plan

Programming an arc of 225 degrees

You want to machine the outside contour of an arc.
Make sure that you enter the correct workpiece dimensions!
Approach starting point in rapid traverse: X=80, Y=50, radius compensation right
Pole: X=60, Y=50
CW rotation, α= 135 degrees absolute



→	N5 RAPID X80 Y50 Z2
⌖	N10 X60 Y50
⤻	N15 F200/min α135
END	Program end

Programming graphics and extract from machining plan

3.7 Drilling

Programming holes and threads

In ShopMill, first program the technology blocks in the exact order in which they need to be performed, e.g.

1. **Centering**, with tool and input of spindle speed and machining feedrate
2. **Deep-hole drilling**, with tool and input of spindle speed and machining feedrate
3. **Tapping** with tool and input of spindle speed and machining feedrate

Once you have programmed the technologies, you need to enter the **position data**. ShopMill provides various positioning patterns (see Sec. "Positions").

This sequence, first technology block and then positioning block must be adhered to in drilling cycles.

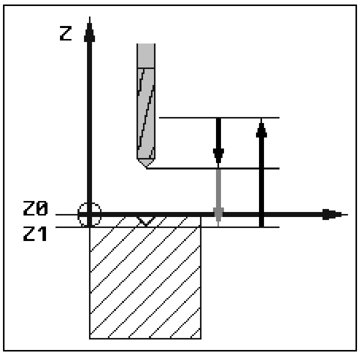
3.7.1 Centering

The tool is moved in rapid traverse to the position to be centered, allowing for the retraction plane and safety clearance. The tool is inserted into the workpiece at programmed feedrate (F) until it reaches Z1 or until the surface diameter is the correct size. When the dwell time expires, the tool is retracted in rapid traverse to either the retraction plane or the safety clearance depending on the setting in parameter "Retraction position pattern". You will find parameter "Retraction position pattern" in the program header or under "Settings" in the "Miscellaneous" menu.

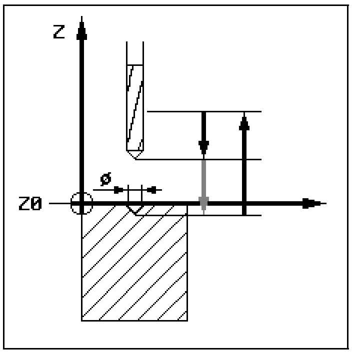
Drill-ing | Centering

➢ Press the "Drilling" and "Centering" softkeys.

Call help display with the HELP key



Help display for centering at depth



Help display for centering on diameter

Parameters	Description	Unit
T, D, F, S, V	See Sec. "Programming the tool, offset value and spindle speed".	
Diameter Tip	The tool is inserted into the workpiece until the diameter on the surface is the correct size. The angle for the center drill entered in the tool list is applied in this case. The drill is inserted into the workpiece until the programmed insertion depth is reached.	
∅ Z1	It is inserted into the workpiece until the diameter is correct. It is inserted into the workpiece until it reaches Z1.	mm mm
Z0	Height of workpiece; Z0 is specified in the position pattern ("Positioning" softkey).	mm
DT	Dwell time for relief cut	s rev

3.7.2 Drilling and reaming

The tool is moved at rapid traverse to the programmed position, allowing for the retraction plane and safety clearance. It is then inserted into the workpiece at the feedrate programmed under F until it reaches depth Z1.

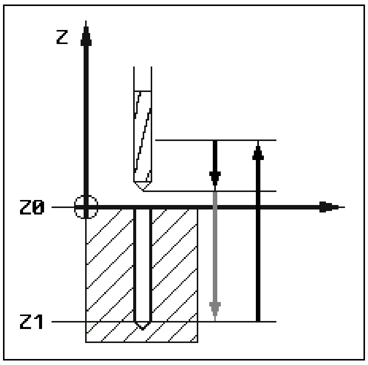
Drilling: If Z1 has been reached and the dwell time expired, the drill is retracted at rapid traverse either to the retraction plane or the safety clearance depending on the setting in parameter "Retraction position pattern". You will find parameter "Retraction position pattern" in the program header or under "Settings" in the "Miscellaneous" menu.

Reaming: If Z1 has been reached and the dwell time expired, the reamer is retracted at the programmed retraction feedrate to the safety clearance.

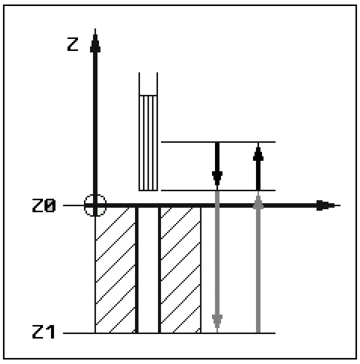
Drill-ing | Drilling reaming

➢ Press the "Drilling" and "Drilling reaming" softkeys.

Call help display with the HELP key



Help display for drilling



Help display for reaming

Parameters	Description	Unit
T, D, F, S, V	See Sec. "Programming the tool, offset value and spindle speed".	
Shank Tip	The drill is inserted into the workpiece until the drill shank reaches the value programmed for Z1. The insertion angle entered in the tool list is applied. The drill is inserted into the workpiece until the drill tip reaches the value programmed for Z1 (does not apply in reaming).	
Z1	Insertion depth for drill tip or drill shank.	mm
Z0	Height of workpiece; Z0 is specified in the position pattern ("Positioning" softkey).	mm
DT	Dwell time for relief cut.	s rev
FB	Retraction feedrate (for reaming only)	

3.7.3 Deep-hole drilling

The tool is moved at rapid traverse to the programmed position, allowing for the retraction plane and safety clearance. It is then inserted into the workpiece at the programmed feedrate.

Deep hole drilling

➢ Press the "Drilling" and "Deep hole drilling" softkeys.

Stock removal

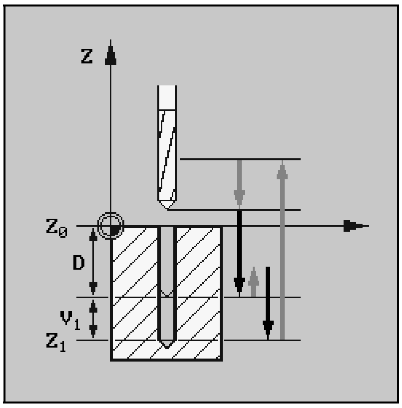
The tool drills at the programmed feedrate (F) until the 1st infeed depth is reached. On reaching the 1st depth, the tool is retracted from the workpiece at rapid traverse for stock removal and is then re-inserted at the 1st infeed depth reduced by a clearance distance (V3). The tool then drills to the next infeed depth and is then retracted again, repeating this process until the final drill depth (Z1) is reached. On expiry of the dwell time (DT), the tool is retracted at rapid traverse to the safety clearance.

Chipbreaking

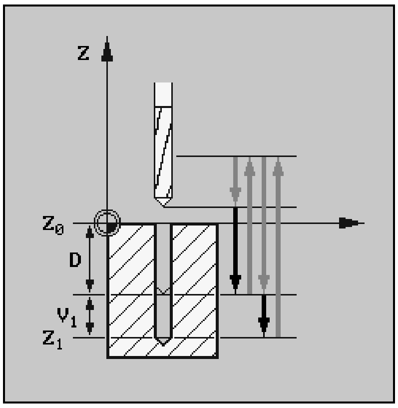
The tool drills at the programmed feedrate (F) until the 1st infeed depth is reached. Once this depth is reached, the tool is retracted by a withdrawal distance (V2) for chip breaking and is then inserted again down to the next drilling depth. It repeats this process until the final drilling depth (Z1) is reached.
The specified amount can be defined either per machine data or in the parameter screenform. If the parameter is pre-assigned via machine data, it does not appear in the parameter screen.

Please refer to the machine manufacturer's instructions.

Call help display with the HELP key



Help display for deep hole drilling with chipbreaking



Help display for deep hole drilling with stock removal

Parameters	Description	Unit
T, D, F, S, V	See Sec. "Programming the tool, offset value and spindle speed".	
Stock removal Chipbreaking	The drill is retracted from the workpiece for stock removal. The drill is retracted by the retraction amount V2 for chipbreaking.	
Tip Shank	The final drilling depth (Z1) refers to the drill tip The final drilling depth (Z1) refers to the drill shank	
Z1	Final drilling depth (inc.)	mm
D	Max. infeed	mm
DF	Percentage for each additional infeed DF=100: Amount of infeed remains constant DF<100: Amount of infeed is reduced in direction of final drilling depth. **Example:** Last infeed was 4mm; DF is 80 next infeed = 4 x 80% = 3.2 mm next infeed = 3.2 x 80% = 2.56 mm etc.	%
V1	Minimum infeed Parameter V1 is provided only if DF< 100 has been programmed. If the infeed increment becomes minimal, a minimum infeed can be programmed in parameter "V1". V1 < Amount of infeed: The tool is inserted by the infeed increment. V1 < Amount of infeed: The tool is inserted by the infeed value programmed under V1.	mm
V2	Specified amount or defined per machine data – for chip breaking only Amount by which the drill is retracted for chipbreaking. V2=0: The tool is not retracted but is left in place for one revolution.	mm
V3	Limit distance – for unclamping only Distance to last infeed depth that the drill approaches at rapid traverse after unclamping. Automatic: The limit distance is calculated by ShopMill.	mm
DT	Dwell time for relief cut.	s rev

3.7.4 Boring

The tool is moved at rapid traverse to the programmed position, allowing for the retraction plane and safety clearance. It is then inserted into the workpiece at the feedrate programmed under F until it reaches the programmed depth (Z1). The spindle stops are a specific position there. "Lift off contour" or "Do not lift off contour" can be programmed on expiry of the dwell time.

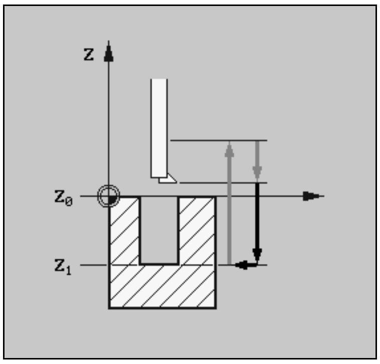
With retraction, withdrawal distance D and the tool orientation angle α can either be defined via machine data or in the parameter screen. If both parameters are pre-assigned via machine data, they do not appear in the parameter screen.

Please refer to the machine manufacturer's instructions.

Drill-ing Boring

➢ Press the "Drilling" and "Boring" softkeys.

Call help display with the HELP key



Help display for boring

Parameters	Description	Unit
T, D, F, S, V	See Sec. "Programming the tool, offset value and spindle speed".	
Lift off Do not lift off	The cutting edge is retracted from the bore edge and then moved back to the retraction plane. The cutting edge is not retracted, but traverses back to the safety clearance in rapid traverse.	
Z1	Depth with reference to Z0 (abs. or inc.)	mm
Z0	Height of workpiece; Z0 is specified in the position pattern ("Positioning" softkey).	mm

DT	Dwell time for relief cut.	S rev
D	Withdrawal (retract) distance (or defined in machine data) - only for retraction	mm
α	Tool orientation angle (or defined via machine data) - only for retraction	Degr.

3.7.5 Tapping

The "Tapping" function is used for tapping an inside thread.

The spindle speed can be controlled with the spindle override during tapping. The feed override is inoperative during this process.

You can select drilling in one cut, chipbreaking or retraction from the workpiece for stock removal.

The tool is moved at rapid traverse to the programmed position, allowing for the retraction plane and safety clearance.
With the spindle stationary, the tool moves at rapid traverse to the retraction plane and then to the safety clearance.
Here the spindle begins to rotate and the spindle speed and feedrate are synchronized.
The tool continues to move at rapid traverse towards the programmed position.

1 cut

1. The tool drills at the programmed spindle speed S or cut rate V as far as the tapping depth Z1.
2. The direction of rotation of the spindle reverses and the tool retracts to the safety clearance at the programmed spindle speed SR or cut rate VR.

Stock removal

1. The tool drills at the programmed spindle speed S or feedrate V as far as the first infeed depth (maximum infeed depth D).
2. The tool retracts from the workpiece to the safety clearance at spindle speed SR or cut rate VR for stock removal.
3. Then the tool is inserted again as far as the 1st infeed depth at spindle speed S or feedrate V and drills to the next infeed depth.
4. Steps 2 and 3 are repeated until the programmed final drilling depth Z1 is reached.
5. The direction of rotation of the spindle reverses and the tool retracts to the safety clearance at spindle speed SR or cut rate VR.

Chipbreaking

1. The tool drills at the programmed spindle speed S or feedrate V as far as the first infeed depth (maximum infeed depth D).
2. The tool retracts by the retraction amount V2 for chipbreaking.
3. The tool then drills to the next infeed depth at spindle speed S or feedrate V.
4. Steps 2 and 3 are repeated until the programmed final drilling depth Z1 is reached.
5. The direction of rotation of the spindle reverses and the tool retracts to the safety clearance at spindle speed SR or cut rate VR.

For tapping with an analog spindle, a floating tapholder is required. This can only be used to dill in one cut.

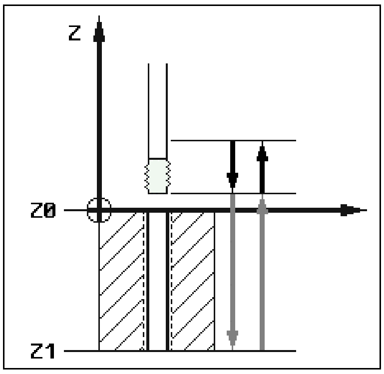
The machine manufacturer may have made specific settings for tapping in a machine data code.

Please refer to the machine manufacturer's instructions.

Drill-ing | Boring > | Tapping

➢ Press the "Drilling", "Boring", and "Tapping" softkeys.

Call help display with the HELP key



Help display for tapping

Parameters	Description	Unit
T, D, S, V	See Sec. "Programming the tool, offset value and spindle speed".	
P	Pitch The pitch is determined by the tool used. MODULUS: Used with endless screws, for example, which extend into a gear wheel. Turns/ ": Used with pipe threads, for example. For values entered in turns/", enter the integer in front of the decimal point in the first parameter field and the figures after the decimal point as a fraction in the second and third field. For example, 13.5 turns/" is entered as follows: P 13 1/ 2 Thrds/"	mm/rev in/rev MODULE Turns/ "

SR	Spindle speed for retraction (not for tapping with a floating tapholder)	rev/min
VR	Cutting rate for retraction (alternative to SR) (not for tapping with a floating tapholder)	m/min
1 cut Stock removal Chipbreaking	The thread is drilled in one cut without stopping. The drill is retracted from the workpiece for stock removal (not for tapping with a floating tapholder) The drill is retracted by the retraction amount V2 for chipbreaking (not for tapping with a floating tapholder)	
Z1	Tapping depth with reference to Z0 (abs. or inc.) Z0 is specified in the position pattern ("Positioning" softkey).	mm
D	Maximum infeed (for stock removal or chipbreaking only)	mm
V2	Retraction amount (for chipbreaking only) Amount by which the drill is retracted for chipbreaking. V2=automatic: The tool is retracted by one revolution.	mm

3.7.6 Thread milling

You can use a form cutter to machine any type of right-hand or left-hand thread.
Threads can be machined as right-hand or left-hand threads and from top to bottom or vice versa.

For metric threads (thread pitch P in mm/rev) ShopMill assigns a value calculated from the thread pitch to the Thread depth K parameter. You can change this value. The default selection must be activated via a machine data code.

Please refer to the machine manufacturer's instructions.

Drill-ing | Thread > | Cut thread

➢ Press the "Drilling", "Thread" and "Cut thread" softkeys.

Internal thread

Sequence:

- Position on thread center point on retraction plane in rapid traverse
- Infeed at rapid traverse to reference plane shifted forward by amount corresponding to safety clearance
- Approach along an approach circle calculated in the control at programmed feedrate
- Approach motion to thread diameter on circular path
- Cut thread along a spiral path in clockwise or counterclockwise direction (depending on whether it is left-hand or right-hand thread)

- Exit motion along a circular path in the same rotational direction at programmed feedrate
- Retract to thread center point and then to retraction plane in rapid traverse

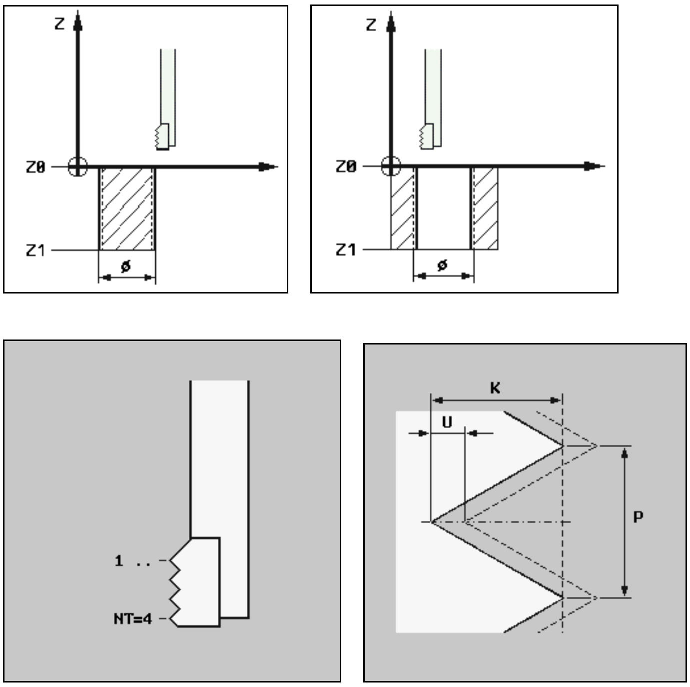
External thread

Sequence:

- Position on starting point in retraction plane at rapid traverse
- Infeed at rapid traverse to reference plane shifted forward by amount corresponding to safety clearance
- Approach along an approach circle calculated in the control at programmed feedrate
- Approach motion to thread diameter on circular path
- Cut thread along a spiral path in clockwise or counterclockwise direction (depending on whether it is left-hand or right-hand thread)
- Exit motion along a circular path in opposite rotational direction at programmed feedrate
- Retract to retraction plane at rapid traverse

Call help display with the 

key



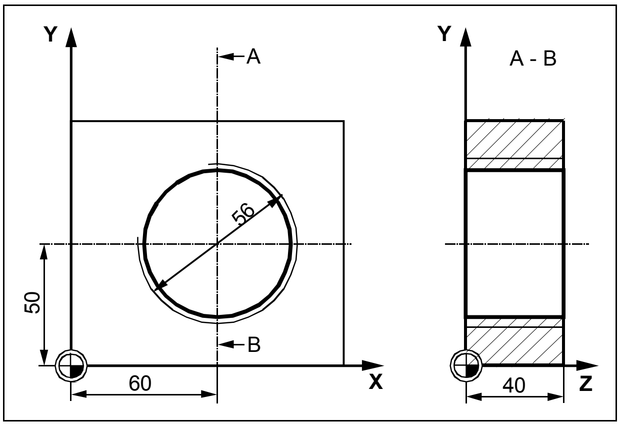
Help displays for thread cutting

Parameters	Description	Unit
Machining type	Roughing Thread cutting up to programmed finishing allowance (U) Finishing	
Direction	Depending on the rotational direction of the spindle, a change in direction also changes the machining direction (climb/conventional). Z0 to Z1: Machining starts at workpiece surface Z0. Z1 to Z0: The machining starts at thread depth, e.g. for blind hole tapping	
Internal thread External thread	An inside thread is cut. An outside thread is cut.	
Left-hand thread Right-hand thread	A left-hand thread is cut. A right-hand thread is cut.	
NT	Number of teeth in a milling insert. Single or multiple toothed milling inserts can be used. The cutting teeth are entered in parameter NT. The motions required are executed by the cycle internally, so that the tip of the bottom tooth on the milling insert corresponds to the programmed end position when the thread end position is reached. Depending on the cutting edge geometry of the milling insert, the retraction path must be taken into account at the base of the workpiece.	
Z1	Thread length	mm
Z0	Height of workpiece; Z0 is specified in the position pattern ("Positioning" softkey).	mm
∅	Nominal thread diameter, example: Nominal diameter of M12 = 12mm	mm
P	Pitch If the cutter has several teeth, the thread pitch is determined by the tool. For a thread pitch entered in turns/", enter the integer in front of the decimal point in the first parameter field and the figures after the decimal point as a fraction in the second and third fields. For example, 13.5 turns/" is entered as follows: P 13 1/ 2 Thrds/"	mm/rev inch/rev MODULE Turns/ "
K	Thread depth	mm
DXY	Infeed per cut Alternately, you can specify the plane infeed as a %, as a ratio --> plane infeed (mm) to milling cutter diameter (mm).	mm %
U	Final machining allowance	mm
$\alpha 0$	Start angle	Degr.

Programming example for thread cutting

Cut circular pocket in a solid blank and cut a thread.
The milling tool cannot cut across center. The circular pocket will therefore have to be predrilled with a Ø22mm drill. The milling tool can then be inserted centrally.

Using position patterns, the positions of the above-mentioned cycles can be programmed (see Sec. "Using position patterns in milling").



Workshop drawing of circular pocket with thread

N10	CENTERING		T=center F250/min S900rev. ø5
N15	DRILL		T=drill22 F80/min S400rev. Z1=42inc
N20	Circ. pocket	▽	T=12 F500/min S600rev. Z1=40inc ø50
N25	Inside thread	▽	T=thread56 F100/min S400rev. Z1=40 ø56
N30	001: Positions		Z0=0 X0=60 Y0=50

Extract from machining plan; cut a circular pocket with thread

3.7.7 Drill and thread milling

You can use a drill and thread milling cutter to manufacture an internal thread with a specific depth and pitch in one operation. This means that you can use the same tool for drilling and thread milling, a change of tool is superfluous.
The thread can be machined as a right- or left-hand thread.

Sequence:

- The tool travels at rapid traverse to the safety distance.
- If pre-drilling is required, the tool travels at a reduced drilling feedrate to the pre-drilling depth defined in a machine data.

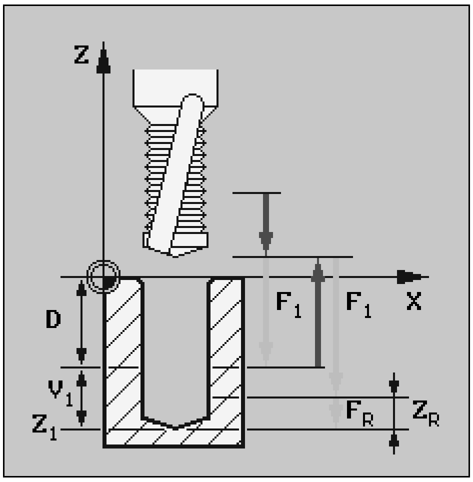
Please refer to the machine manufacturer's instructions.

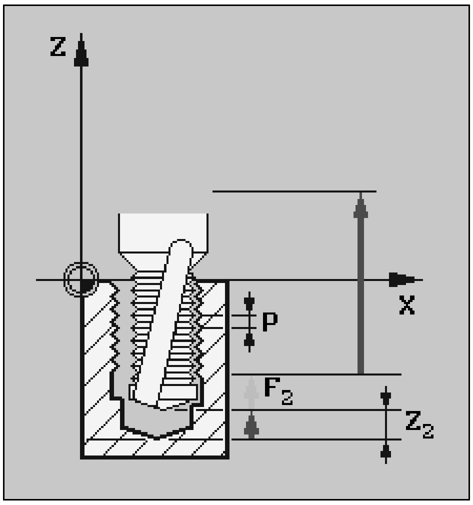
- The tool drills to the first drilling depth D with drilling feedrate F1. If the final drilling depth is not reached, the tool will travel back to the workpiece surface in rapid traverse for stock removal. Then the tool will travel at rapid traverse to a position 1mm above the drilling depth previously achieved - allowing it to continue drillling at drill feedrate F1 at the next infeed.
- If another feedrate FR is required for through-boring, the residual drilling depth ZR is drilled with this feedrate.
- If required, the tool travels back to the workpiece surface for stock removal before thread milling at rapid traverse.
- The tool travels directly to the starting position for thread milling.
- The thread milling is carried out (climb milling, conventional milling or conventional milling + climb milling) with milling feedrate F2. The thread milling acceleration path and deceleration path is traversed in a semicircle with concurrent infeed in the tool axis.

Drill- ing | Thread > | Cut thread

➢ Press the "Drilling", "Thread" and "Cut thread" softkeys.

Call help display with the HELP key





Displays for drill and thread milling cutter

Parameters	Description	Unit
T, D, S, V	See Sec. "Programming the tool, offset value and spindle speed".	
F1	Drilling feed	mm/min mm/rev
Z1	Drilling depth	mm
D	Maximum infeed	mm
DF	Percentage for each additional infeed DF=100: Amount of infeed remains constant DF<100: Amount of infeed is reduced in direction of final drilling depth Z1. Example: Last infeed 4 mm; DF 80% next infeed = 4 x 80% = 3.2 mm infeed after next = 3.2 x 80% = 2.56 mm etc.	%
V1	Minimum infeed Parameter V1 is provided only if DF<100 has been programmed. If the infeed increment becomes minimal, a minimum infeed can be programmed in parameter "V1". V1 < Amount of infeed: The tool is inserted by the infeed increment. V1 < Amount of infeed: The tool is inserted by the infeed value programmed under V1.	mm

Pre-drilling	When drilling, start initially with a reduced feedrate. The reduced drilling feedrate results as follows: Drilling feedrate F1 < 0.15mm/rev: Pre-drilling feedrate = 30% of F1 Pre-drilling feedrate F1 ≥ 0.15mm/rev: Pre-drilling feedrate = 0.1mm/rev	
Through-drilling	When drilling the residual drilling depth ZR drill with feedrate FR.	
ZR	Residual drilling depth (for through-drilling only)	mm
FR	Feedrate through-drilling (for through-drilling only)	mm/min mm/rev
Stock removal	Return to workpiece surface for stock removal before thread milling.	
Thread	Right-hand thread Left-hand thread	
F2	Milling feed	mm/min mm/tooth
P	Pitch For a thread pitch entered in turns/", enter the integer in front of the decimal point in the first parameter field and the figures after the decimal point as a fraction in the second and third fields. For example, 13.5 turns/" is entered as follows: P 13 1/ 2 Thrds/"	in/rev Turns / "
Z2	Retraction before thread milling Z2 is for defining the thread depth in the direction of the tool axis. Z2 is relative to the tool tip.	mm
∅	Nominal thread diameter	mm
Machining direction	Climb milling: Mill thread in one cycle. Conventional milling: Mill thread in one cycle. Conventional milling + climb milling: Mill thread in two cycles: rough cutting is performed by conventional milling with defined allowances, then finish cutting is performed by climb milling with milling feedrate FS.	
FS	Milling feed finish cutting (for conventional milling + climb milling only)	mm/min mm/tooth

3.7.8 Positioning on freely programmable positions and position patterns

After you have programmed the machining technologies, you must program the positions. ShopMill offers a variety of positioning patterns, i.e.:

- Freely programmable positions
- Position on a line, on a matrix or box
- Position on a full or pitch circle

Several position patterns can be programmed in succession (up to 20 technologies and position patterns together). They are traversed in the order in which you program them.
The programmed technologies and subsequently programmed positions are automatically chained by the control.

Machining sequence and tool travel path

The first tool in the program traverses all programmed positions, e.g. center all positions. Machining of the positions always starts at the reference point.
In the case of a matrix, machining is performed first in the direction of the 1st axis and then back and forth. The box and hole circle are machined counterclockwise. The second tool in the program then machines all programmed positions, etc. This process is repeated until every programmed drilling operation has been performed at every programmed position.
Inside a position pattern or on the approach from one position pattern to the next, the tool is retracted to safety clearance in the case of optimized retraction, or otherwise to the return plane (see also Sec. "Creating new program; defining a blank"). The new position is then approached at rapid traverse.
If the position pattern consists of only one position, the tool is retracted to the retraction plane after machining.

Including/skipping positions

You can include or skip any positions.

Rotary axis

If an A or B axis is set up on your machine, this is supported during drilling (any position pattern, full circle, and pitch circle).

Please also refer to your machine manufacturer's instructions.

In the A or B axis is present, it makes sense to define a work offset with
X = end face of the cylinder
Y = center point of the cylinder
Z = center point of the cylinder
The "cylinder" in this case refers to any part that is clamped in the A/B axis.

Cylinder surface transformation

When working with the cylinder surface transformation, please note that the A axis or B axis is not supported in all cases. Programming of any position in the XYA plane is not possible while cylinder surface transformation is active.
A work offset in the rotary axis A or B is effective even while cylinder surface transformation is active.

Select with softkey

Positions

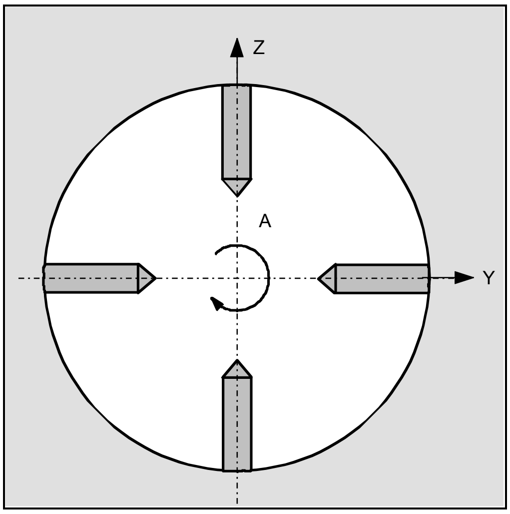
3.7.9 Freely programmable positions

This pattern allows you to program positions freely, i.e. rectangular or polar, in the X/Y, X/A, and XYA plane.
Individual positions are approached in the order in which you program them. Press softkey "Delete all" to delete all positions programmed in X/Y.

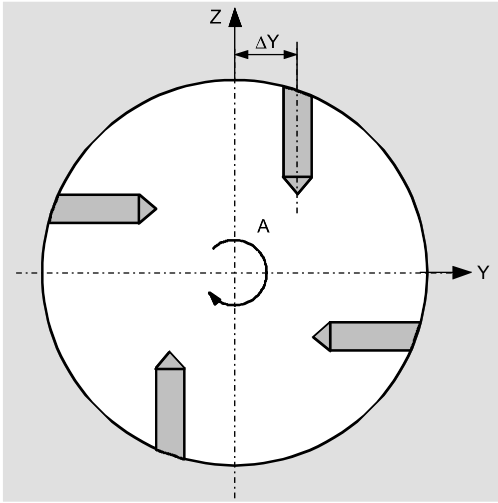
Rotary axis

- XA plane

You program in XA to prevent the Y axis moving during machining, if required.
To ensure that the drill-holes point to the center of the "Cylinder", you must first position the Y axis centrally above the "Cylinder".



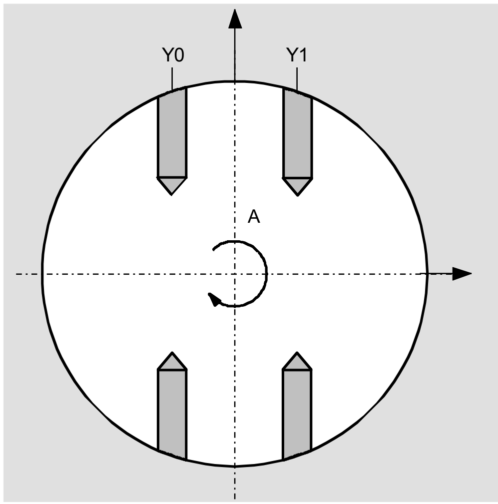
Drill-holes point toward the center



Y axis is not central above the cylinder (ΔY)

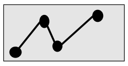
- XYA plane

You program in XYA if the Y axis is required to move during machining. A value can be specified for each position.
In addition to the possibilities of XA, the following is also possible, for example.



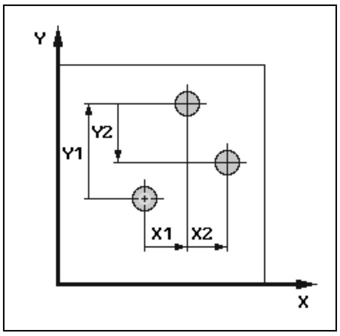
Y axis is traversed (Y0, Y1)

Select with softkey 

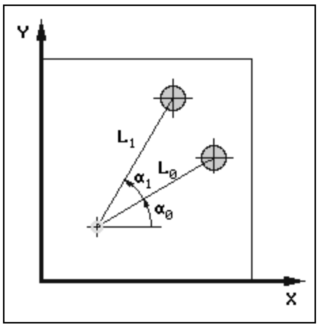
Positions > 

Call help display with the 

key



Help display for "Freely programmable positions, rectangular"



Help display for "Freely programmable positions, polar"

Parameters	Description XY (without A/ or B axis support)	Unit
Rectangular/ polar	Programming with rectangular or polar dimensions.	
Z0	Height of workpiece (abs. or inc.)	mm
X0 Y0	1. position of the hole in X (abs. or inc.) 1. position of the hole in Y (abs. or inc.)	mm mm
Rectangular: X1 ... X8 Y1 ... Y8	 Other positions in the X axis (abs. or inc.) Other positions in the Y axis (abs. or inc.) If you want to program further positions, store the ones you have already programmed and then open the parameter input form again by pressing softkey "Any positions".	 mm mm
Polar: L1... L7 α1 ... α7	 Position distance (abs.) Angle of rotation of line in relation to the X axis. Positive angle: Line is rotated counterclockwise. Negative angle: Line is rotated clockwise. If you want to program further positions, store the ones you have already programmed and then open the parameter input form again by pressing softkey "Any positions".	 mm Degr.

Parameters	Description (with A/ or B axis support)	Unit
Z0	Height of workpiece (abs. or inc.)	mm
XA:	(B can be used everywhere instead of A; Y can be used instead of X)	
X0 A0	1. position of the hole in X (abs. or inc.) 1. position of the hole in A (abs.)	mm Degr.
X1 ... X8 A1 ... A8	Other positions in the X axis (abs. or inc.) Other positions in the A axis (abs. or inc.) If you want to program further positions, store the ones you have already programmed and then open the parameter input form again by pressing softkey "Any positions".	mm Degr.
XYA:	(B can be used everywhere instead of A)	
X0 Y0 A0	1. position of the hole in X (abs. or inc.) 1. position of the hole in Y (abs. or inc.) 1. position of the hole in A (abs.)	mm mm Degr.
X1 ... X5 Y1 ... Y5 A1 ... A5	Other positions in the X axis (abs. or inc.) Other positions in the Y axis (abs. or inc.) Other positions in the A axis (abs. or inc.) If you want to program further positions, store the ones you have already programmed and then open the parameter input form again by pressing softkey "Any positions".	mm mm Degr.

3.7.10 Line position pattern

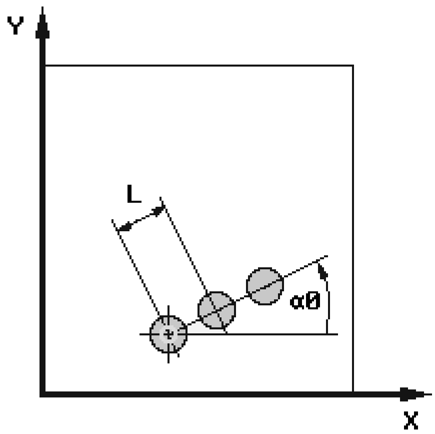
You can use this function to program any number of positions spaced at the same distance along a line.

Select with softkey

Drill-ing | Positions >

Position the cursor in the "Line/matrix/box" field. With the "Alternat." softkey you can select the "Line" position pattern.

Call help display with the HELP key



Help display for "Line"

Parameters	Description	Unit
Z0	Height of workpiece (abs. or inc.) This position must be programmed absolutely in the first call.	mm
X0	Reference point (first position) This position must be programmed absolutely in the first call.	mm
Y0	Reference point (first position) This position must be programmed absolutely in the first call.	mm
α0	Angle of rotation of line in relation to the X axis. Positive angle: Line is rotated counterclockwise. Negative angle: Line is rotated clockwise.	Degr.
L	Position spacing.	mm
N	Number of positions.	

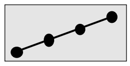
3.7.11 Matrix position pattern

You can use this function to program any number of positions spaced at an equal distance along one or several parallel lines.

If you want to program a rhombus-shaped matrix, enter the angle αX or αY.

Select with softkey

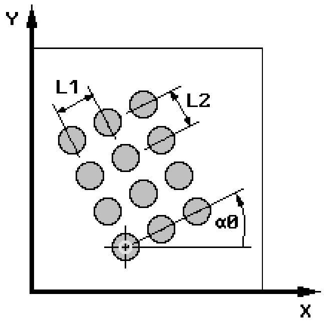
Positions >



Position the cursor in the "Line/matrix/box" field. With the "Alternat." softkey you can select the "Matrix" position pattern.

Call help display with the 

key



Help display for "Matrix"

Parameters	Description	Unit
Z0	Height of workpiece (abs. or inc.) This position must be programmed absolutely in the first call.	mm
X0	Reference point (first position) This position must be programmed absolutely in the first call.	mm
Y0	Reference point (first position) This position must be programmed absolutely in the first call.	mm
α0	Angle of rotation of matrix. Positive angle: Matrix is rotated counterclockwise. Negative angle: Matrix is rotated clockwise.	Degr.
αX	Shear angle of matrix relative to X axis. Positive angle: Matrix shears in CCW direction. Negative angle: Matrix shears in CW direction.	Degr.
αY	Angle of rotation of matrix relative to Y axis Positive angle: Matrix is rotated counterclockwise. Negative angle: Matrix is rotated clockwise.	Degr.
L1 L2	Position spacing in X direction Position spacing in Y direction	mm
N1 N2	Number of positions in X direction Number of columns in Y direction	

3.7.12 Box position pattern

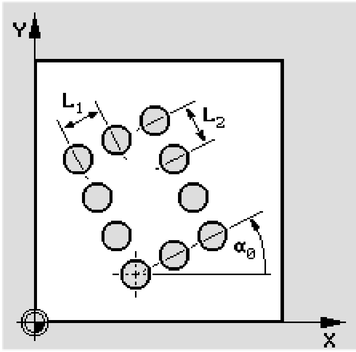
You can use this function to program any number of positions spaced at an equal distance along on a box. The spacing may be different on both axes.
If you want to program a rhombus-shaped box, enter the angle αX or αY.

Select with softkey

Drill-ing | Positions

Position the cursor in the "Line/matrix/box" field. With the "Alternat." softkey you can select the "Box" position pattern.

Call help display with the HELP key

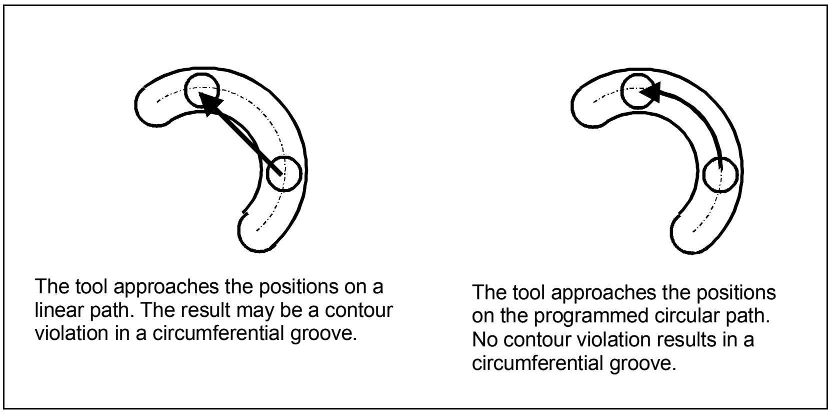


Help display for "box"

Parameters	Description	Unit
Z0	Height of workpiece (abs. or inc.) This position must be programmed absolutely in the first call.	mm
X0	Reference point (first position) This position must be programmed absolutely in the first call.	mm
Y0	Reference point (first position) This position must be programmed absolutely in the first call.	mm
α0	Angle of rotation of box Positive angle: Box is rotated counterclockwise. Negative angle: Box is rotated clockwise.	Degr.
αX	Shear angle of box relative to X axis. Positive angle: Box shears in CCW direction. Negative angle: Box shears in CW direction.	Degr.
αY	Shear angle of box relative to Y axis. Positive angle: Box shears in CCW direction. Negative angle: Box is rotated clockwise.	Degr.
L1 L2	Position spacing in X direction Position spacing in Y direction	mm
N1 N2	Number of positions in X direction Number of columns in Y direction	

3.7.13 Full circle position pattern

This function can be used to program drill holes on a circle with a defined radius. The basic angle of rotation ($\alpha 0$) for the 1st position is relative to the X axis. The control calculates the angle of the next hole position as a function of the total number of holes. The angle it calculates is identical for all positions.
If the A or B axis is used, the angle refers to a set reference point (A0).
You can use the XA selection if use of the rotary axis on the machine is required.
The Y axis is not traversed, i.e. the Y axis must first be positioned centrally over the cylinder.
The tool can approach the next position along a linear or circular path.

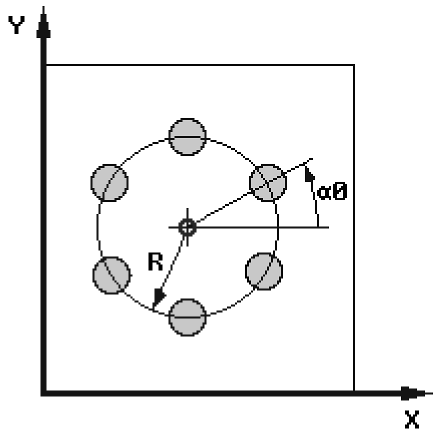


Approaching positions on a linear or circular path

Select with softkey

If you position the cursor on the "Full/pitch circle" field, you can toggle between the two options using the "Alternat." softkey.

Call help display with the  **key**



Help display for "Full circle of holes"

Parameters	Description XY (without A/B axis)	Unit
Z0	Height of workpiece (abs. or inc.)	mm
X0	X position of full circle center point (abs. or inc.)	mm
Y0	Y position of full circle center point (abs. or inc.)	mm
α0	Basic angle of rotation; angle of 1st hole in relation to X axis. Positive angle: Full circle is rotated counterclockwise. Negative angle: Full circle is rotated in clockwise direction.	Degr.
R	Radius of full circle	mm
N	Number of positions on full circle	
FP	Feed for positioning on a circular path.	mm/min
Positioning	Linear: Next position is approached linearly at rapid traverse. Circular: Next position is approached at the programmed feedrate (FP) along a circular path.	

Parameters	Description XA (with A/B axis)	Unit
Z0	Height of workpiece surface (abs. or inc.)	mm
X0	Reference position (abs. or inc.)	mm
A0	Start angle (abs.) Angle of 1st hole with reference to X axis. Positive angle: Full circle is rotated counterclockwise. Negative angle: Full circle is rotated in clockwise direction.	Degr.
N	Number of positions on full circle	

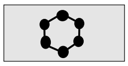
3.7.14 Pitch circle position pattern

This function can be used to program holes on a pitch circle with a defined radius.
The tool can approach the next position along a linear or circular path. (for XY selection only)
(please see "Full circle" for detailed description).

You can use the XA selection if use of the rotary axis on the machine is required.
The Y axis is not traversed, i.e. the Y axis must first be positioned centrally over the cylinder.

Select with softkey

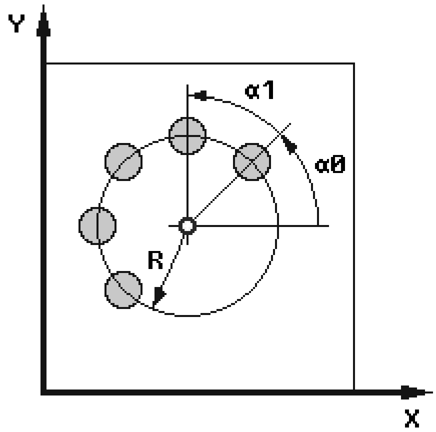
Positions >



If you position the cursor on the "Full/pitch circle" field, you can toggle between the two options using the "Alternat." softkey.

Call help display with the 

key



Help display for "Pitch circle"

Parameters	Description normal/XY (without A/B axis)	Unit
Z0	Height of workpiece (abs. or inc.)	mm
X0	X position of pitch circle center point (abs. or inc.)	mm
Y0	Y position of pitch circle center point (abs. or inc.)	mm
α0	Basic angle of rotation; angle of 1st position in relation to X axis.	Degr.
α1	Advance angle; after the first hole has been drilled, all further positions are advanced by this angle. Positive angle: Further positions are rotated counterclockwise. Negative angle: Further positions are rotated clockwise.	Degr.
R	Radius of pitch circle	mm
N	Number of positions (holes) on the pitch circle	
FP	Feed for positioning on a circular path.	mm/min
Positioning	Linear: Next position is approached linearly at rapid traverse. Circular: Next position is approached at the programmed feedrate (FP) along a circular path.	

Parameters	Description XA (with A/B axis)	Unit
Z0	Height of workpiece surface (abs. or inc.)	mm
X0	Reference position (abs. or inc.)	mm
A0	Start angle (abs.) Angle of 1st position relative to X axis.	Degr.
A1	Advance angle (abs.) after the first hole has been drilled, all further positions are advanced by this angle. Positive angle: Further positions are rotated counterclockwise. Negative angle: Further positions are rotated clockwise.	Degr.
N	Number of positions on pitch circle	

3.7.15 Including and skipping positions

You can skip any positions in the following position patterns:

- Position pattern line
- Position pattern matrix
- Position pattern box
- Position pattern rhombus
- Position pattern full circle (XY only)
- Position pattern pitch circle (XY only)

The suppressed positions are skipped during machining.

Including / skipping any positions

Drill-ing | Positions > | Skip pos. >

➢ Select the required position pattern and press the "Skip pos." softkey.

The "Skip positions" window opens on top of the input form of the position pattern.
The number of the current position is displayed along with its status (on/off) and its coordinates (X, Y).
The diagram shows the skipped positions dotted. The current position is highlighted by a circle.

➢ Enter the number of the point you want to skip in the "Position" field (in accordance with the machining sequence).

- OR -

Position +

➢ Press the "Position +" softkey to select the next position (in the machining sequence).

- OR -

Position -

➢ Press the "Position -" softkey to select the previous position (opposite direction to the machining sequence).

Alternat.

➢ Press the "Alternat." softkey to include or skip the current position.

Including or skipping all positions at once

Skip all

➢ Press the "Skip all" softkey to skip all positions.

Include all

➢ Press the "Include all" softkey to include all positions again.

3.7.16 Obstacle

Function

If there is an obstacle between 2 position patterns, it can be crossed. The height of the obstacle can be programmed absolutely or incrementally.

If all positions in the 1st pattern have been machined, the tool axis travels in rapid traverse to a height corresponding to the obstacle height + safety clearance. The new position is approached in rapid traverse at this height. The tool axis then approaches a position corresponding to Z0 of the position pattern + safety clearance.

Select with softkey

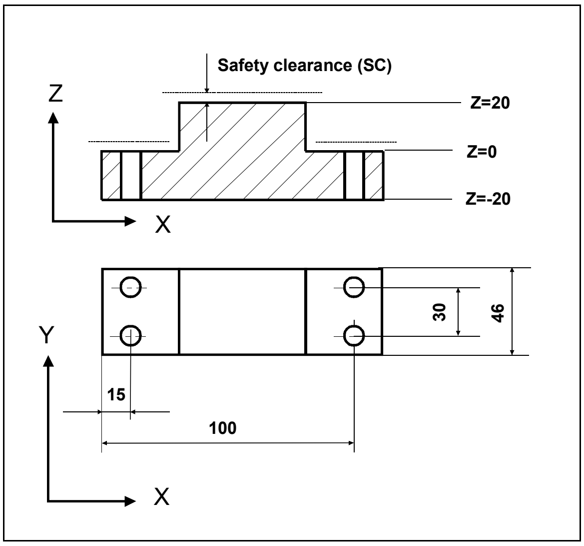
Drill-ing | Positions > | Obstacle

Note

Obstacles are registered only if they lie between 2 position patterns. If the tool change point and the programmed retraction plane are positioned below the obstacle, the tool travels to the retraction plane height and on to the new position without taking the obstacle into account. The obstacle must not be higher than the retraction plane.

Programming example

Drilling 4 positions with an obstacle in-between.
The holes are first centered and then drilled. When you have programmed the first two positions at X=15, you need to program the obstacle. The remaining positions are then programmed at X=100.



Workshop drawing

N10	CENTERING	T=4 F250/min S900rev. ø3
N15	DRILL	T=DRILL10 F80/min S600rev. Z1=22ink
N20	001: Positions	Z0=0 X0=15 Y0=8 X1=15 Y1=38
N25	Obstacle	Z20
N30	002: Positions	Z0=0 X0=100 Y0=8 X1=100 Y1=38

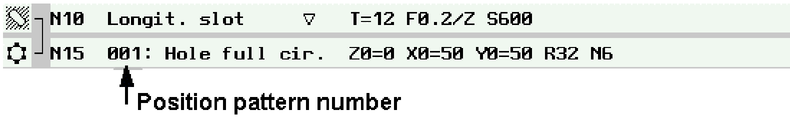
Extract from machining plan for "Obstacle" programming example

3.7.17 Repeating positions

Function

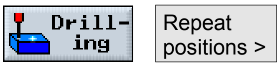
If you want the tool to re-approach positions that you have already programmed, the "Repeat positions" function is a quick and easy solution.

You must specify the number of the position pattern. This is a number assigned automatically by ShopMill. You will find it inserted after the block number in the machining plan.

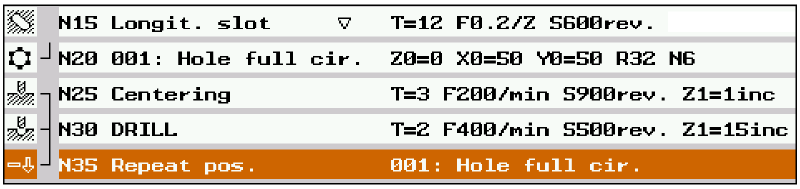


Extract from machining plan, position pattern number=001

Select with softkey



After you have entered the position pattern number, e.g. 1, press the "Accept" softkey. The position pattern you have selected is then approached again.



Extract from machining plan; repeat positions in block no. 60

3.7.18 Programming examples for drilling

Drilling at different heights

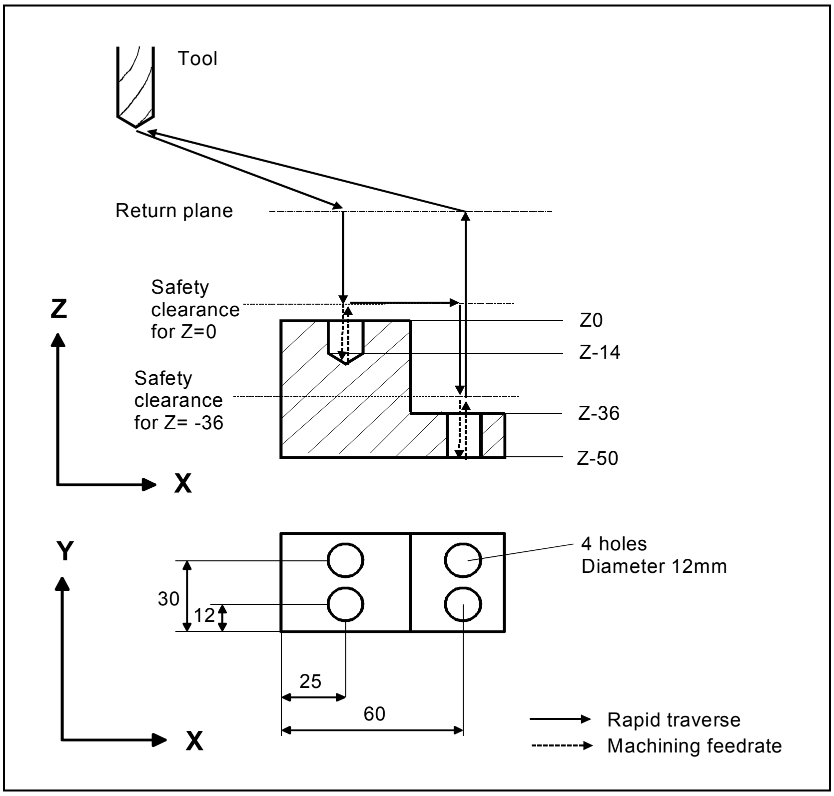
Machining task: You have already cut a recess in a workpiece. You now want to machine blind and through holes of Ø 12 mm on this workpiece with different machining planes.

Programming:

Center the 4 holes

Deep drill the blind holes with stock removal

Deep drill the through holes with chipbreaking



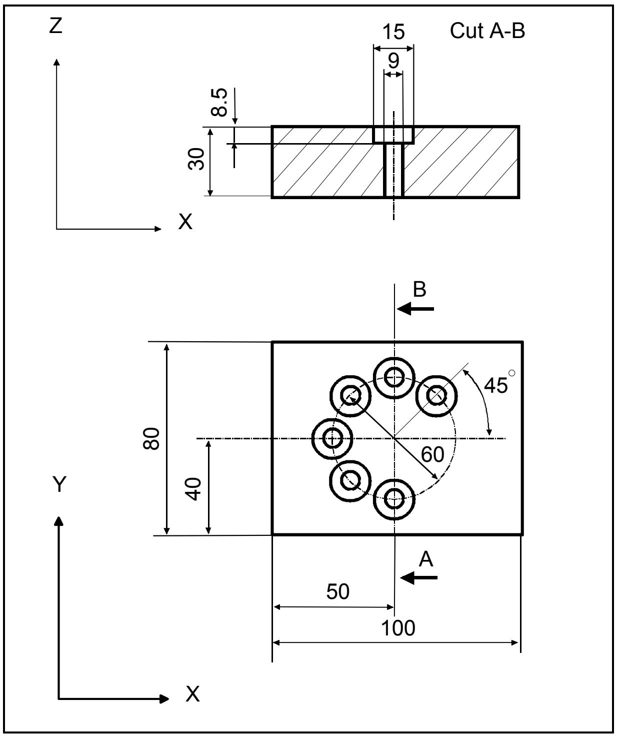
Workshop drawing

N10	CENTERING	T=center F250/min S900rev. Z1=2ink
N15	001: Positions	Z0=0 X0=25 Y0=12 X1=25 Y1=30
N20	002: Positions	Z0=-36 X0=60 Y0=12 X1=60 Y1=30
N25	Deep hole dr.	T=DRILL12 F80/min S600rev. Z1=14ink
N30	Repeat pos.	001: Positions
N35	Deep hole dr.	T=DRILL12 F80/min S600rev. Z1=-52
N40	Repeat pos.	002: Positions

Extract from machining plan

Drilling with a counterbore

You want to machine through holes with screw head recesses around a pitch circle on a workpiece.
When you program the counterbore, you must select offset value D2 (see Sec. "Creating a tool offset block for tool edge ½").



Workshop drawing

N5	Centering	T=center F200/min S600rev. ø3
N10	DRILL	T=drill9 F100/min S400rev. Z1=31inc
N15	DRILL	T=spot_facer F60/min S400rev. Z1=8inc
N20	001: Hole pitch cir	Z0=0 X0=50 Y0=40 R30 N6

Extract from machining plan

3.8 Milling

3.8.1 Face milling

You can use this cycle to face mill any workpiece. A rectangular surface is always machined. The rectangle results from corner points 1 and 2 that are pre-assigned with the values of the blank part dimensions from the program header.
Workpieces with and without limits can be face-milled.
To machine a work piece with four limits select the pocket cycles.

The cycle makes a distinction between roughing and finishing:
Roughing:
- Several material removal operations on surface
- Tool turns above the workpiece edge

Finishing:
- First material removal operation on surface
- Tool turns at safety distance in the X/Y plane
- Retraction of mill

Depth infeed always takes place outside the workpiece.

For a workpiece with edge breaking, select the rectangular spigot cycle.

In face milling, the effective tool diameter for a tool of type "Milling cutter" is stored in machine data.

Please refer to the machine manufacturer's instructions.

Start point

For vertical machining the start point is always above or below. For horizontal machining it is right or left.
Machining is performed from outside to inside, if possible.
The starting point is marked in the help display.

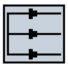
Select with softkey

Mill-ing | Face milling

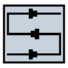
Selecting the machining direction

Toggle in the "Direction" field using the "Alternat." softkey until the icon for the required machining direction appears.

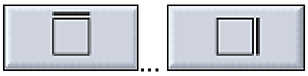
- Same direction of machining



- Alternating direction of machining

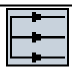
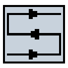


Selecting limits



Press the softkey for each of the required limits.

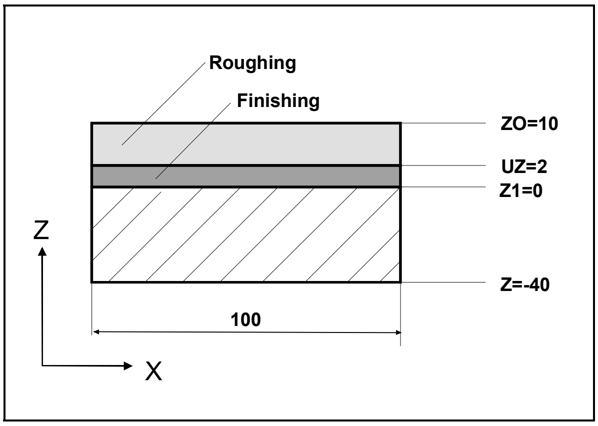
The selected limits are shown in the help display and in the broken-line graphics.

Parameters	Description	Unit
Machining type	Roughing: Face milling up to programmed finishing allowance (UZ). Finishing: The surface is milled once in the plane. The tool is retracted after every cut.	
Machining direction	 Same direction of machining.  Alternating direction of machining	
X0, Y0 Z0	Corner point 1 of surface in X or Y direction (abs. or inc.) Height of blank (abs. or inc.)	mm
X1 Y1 Z1	Corner point 2 of surface in X direction (abs. or inc.) Corner point 2 of surface in Y direction (abs. or inc.) Height of finished part (abs. or inc.)	mm
DXY	Max. infeed in the XY plane (dependent on mill diameter) Alternately, you can specify the plane infeed as a %, as a ratio → plane infeed (mm) to milling cutter diameter (mm).	mm %
DZ	Max. infeed in Z direction	mm
UZ	Final machining allowance	mm

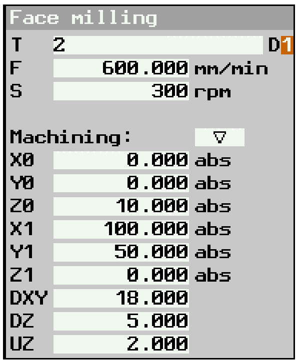
The same finishing allowance must be entered for both roughing and finishing. The finishing allowance is used to position the tool for retraction.

Programming example
Face milling

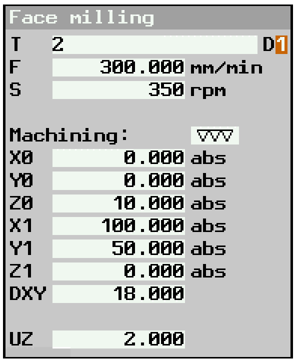
You want to cut to a depth of 10mm on a workpiece surface. 8mm must be removed in a rough cut and 2mm in a finish cut. The cutter diameter is 40mm. Blank dimensions: X0=0, Y0=0, Z0=10, X1=100 abs., Y1=50 abs., Z1=0 abs



Face milling: Roughing and finishing



Face milling, roughing



Face milling, finishing

Extract from machining plan; Roughing and finishing in face milling

3.8.2 Rectangular pocket

If you want to mill a rectangular pocket, use the "Rectangular pocket" function.

The following machining methods are available:

- Mill rectangular pocket from solid material.
- Predrill rectangular pocket in the center first if, for example, the milling cutter does not cut across center (program the drilling, rectangular pocket and position program blocks one after another).
- Machine pre-machined rectangular pocket (see "Machining" parameter).

Depending on the dimensions of the rectangular pocket in the workpiece drawing, you can select a corresponding reference point for the rectangular pocket.

Approach/retraction

1. The tool approaches the center point of the pocket at rapid traverse at the height of the retraction plane and adjusts to the safety clearance.
2. The tool is inserted into the material according to the chosen strategy.
3. The pocket is always machined with the chosen machining type from inside out.
4. The tool moves back to the safety clearance at rapid traverse.

Machining type

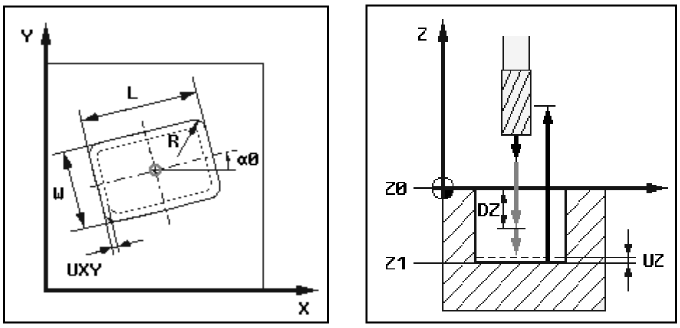
You can select the machining mode for milling the rectangular pocket as follows:

- Roughing
  During roughing, the individual planes of the pocket are machined one after the other from center point until depth Z1 is reached.
- Finishing
  In "Finishing" mode, the edge is always machined first. The pocket edge is approached on the quadrant which joins the corner radius. In the last infeed, the base is finished from the center out.
- Edge finishing
  Edge finishing is performed in the same way as finishing, except that the last infeed (finish base) is omitted.
- Chamfer
  Chamfering involves edge breaking at the upper edge of the pocket.

Select with softkey

Mill-ing | Pocket > | Rectangular pocket

Call help display with the HELP key



Help display for milling a rectangular pocket

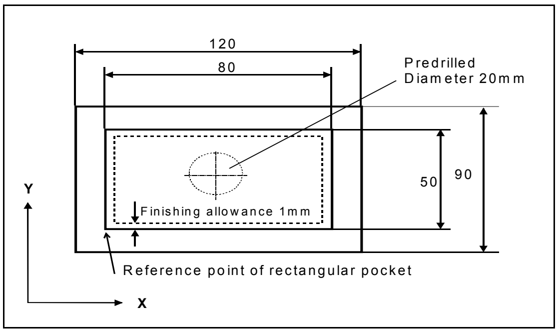
If you want to mill a chamfer and the corner radius was R = 0 during finishing, you must specify the radius of the finishing milling tool in parameter R during chamfering.

Parameters	Description	Unit
T, F, V	See Sec. "Programming the tool, offset value and spindle speed".	
Position of reference point	5 different positions for the reference point can be selected: • Pocket center • Lower left-hand corner • Lower right-hand corner • Upper left-hand corner • Upper right-hand corner The reference point (highlighted in yellow) is displayed in the Help screen.	
Machining type	Roughing Finishing Finishing edge Chamfer	
Single pos. Pos. pattern	A rectangular pocket is machined at the programmed position (X0, Y0, Z0). Several rectangular pockets are machined in a position pattern (e.g. full circle, pitch circle, matrix, etc.).	
 X0 Y0 Z0	The positions refer to the reference point: Position in X direction (single position only), abs. or inc. Position in Y direction (single position only), abs. or inc. Workpiece height (single position only), abs. or inc.	 mm mm mm
W	Pocket width	mm
L	Pocket length	mm
R	Radius at pocket corners	mm
$\alpha 0$	Angle of rotation of pocket in relation to X axis.	Degr.
Z1	Depth of pocket in relation to Z0 (abs. or inc.) (not for chamfer)	mm
DXY	Max. infeed in plane (XY direction) Alternatively, you can specify the plane infeed as a %, as a ratio → plane infeed (mm) to milling cutter diameter (mm). (not for chamfer)	mm %

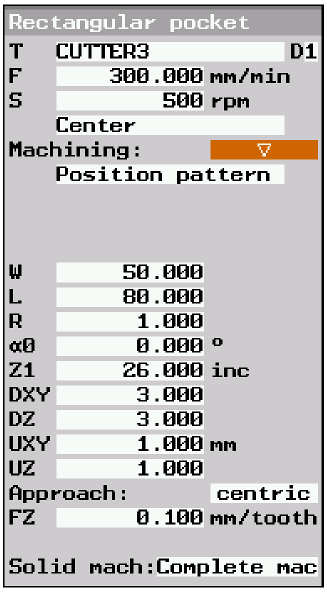
DZ	Max. depth infeed (Z direction) (not for chamfer)	mm
UXY	Finishing allowance in plane (pocket edge) (not for chamfer)	mm
UZ	Finishing allowance in depth (pocket base) (not for chamfer)	mm
Insertion	You can select one of several insertion strategies: **Helical: Insertion along helical path** The cutter center point traverses along the helical path determined by the radius and depth per revolution. If the depth for one infeed has been reached, a full circle motion is executed to eliminate the inclined insertion path. **Oscillation: Insertion with oscillation along center axis of pocket** The cutter center point oscillates along a linear path until it reaches the depth infeed. When the depth has been reached, the path is traversed again without depth infeed in order to remove the slope caused by insertion. **Center: Insert vertically in center of pocket** The tool executes the calculated depth infeed vertically in the center of the pocket. Note: This setting can be used only if the cutter can cut across center or if the pocket has been predrilled.	
EP	Max. insertion gradient (only for helical insertion) The gradient of the helix may be smaller in some geometric conditions.	mm/rev
ER	Insertion radius (only for helical insertion) The radius must not be larger than the cutter radius, otherwise material will remain. Also make sure the pocket is not violated.	mm
EW	Insertion angle (for insertion with oscillation only)	Degr.
FZ	Depth infeed rate (for insertion in center only)	mm/min mm/tooth
Remove stock	**Complete mach.:** The pocket must be milled from a solid workpiece (e.g. casting). **Remachining:** A small pocket or hole has already been machined in the workpiece. This needs to be enlarged in one or several axes. You must program parameters AZ, W1 and L1 for this purpose.	
FS	Chamfer width (for chamfer only), inc.	mm
ZFS	Insertion depth tool tip (for chamfer only), abs. or inc.	mm
AZ	Depth of premachined pocket (for remachining only)	mm
W1	Width of premachined pocket (for remachining only)	mm
L1	Length of premachined pocket (for remachining only)	mm

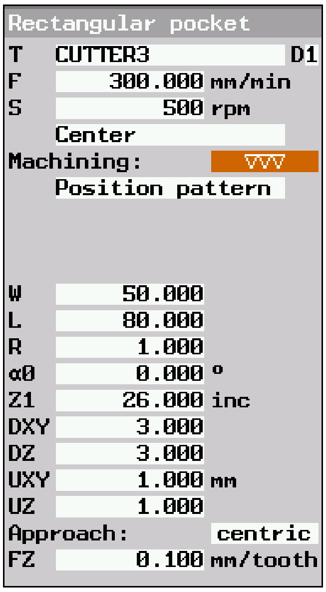
Programming example

You wish to cut a rectangular pocket, starting with a rough cut operation and followed by a finish cut. Since the cutting tool you are using cannot cut across center, the workpiece needs to be predrilled first with a drill of Ø 20 mm.



Workpiece drawing of the rectangular pocket





Rough cut a rectangular pocket *Finishing a rectangular pocket*

Extract from machining plan; predrilling and milling a rectangular pocket

3.8.3 Circular pocket

Use the "Circular pocket" function if you want to mill any kind of circular pocket.

The following machining methods are available:

- Mill circular pocket from solid material.
- Predrill circular pocket in the center first if, for example, the milling cutter does not cut across center (program the drilling, circular pocket and position program blocks one after another).
- Machine pre-machined circular pocket (see "Machining" parameter).

Approach/retraction

1. The tool approaches the center point of the pocket at rapid traverse at the height of the retraction plane and adjusts to the safety clearance.
2. The tool is inserted into the material according to the chosen strategy.
3. The pocket is always machined with the chosen machining type from inside out.
4. The tool moves back to the safety clearance at rapid traverse.

Machining type

You can select the machining mode for milling the circular pocket as follows:

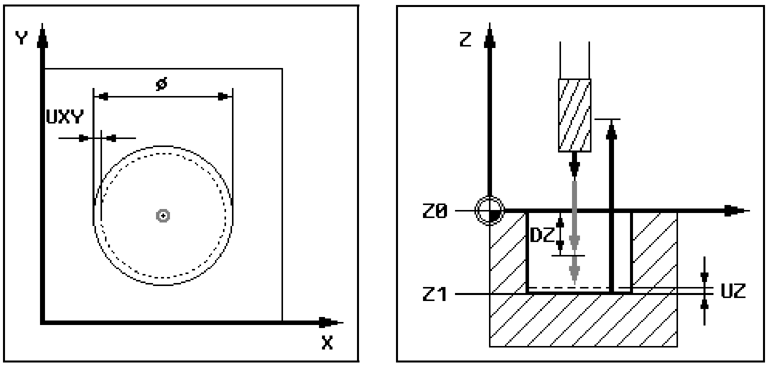
- Roughing
  During roughing, the individual planes of the pocket are machined one after the other from center point until depth Z1 is reached.
- Finishing
  In "Finishing" mode, the edge is always machined first. The pocket edge is approached on the quadrant, which joins the pocket radius. In the last infeed, the base is finished from the center out.
- Edge finishing
  Edge finishing is performed in the same way as finishing, except that the last infeed (finish base) is omitted.
- Chamfer
  Chamfering involves edge breaking at the upper edge of the pocket.

Select with softkey

Mill-ing | Pocket > | Circular pocket

Call help display with

the  **key**



Help display for milling a circular pocket

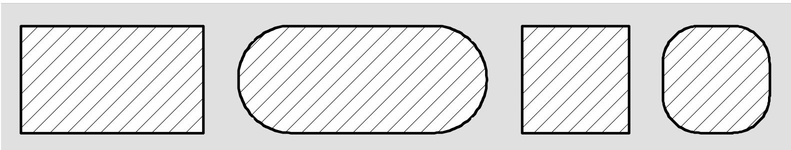
Parameters	Description	Unit
T, F, V	See Sec. "Programming the tool, offset value and spindle speed".	
Machining type	Roughing Finishing Finishing on edge Chamfer	
Single pos. Pos. pattern	A circular pocket is machined at the programmed position (X0, Y0, Z0). Several circular pockets are machined in a position pattern (e.g. full circle, pitch circle, matrix, etc.).	
 X0 Y0 Z0	The positions refer to the center point of the circular pocket: Position in X direction (single position only), abs. or inc. Position in Y direction (single position only), abs. or inc. Workpiece height (single position only), abs. or inc.	 mm mm mm
∅	Diameter of pocket	mm
Z1	Depth of pocket in relation to Z0, abs. or inc. (not for chamfer)	mm
DXY	Max. infeed in plane (XY direction) Alternatively, you can specify the plane infeed as a %, as a ratio → plane infeed (mm) to milling cutter diameter (mm). (not for chamfer)	mm %
DZ	Max. depth infeed (Z direction) (not for chamfer)	mm
UXY	Finishing allowance in plane (pocket edge) (not for chamfer)	mm
UZ	Finishing allowance in depth (pocket base) (not for chamfer)	mm
Insertion:	You can select one of several insertion strategies: **Helical: Insertion along helical path** The cutter center point traverses along the helical path determined by the radius and depth per revolution. If the depth for one infeed has been reached, a full circle motion is executed to eliminate the inclined insertion path. Feedrate: Machining feedrate **Center: Insert vertically in center of pocket** The tool executes the calculated depth infeed vertically in the center of the pocket. Feedrate: Infeed rate as programmed under FZ Note: The vertical insertion into pocket center method can be used only if the tool can cut across center or if the workpiece has been predrilled.	

EP	Max. insertion gradient (only for helical insertion) The gradient of the helix may be smaller in some geometric conditions.	mm/rev
ER	Insertion radius (only for helical insertion) The radius must not be larger than the cutter radius, otherwise material will remain. Also make sure the pocket is not violated.	mm
FZ	Depth infeed rate (for insertion in center only)	mm/min mm/tooth
Remove stock	**Complete mach.:** The pocket must be milled from a solid workpiece (e.g. casting). **Remachining:** A small pocket or hole has already been machined in the workpiece. This needs to be enlarged. Parameters AZ, and ∅ must be programmed.	
FS	Chamfer width (for chamfer only), inc.	mm
ZFS	Insertion depth tool tip (for chamfer only), abs. or inc.	mm
AZ	Depth of premachined pocket or hole (for remachining only)	mm
∅1	Diameter of premachined pocket or hole (for remachining only)	mm

3.8.4 Rectangular spigot

The "Rectangular spigot" function is used when you want to mill various rectangular spigots.

You can select from the following shapes with or without a corner radius:



Rectangular spigot

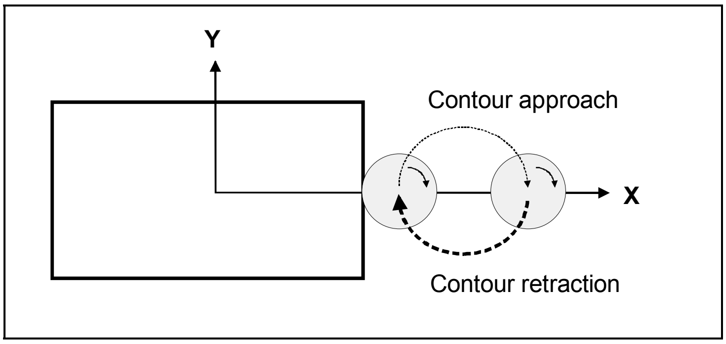
Depending on the dimensions of the rectangular spigot in the workpiece drawing, you can select a corresponding reference point for the rectangular spigot.

In addition to the required rectangular spigot, you must also define a blank spigot, i.e. the outer limits of the material. The tool moves at rapid traverse outside this area. The blank spigot must not overlap adjacent blank spigots and is placed by ShopMill automatically on the finished spigot in a centered position.

The spigot is machined using only one infeed. If you want to machine the spigot using multiple infeeds, you must program the "Rectangular spigot" function several times with a reducing finishing allowance.

Contour approach/retraction

1. The tool approaches the starting point at rapid traverse at the height of the retraction plane and adjusts to the safety clearance. The starting point is on the positive X axis rotated through $\alpha 0$.
2. The tool traverses the spigot contour sideways in a semicircle at machining feed. The tool first executes infeed at machining depth and then moves in the plane. The spigot is machined as a function of the programmed machining direction (climb/conventional) in a clockwise or counterclockwise direction.
3. When the spigot has been circumnavigated once, the tool retracts from the contour in the plane in a semicircle and then infeeds to the next machining depth.
4. The spigot is approached again in a semicircle and circumnavigated once. It repeats the process until the spigot depth has been reached.
5. The tool moves back to the safety clearance at rapid traverse.



Contour approach/retraction along semi-circle with CW rotating spindle and conventional milling operation

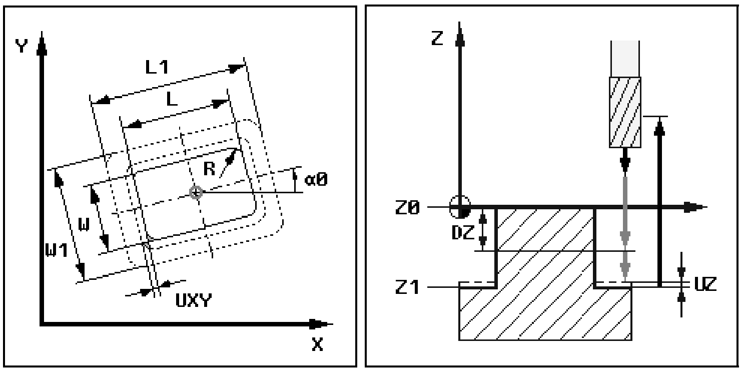
Machining type

You can select the machining mode for milling the rectangular spigot as follows:

- Roughing
  Roughing involves moving round the spigot until the programmed finishing allowance has been reached.
- Finishing
  If you have programmed a finishing allowance, the spigot is moved round until depth Z1 is reached.
- Chamfer
  Chamfering involves edge breaking at the upper edge of the rectangular spigot.

Select with softkey

Call help display with the HELP key



Help displays for milling rectangular spigots

Parameters	Description	Unit
T, F, S, V	See Sec. "Programming the tool, offset value and spindle speed".	
Reference point	You can select 5 different reference points: • Spigot center • Bottom left • Bottom right • Top left • Top right	
Machining type	Roughing Finishing Chamfer	
Single pos. Pos. pattern	A rectangular spigot is machined at the programmed position (X0, Y0, Z0). Several rectangular spigots are machined in a position pattern (e.g. full circle, pitch circle, matrix, etc.).	
 X0 Y0 Z0	The positions refer to the reference point: Position in X direction (single position only), abs. or inc. Position in Y direction (single position only), abs. or inc. Workpiece height (single position only), abs. or inc.	 mm mm mm
W	Width of spigot after machining	mm
L	Length of spigot after machining	mm
R	Radius at edges of spigot (corner radius)	mm
α0	Angle of rotation	Degr.
Z1	Depth of spigot (abs. or inc.) (not for chamfer)	mm
DZ	Max. depth infeed (Z direction) (not for chamfer)	mm
FS	Chamfer width (for chamfer only), inc.	mm
ZFS	Insertion depth tool tip (for chamfer only), abs. or inc.	mm
UXY	Finishing allowance in the plane in relation to length (L) and width (W) of the spigot; Smaller spigot dimensions are obtained by calling the cycle again and programming it with a lower finishing allowance. (not for chamfer)	mm
UZ	Finishing allowance in depth (tool axis) (not for chamfer)	mm
W1	Width of specified blank spigot (important for determining approach position)	mm
L1	Length of specified blank spigot (important for determining approach position)	mm

3.8.5 Circular spigot

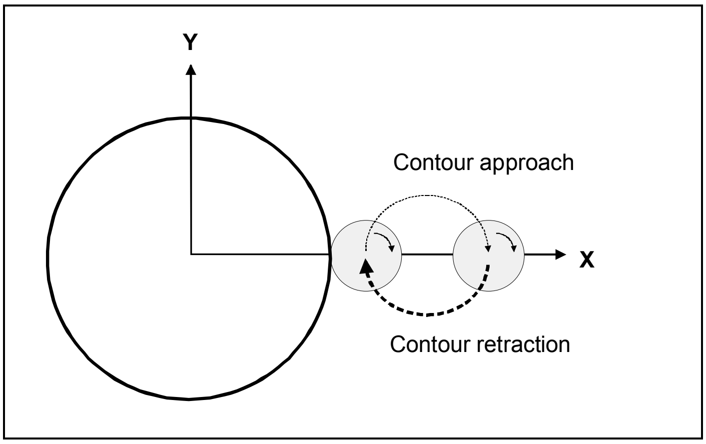
The "Circular spigot" function is used when you want to mill a circular spigot.

In addition to the required circular spigot, you must also define a blank spigot, i.e. the outer limits of the material. The tool moves at rapid traverse outside this area. The blank spigot must not overlap adjacent blank spigots and is placed by ShopMill automatically on the finished spigot in a centered position.

The spigot is machined using only one infeed. If you want to machine the spigot using multiple infeeds, you must program the "Circular spigot" function several times with a reducing finishing allowance.

Approach/retraction

1. The tool approaches the starting point at rapid traverse at the height of the retraction plane and adjusts to the safety clearance. The starting point is always on the positive X axis.
2. The tool traverses the spigot contour sideways in a semicircle at machining feed. The tool first executes infeed at machining depth and then moves in the plane. The spigot is machined as a function of the programmed machining direction (climb/conventional) in a clockwise or counterclockwise direction.
3. When the spigot has been circumnavigated once, the tool retracts from the contour in the plane in a semicircle and then infeeds to the next machining depth.
4. The spigot is approached again in a semicircle and circumnavigated once. It repeats the process until the spigot depth has been reached.
5. The tool moves back to the safety clearance at rapid traverse.



Contour approach/retraction along semi-circle with CW rotating spindle and conventional milling operation

Machining type

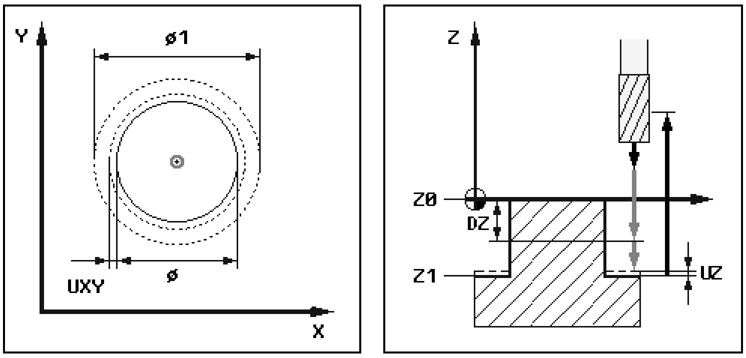
You can select the machining mode for milling the circular spigot as follows:

- Roughing
  Roughing involves moving round the spigot until the programmed finishing allowance has been reached.
- Finishing
  If you have programmed a finishing allowance, the spigot is moved round until depth Z1 is reached.
- Chamfer
  Chamfering involves edge breaking at the upper edge of the circular spigot.

Select with softkey

Spigot >
Circular spigot

Call help display with the  **key**



Help display for milling a circular spigot

Parameters	Description	Unit
T, F, S, V	See Sec. "Programming the tool, offset value, and spindle speed".	
Machining type	Roughing Finishing Chamfer	
Single pos. Pos. pattern	A circular spigot is machined at the programmed position (X0, Y0, Z0). Several circular spigots are machined in a position pattern (e.g. full circle, matrix, line).	
 X0 Y0 Z0	The positions refer to the reference point: Position in X direction (single position only), abs. or inc. Position in Y direction (single position only), abs. or inc. Workpiece height (single position only), abs. or inc.	 mm mm mm
∅	Diameter of spigot after machining	mm
Z1	Depth of spigot (abs. or inc.) (not for chamfer)	mm
FS	Chamfer width (for chamfer only), inc.	mm
ZFS	Insertion depth tool tip (for chamfer only), abs. or inc.	mm
DZ	Max. depth infeed (Z direction) (not for chamfer)	mm
UXY	Finishing allowance in plane (spigot diameter) (not for chamfer)	mm
ZU	Finishing allowance depth (spigot base) (not for chamfer)	mm
∅1	Diameter of blank spigot (important for determining approach position)	mm

3.8.6 Longitudinal slot

Use the "Longitudinal slot" function if you want to mill any kind of longitudinal slot.

The following machining methods are available:

- Mill longitudinal slot from solid material.
- Predrill longitudinal slot in the center first if, for example, the milling cutter does not cut across center (program the drilling, rectangular pocket and position program blocks one after another).

Depending on the dimensions of the longitudinal slot in the workpiece drawing, you can select a corresponding reference point for the longitudinal slot.

Approach/retraction

1. The tool moves at rapid traverse to the retraction plane and infeeds at safety clearance.
2. The tool is inserted into the material according to the chosen strategy.
3. The longitudinal slot is always machined with the chosen machining type from inside out.
4. The tool moves back to the safety clearance at rapid traverse.

Machining type

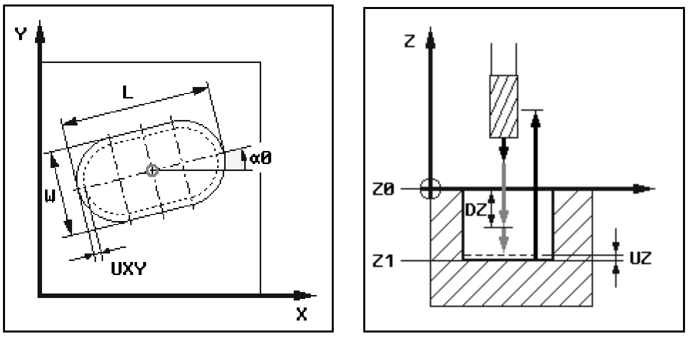
You can select the machining mode for milling the longitudinal slot as follows:

- Roughing
  During roughing, the individual planes of the slot are machined one after the other until depth Z1 is reached.
- Finishing
  In "Finishing" mode, the edge is always machined first. The slot edge is approached on the quadrant, which joins the corner radius. In the last infeed, the base is finished from the center out.
- Edge finishing
  Edge finishing is performed in the same way as finishing, except that the last infeed (finish base) is omitted.
- Chamfer
  Chamfering involves edge breaking at the upper edge of the longitudinal slot.

Select with softkey

Mill-ing | Slot > | Longitu-dinal slot

Call help display with the HELP key



Help display for a longitudinal slot

Parameters	Description	Unit
T, F, S, V	See Sec. "Programming the tool, offset value, and spindle speed".	
Reference point	The reference point position must be defined: • Center point of longitudinal slot: • Inside left • Inside right • Left-hand edge • Right-hand edge	
Machining type	Roughing Finishing Edge finishing Chamfer	
Single pos. Pos. pattern	A longitudinal slot is milled at the programmed position (X0, Y0, Z0). Several longitudinal slots are milled in a position pattern (e.g. full circle, pitch circle, matrix, etc.).	
 X0 Y0 Z0	The positions refer to the reference point: Position in X direction (single position only), abs. or inc. Position in Y direction (single position only), abs. or inc. Workpiece height (single position only), abs. or inc.	 mm mm mm
W	Slot width	mm
L	Slot length	mm
α0	Angle of rotation	Degr.
Z1	Depth of the slot (not for chamfer)	mm
DXY	Max. infeed in plane (XY direction) Alternatively, you can specify the plane infeed as a %, as a ratio → plane infeed (mm) to milling cutter diameter (mm). (not for chamfer)	mm %
DZ	Max. depth infeed (Z direction) (not for chamfer)	mm
FS	Chamfer width (for chamfer only), inc.	mm
ZFS	Insertion depth tool tip (for chamfer only), abs. or inc.	mm
UXY	Finishing allowance in plane (slot edge) (not for chamfer)	mm
ZU	Finishing allowance in depth (slot base) (not for chamfer)	mm
Insertion	It can be inserted vertically over slot center (Mi) or with oscillating motion (Pe): **Center=Insert vertically in center of longitudinal slot:** The tool is inserted to infeed depth in the pocket center. Note: This setting can be used only if the cutter can cut across center. **Oscillation=Insert with oscillation along center axis of longitudinal slot:** The cutter center point oscillates along a linear path until it reaches the depth infeed. When the depth has been reached, the path is traversed again without depth infeed in order to remove the slope caused by insertion.	mm
FZ	Depth infeed rate (for insertion in center only)	mm/min mm/tooth
EW	Insertion angle (for oscillation only)	Degr.

3.8.7 Circumferential slot

The "Circumferential slot" function is used when you want to mill one or more circumferential slots of the same size in a full circle or pitch circle.

Tool size

Please note that there is a minimum size for the milling cutter used to machine the circumferential slot:

- Roughing:
  ½ slot width W – finishing allowance UXY ≤ milling cutter diameter
- Finishing:
  ½ slot width W ≤ cutter diameter
- Edge finishing:
  Finishing allowance UXY ≤ milling cutter diameter

Annular slot

To create an annular slot, you must enter the following values for the "Number N" and "Aperture angle α_1" parameters:

N = 1

α_1 = 360°

Approach/retraction

1. The tool approaches the center point of the semicircle at the end of the slot at rapid traverse at the height of the retraction plane and adjusts to the safety clearance.
2. Then, the tool enters the workpiece at machining infeed (taking into consideration the maximum infeed in the Z direction and the finishing allowance). The circumferential slot is machined in the programmed machining direction (climb or conventional) in a clockwise or counterclockwise direction.
3. When the first circumferential slot is finished, the tool moves to the retraction plane at rapid traverse.
4. The next circumferential slot is approached along a straight line or circular path and then machined.
5. The tool moves back to the safety clearance at rapid traverse.

Machining type

You can select the machining mode for milling the circumferential slot as follows:

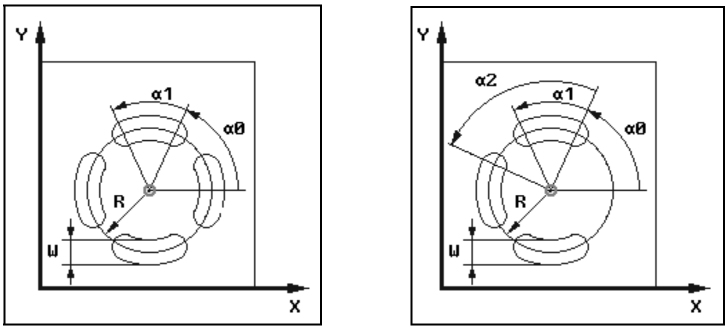
- Roughing
  During roughing, the individual planes of the slot are machined one after the other from center point of the semicircle at the end of the slot until depth Z1 is reached.
- Finishing
  In "Finishing" mode, the edge is always machined first until depth Z1 is reached. The slot edge is approached on the quadrant, which joins the radius. In the last infeed, the base is finished from the center point of the semicircle to the end of the slot.
- Edge finishing
  Edge finishing is performed in the same way as finishing, except that the last infeed (finish base) is omitted.
- Chamfer
  Chamfering involves edge breaking at the upper edge of the circumferential slot.

Select with softkey

Mill- ing | Slot > | Longitu- dinal slot

With the "Alternat." softkey, you can position the circumferential slots on a full circle or pitch circle.

Call help display with the HELP **key**

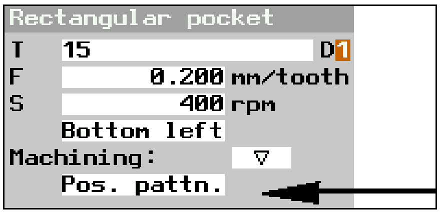


Help display for circumferential slot as full and pitch circle

Parameters	Description	Unit
T, F, S, V	See Sec. "Programming the tool, offset value and spindle speed".	
FZ	Depth infeed rate	mm/min mm/tooth
Machining type	Roughing Finishing Edge finishing Chamfer	
Full circle Pitch circle	The circumferential slots are positioned around a full circle. The slot spacing is uniform and is calculated by the control. The slots are positioned around a pitch circle. The slot spacing can be determined on the basis of angle $\alpha 2$.	
 X0 Y0 Z0	The positions refer to the center point: Position in the X direction, abs. or inc. Position in the Y direction, abs. or inc. Workpiece height, abs. or inc.	 mm mm mm
W	Slot width	mm
R	Radius of circumferential slot	mm
$\alpha 0$	Angle of rotation in relation to X axis	Degr.
$\alpha 1$	Angle of aperture of a slot	Degr.
$\alpha 2$	Advance angle (for pitch circle only)	Degr.
N	Number of slots	
Z1	Depth of the slot, relative to Z0 (not for chamfer)	mm
DZ	Max. depth infeed (Z direction) (not for chamfer)	mm
FS	Chamfer width (for chamfer only), inc.	mm
ZFS	Insertion depth tool tip (for chamfer only), abs. or inc.	mm
UXY	Finishing allowance in XY plane (edge of slot) (not for chamfer)	mm
Positioning	Linear: Next position is approached linearly at rapid traverse. Circular: Next position is approached at the programmed feedrate FP along a circular path.	
FP	Feed for positioning on a circular path	mm/min

3.8.8 Use of position patterns for milling

If you want to mill a pocket, spigot or longitudinal slot at more than one position, you must program a separate positioning block. When you call the milling cycle, use softkey "Alternat." to select "Pos. pattern" in the "Single position" parameter field. The parameters for single positions X0, Y0 and Z0 then disappear from the display.



Extract from the parameter form of a rectangular pocket with "Pos. pattern" entry

After you have finished programming the cycle and stored it, you need to program the position pattern.

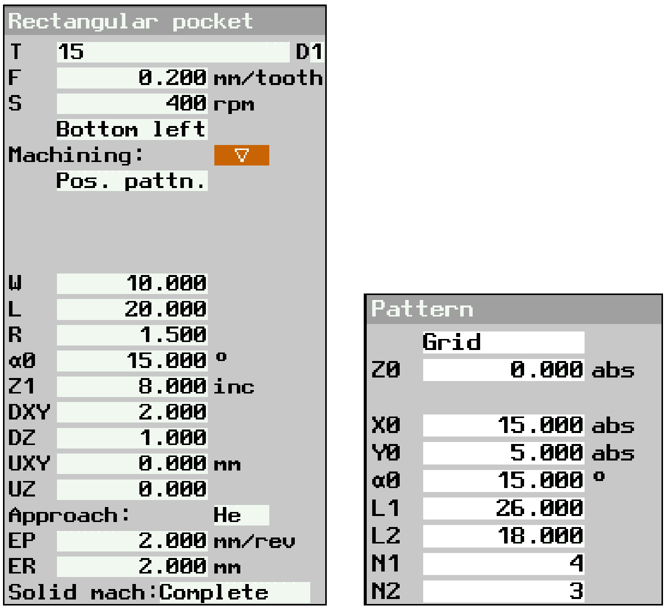
Select with softkey

Positions >

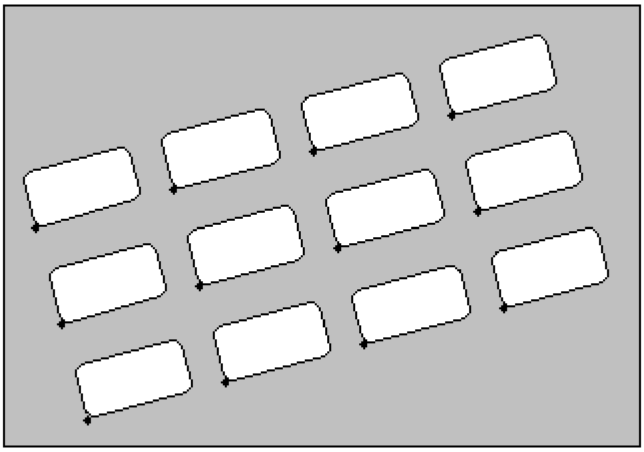
ShopMill automatically chains the milling cycle and the subsequently programmed position pattern.

Programming example 1

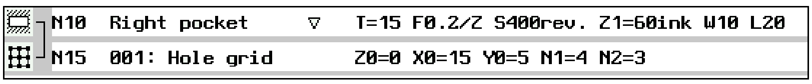
You want to mill 12 mutually parallel rectangular pockets at an angle of 15 degrees. Arrangement on matrix: 4 columns, 3 rows.
Blank dimensions: X=115mm, Y=80mm, Z=30mm
Rectangular pocket dimensions: Length 20 mm, width 10 mm, depth 8 mm
Corner radius 1.5mm.
You have selected "Bottom left" as the pocket reference point.



Parameter input fields for rectangular pocket and position pattern



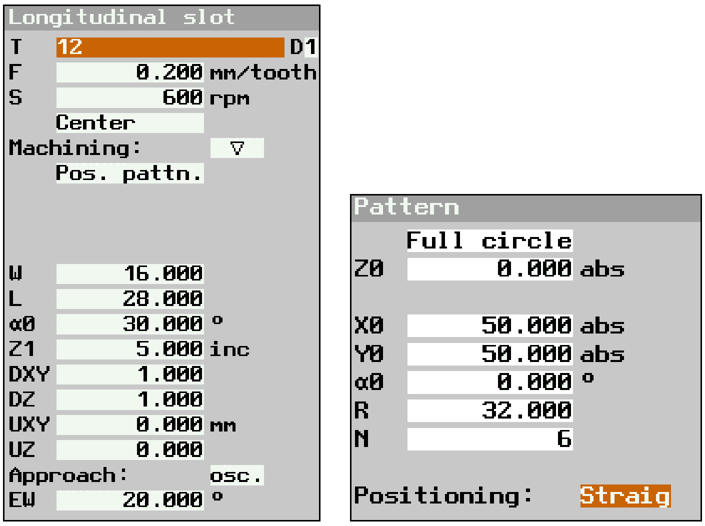
Programming graphic, rectangular pockets on matrix at angle of 15 degrees



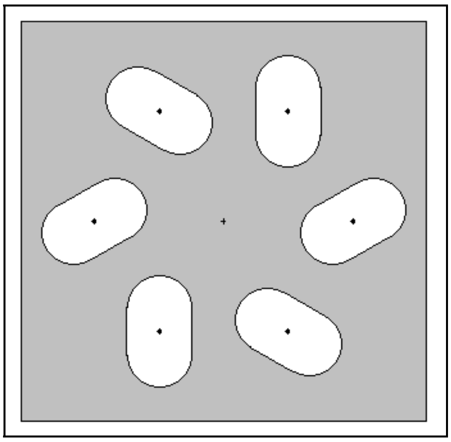
Extract from machining plan; milling rectangular pockets on a matrix

Programming example 2

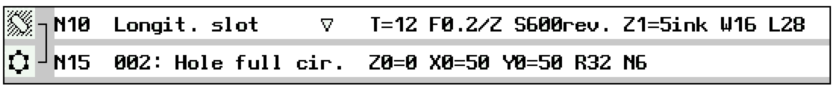
You want to rough cut 6 longitudinal slots on a full circle of Ø 32mm. The slots are rotated through 30 degrees.
Blank dimensions: X=100mm, Y=100mm, Z=20mm
Slot dimensions: Length 28mm, width 16mm, depth 5mm
You have selected "center point" as the slot reference point.



Parameter input fields for longitudinal slot and position pattern



Programming graphics, longitudinal slots at angle of 30 degrees on full circle



Extract from machining plan; milling longitudinal slots on a full circle

3.8.9 Engraving

The "Engraving" function is used to engrave a text on a workpiece along a line or arc.
You can enter the text directly in the text field as "fixed text" or assign it via a variable as "variable text".

ShopMill uses a proportional font for engraving, i.e. the width of the individual characters varies.

Approach/retraction

1. The tool approaches the starting point at rapid traverse at the height of the retraction plane and adjusts to the safety clearance.
2. The tool moves to the machining depth Z1 at the infeed feedrate FZ and mills the characters.
3. The tool retracts to the safety clearance at rapid traverse and moves along a straight line to the next character.
4. Steps 2 and 3 are repeated until the entire text has been milled.

Variable texts

There are various ways of defining variable text:

- Date and time
  You can engrave workpieces with the production date and current time of day, for example. The values for data and tiem are read from the CNC.
- Workpiece count
  Using the workpiece variables you can assign a consecutive number to the workpieces. You can define the format (number of digits, leading zeroes). The "workpiece count" variable is defined as a user variable (_E_PART[0]) in the GUD 7 data block.
  The place holder (#) is used to format the number of digits at which the workpiece counts output will begin.

  If you output the workpiece count 1 for the first workpiece, you can specify an additive value (e.g., <#,_E_PART[0] + 100>). The workpiece count output is then incremented by this value (e.g. 101, 102, 103,...).
- Numbers
  When outputting number (e.g. measurement results), you can select the output format (digits either side of the point) of the number to be engraved.
- Text
  Instead of entering a fixed text in the engraving text field, you can specify the text to be engraved via a text variable (e.g., _VAR_TEXT=“ABC123“)

Mirror writing

If you want to engrave mirror writing, first program the mirroring (see Section "Defining coordinate transformations" and then enter the text in the "Engraving" function.

Full circle

If you want to distribute the characters evenly around a full circle, enter the arc angle α2=360°. ShopMill then distributes the characters evenly around the full circle.

Mill-ing | Engraving >

- Press the "Milling" and "Engraving" softkeys.

Lowercase letters

Lowercase

- Press the "Lowercase" softkey to enter lowercase letters.
  Press it again to enter uppercase letters.

Special characters

Special characters>

- Press the "Special characters" softkey if you need a character that does not appear on the input keys.

The "Special characters" window appears.

- Place the cursor on the character you require.

OK

- Press the "OK" softkey.

The selected character is inserted into the text at the cursor position.

Entering a date

Variable > | Date

- Press the "Variable" and "Date" softkeys if you want to engrave the current date.

The data is inserted in the European date format (<DD>.<MM>.<YYYY>).
To obtain a different date format, you must adapt the format specified in the text field. For example, to engrave the date in the American date format (month/day/year => 8/16/04), change the format to <M>/<D>/<YY> .

Entering a time

Variable > | Time

➢ Press the "Variable" and "Time" softkeys if you want to engrave the current time.

The time is inserted in the European format (<TIME24>).
To have the time in the American format, change the format to <TIME12>.

Example:

Text entry:	Time: <TIME24>	Execution:	Time: 16.35
	Time: <TIME12>		Time: 04.35 PM

Entering workpiece counts

Variable > | Workpiece count 000123

➢ Press the "Variable" and "Workpiece count 000123" softkeys to engrave a workpiece count with a fixed number of digits and leading zeroes.

The format text <######,_E_PART[0]> is inserted and you return to the engraving field with the softkey bar.

➢ Define the number of digits by adjusting the number of place holders (#) in the engraving field.

If the specified number of digits (e.g. ##) is not enough to display the workpiece count, ShopMill will increase the number digits automatically.

- OR -

Variable > | Workpiece count 123

➢ Press the "Variable" and "Workpiece count 123" softkeys if you want to engrave a workpiece count without lead zeroes.

The format text <#,_E_PART[0]> is inserted and you return to the engraving field with the softkey bar.

➢ Define the number of digits by adjusting the number of place holders in the engraving field.

If the specified number of digits is not enough to display the workpiece count (e.g. 123), ShopMill will increase the number digits automatically.
You can enter an additive value, for example, if you want to resume production of workpieces after an interruption. The workpiece count output is then increased by this value.

Entering a variable number

Variable > | Number 123.456

➢ Press the "Variable" and "Number 123.456" softkeys if you want to engrave a any number in a certain format.

The format text <#.###,_VAR_NUM> is inserted and you return to the engraving field with the softkey bar.

➢ The place holders #.### define the digit format in which the number defined in _VAR_NUM will be engraved.

For example, if you have stored 12.35 in _VAR_NUM, you can format the variable as follows.

Input	Output	Meaning
<#,_VAR_NUM>	12	Integer digits not formatted, no fractional digits
<####,_VAR_NUM>	0012	4 integer digits, leading zeroes, no fractional digits
< #,_VAR_NUM>	12	4 integer digits, leading zeroes, no fractional digits
<#.,_VAR_NUM>	12.35	Integer and fractional digits not formatted.
<#.#,_VAR_NUM>	12.4	Integer digits not formatted, 1 fractional digit (rounded)
<#.##,_VAR_NUM>	12.35	Integer digits not formatted, 2 fractional digits (rounded)
<#.####,_VAR_NUM>	12.3500	Integer digits not formatted, 4 fractional digits (rounded)

If there is insufficient space in front of the decimal point to display the number entered, it is automatically extended. If the specified number of digits is larger than the number to be engraved, the output format is automatically filled with the appropriate number of leading and trailing zeroes.

Instead of the decimal point you can also use a blank.
Instead of _VAR_NUM you can use any other numeric variable (e.g. R0).

Entering a variable text

Variable > | Variable text

➢ Press the "Variable" and "Variable text" softkeys if you want to take the text to be engraved (up to 200 characters) from a variable.

The format text <Text, _VAR_TEXT> is inserted and you return to the engraving field with the softkey bar.
You can use any other text variable instead of _VAR_TEXT.

Deleting text

Delete text >

➢ Press the "Delete text" softkey to delete the entire text.

The format text for the variables is always inserted at the current cursor position.

The "Lowercase", "Special characters", "Variable", and "Delete text" softkeys only appear when you place the cursor in the input field for engraving text.

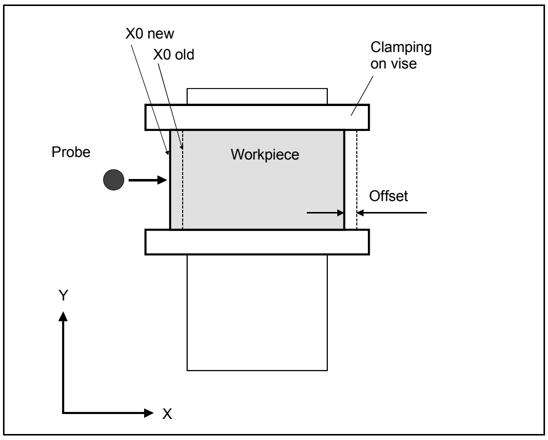
Parameters	Description	Unit
T, D, F, S, V	See Sec. "Creating program blocks".	
Alignment	Align text to line Align text to arc Align text to arc	
Reference point	Position of reference point within text	
Engraving text	a maximum of 91 characters	
X0	Reference point in X direction (abs.)	mm
R	Ref. point on longitudinal polar axis (alternative to X0) – (for curved alignment only)	mm
Y0	Reference point in Y direction (abs.)	mm
α0	Ref. point on angular polar axis (alternative to Y0) – (for curved alignment only)	Degr.
Z0	Reference point in Z direction (abs.)	mm
Z1	Machining depth (abs. or inc.)	mm
FZ	Depth infeed rate	mm/min mm/tooth
W	Character height	mm
DX1	Character spacing	mm
DX2	Total width (alternative to DX1) – (for linear alignment only)	mm
α1	Text direction (for linear alignment only)	Degr.
α2	Arc angle (alternative to DX1) – (for curved alignment only)	Degr.
XM	Center point of arc (abs.) – (for curved alignment only)	mm
YM	Center point of arc (abs.) – (for curved alignment only)	mm

3.9 Measurement

3.9.1 Measuring the workpiece zero

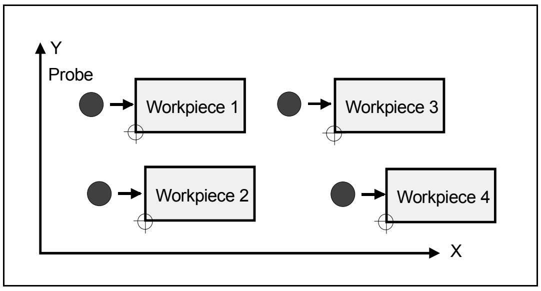
The "Workpiece zero" function is used to determine the workpiece zero in a program by means of an electronic measuring probe.

For example, if you want to produce several workpieces, an offset may arise between the old and the new workpiece zero when clamping the next workpiece in the vise. You can measure the workpiece edges to determine the new workpiece zero and save it in a work offset or in a GUD.



Workpiece clamping with offset in relation to previous clamping

Even if you want to machine several clamped workpieces in parallel, you can determine the zero for each workpiece first.



Several clamped workpieces

You can use electronic measuring probes exclusively to determine the workpiece zero within a program. These must be calibrated beforehand (see Sec. "Calibrating an electronic measuring tool"). These measuring probes must always be specified as 3D probe in the tool management.

During automatic measurement, the measuring probe first moves at rapid traverse and then at measuring infeed to the edge of the workpiece and back. The measuring infeed is defined in a machine data.

Please refer to the machine manufacturer's instructions.

The workpiece radius is considered during calculation of the workpiece zero and is stored in a work offset.

Mics. | Workpiece zero >

X ... Z

Accept

- Insert an electronic measuring probe in the spindle (see Sec. "Programming the tool, offset value, and spindle speed").
- Select the "Misc." and "Workpiece zero" softkeys.
- Use the softkeys to select in which axis direction you want to approach the workpiece first.
- Enter the values for the individual parameters.
- Press the "Accept" softkey.
- Repeat the process for the other two axes.

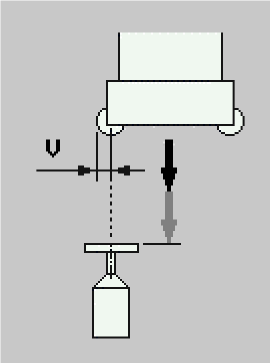
Parameters	Description	Unit
T	Tool of type 3D probe	
X Y Z	Approach position in X direction (abs.) Approach position in Y direction (abs.) Approach position in Z direction (abs.)	mm mm mm
Zero off.	Work offset where the workpiece zero is to be saved. • Basic work offset • Work offset (the values are saved in the coarse offset and existing values in the fine offset are deleted.) • GUD data (you to scan the measurement result in GUD E_MEAS, e.g. for other calculations (tolerance checks, etc.)).	
Approach direction	+: The probe approaches the workpiece in the plus direction –: The probe approaches the workpiece in the minus direction At the approach position in the Z direction, this parameter does not apply as the tool can only approach the workpiece in a negative direction!	
X0, Y0, Z0	Setpoint position of the workpiece edge	mm

3.9.2 Measuring the tool

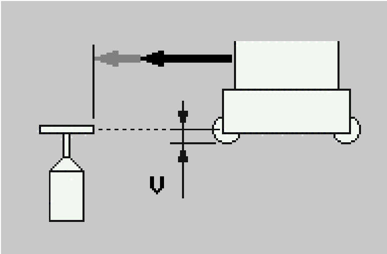
Use the "Measure tool" function if you want to check the tool wear while machining the workpiece.

You can only measure the tools within a program using an electronic measuring probe, which you must calibrate the probe first.

You can consider a lateral or longitudinal offset V when measuring. If the maximum length of the tool is not at the outer edge of the tool or the maximum width is not at bottom edge of the tool, you can store this difference in the offset.



Longitudinal offset



Lateral offset

ShopMill then calculates the tool wear data from the known positions of the toolholder reference point and of the probe and from the tool offset data. The wear values are automatically entered in the wear list and added to any existing values stored there.

If the wear exceeds the maximum permissible value for tool wear ΔL or ΔR, the tool is replaced and disabled against further use. If no replacement tool is available, the machining operation is interrupted.

During the measuring cycle the tool automatically approaches the probe at measurement feedrate. The tool then moves to the retraction plane before returning to the tool change point.
ShopMill automatically executes the measurement with either a rotating or stationary spindle depending on the tool type and selected measurement method (measure radius/length).

Th radius is always measured in the opposite direction to spindle rotation.

The length of a tool is measured with the spindle stationary. However, if the diameter of the milling cutter to be measured is gerater than the diameter of the probe, the rotating spindle is measured in the opposite direction. The tool is then not move over the probe center to center but with the outside edge of the tool above the center of the probe.

➢ Move the tool until it is positioned over the approximate center of the measuring surface of the probe (see Section "Straight or circular path motion").

Mics. | Measure tool >

➢ Press the "Misc." and "Measure tool" softkeys.

Measure length -or- Measure radius

➢ Use the softkeys to select whether you want to measure the radius or the length of the tools.

Parameters	Description	Unit
T	Tool to be measured	
D	Cutting edge of the tool	
V	Lateral offset (if necessary) – for length measurement only Longitudinal offset (if necessary) – for radius measurement only	mm mm
∆L	Max. permissible wear value (see tool data sheet supplied by tool manufacturer) – applies only to length measurement	mm
∆R	Max. permissible wear value (see tool data sheet supplied by tool manufacturer) – applies only to radius measurement	mm

3.9.3 Calibrating the measuring calipers

If you want to use a measuring probe to measure your tools, you must first determine the position of the probe on the machine table relative to the machine zero.
You can determine this position either within a program (see below) or during preparation (see Section "Operation" → "Calibrating the measuring probe").

You must use a mill-type calibration tool to calibrate the measuring probe. You must enter the length and radius/diameter of the tool in the tool list beforehand.

Calibration is automatically executed at the measuring feedrate.
The distance measurements between the machine zero and measuring probe are calculated and stored in an internal data area.

- Insert the calibration tool (see Section "Programming the tool, offset value, and spindle speed").
- Move the calibration tool until it is positioned over the approximate center of the measuring surface of the measuring probe (see Section "Straight or circular path motion").

Mics. | Measure tool >

- Press the "Misc." and "Measure tool" softkeys.

Calibrate probe

- Press "Calibrate probe" softkey.
- Choose whether you want to calibrate the length or the length and diameter of the probe.

3.10 Miscellaneous functions

3.10.1 Calling a subroutine

If you require the same machining steps in the programming of different workpieces, you can define these machining steps in a separate routine. You can then call this subroutine in any programs. Identical machining steps therefore only have to be programmed once.

ShopMill does not differentiate between main program and subroutine. This means that you can call a "standard" sequential control or G code program as subroutine in another sequential control program.
In this subroutine, you can also call another subroutine. The maximum nesting depth is 8 subroutines.
You cannot not insert subroutines among blocks chained by the control.

If you want to call a sequential control program as a subroutine, the program must already have been calculated once (load or simulate program in Machine Auto mode). This is not necessary for G code subroutines.

The subroutine must always be stored in the NC main memory (in a separate directory "XYZ" or in the "ShopMill", "Part programs", "Subroutines" directories).

If you want to call a subroutine located on another drive, you can use G code command "EXTCALL".

Example: Calling program "Form25_1.mpf" on the compact flash card of the PCU 20:

```
EXTCALL "C:\FORM25_1.MPF"
```

Please note that, when a subroutine is called, ShopMill evaluates the settings in the program header of the subroutine. These settings also remain active even after the subroutine has ended.
If you wish to activate the settings from the program header for the main program again, you can make the settings again in the main program after calling the subroutine (see Sec. "Changing program settings").

- How to create a ShopMill or G code program that you can call as a subroutine in another program.
- Place the cursor in the machining plan of the main program on the program block after which the subroutine call is to be inserted.

Sub-
routine

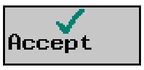
- Press the "Misc." and "Subroutine" softkeys.
- Enter the path of the subroutine if the desired subroutine is not stored in the same directory as the main program.

Directory	**Path to enter**
ShopMill	ShopMill
Separate directory XYZ	XYZ
Part programs	MPF
Subprograms	SPF

- Enter the name of the subroutine that you want to insert.
  You only need to enter the file extension (*.mpf or *.spf) if the subroutine does not have the file extension specified for the directory in which the subroutine is stored.

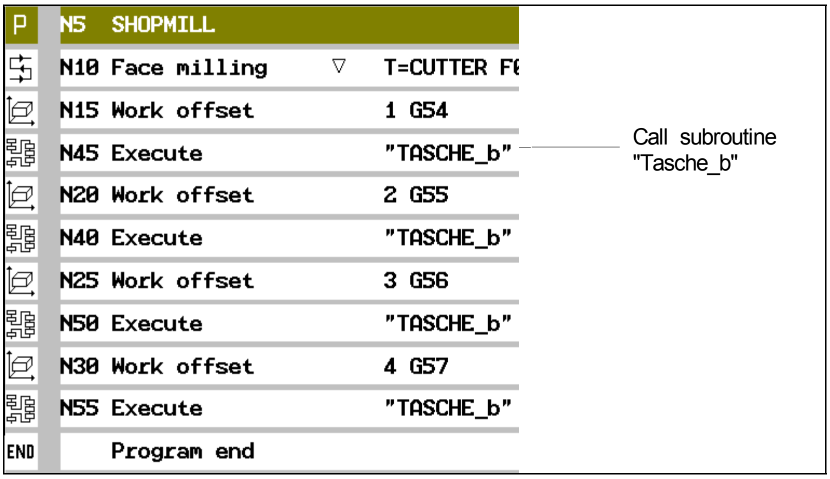
Directory	**Specified file extension**
ShopMill	*.mpf
Separate directory XYZ	*.mpf
Part programs	*.mpf
Subprograms	*.spf

The subroutine is thus executed in the position pattern.



- Press the "Accept" softkey.

The subroutine call is inserted in the main program.



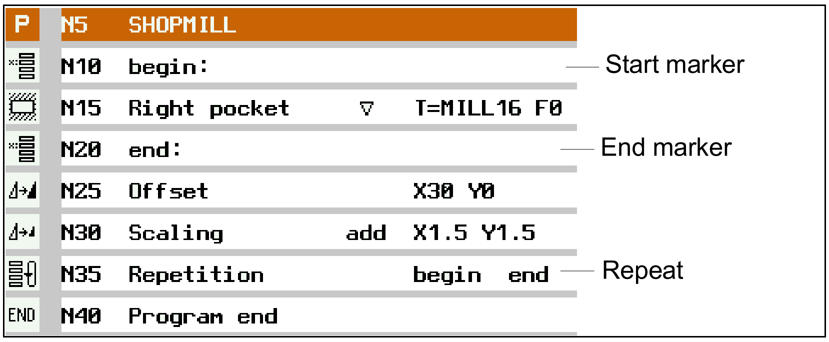
Subroutine call

3.10.2 Repeating program blocks

If certain steps in the machining of a workpiece have to be executed more than once, it is only necessary to program these steps once. ShopMill offers a function for repeating program blocks.

You must mark the program blocks that you want to repeat with a start and end marker. You can then call these program blocks up to 9999 times again within a program. The markers must be unique, i.e. they must have different names.
You can also set markers and repeats after creating the program, but not within chained program blocks.

It is also possible to use the same marker as the end marker of the preceding program blocks and as the start marker for the following program blocks.



Repeating program blocks

Mics. | Set marker

➢ Press the "Misc." and "Set marker" softkeys.

➢ Enter a name.

Accept

➢ Press the "Accept" softkey.

A start marker is inserted behind the current block.

➢ Enter the program blocks that you want to repeat later.

Mics. | Set marker

➢ Press the "Misc." and "Set marker" softkeys.

➢ Enter a name.

Accept

➢ Press the "Accept" softkey.

An end marker is inserted behind the current block.

➢ Continue programming up to the point where you want to repeat the program blocks.

Mics. | Repeat

➢ Press the "Misc." and "Repeat" softkeys.

➢ Enter the names of the start and end markers and the number of times the blocks are to be repeated.

Accept

➢ Press the "Accept" softkey.

The marked program blocks are repeated.

3.10.3 Changing program settings

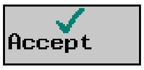
All parameters defined in the program header, with the exception of the unit of measurement, can be changed at any location in the program.

The settings in the program header are modal, i.e. they remain active until they are changed.

Define a new blank, e.g. in the sequential control program, if you want to change the visible cutout during simulation.

This is useful for the work offset, coordinate transformation, cylinder peripheral surface transformation and swiveling functions. First program the functions listed above and then define a new blank.

- Press the "Misc." and "Settings" softkeys.
- Enter the desired parameters.
  For a description of the parameters, see Sec. "Creating a new program".

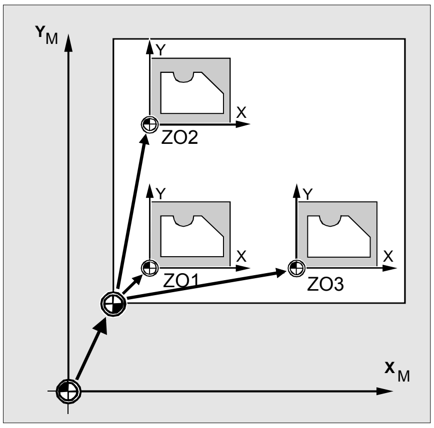


- Press the "Accept" softkey.

The new settings for the program are loaded.

3.10.4 Calling work offsets

You can call work offsets (G54, etc.) from any program.
You can use these offsets, for example, when you want to machine workpieces with various blank dimensions using the same program. The offset will, in this case, adapt the workpiece zero to the new blank.



Work offset in X and Y directions

You define the work offsets in the work offset list (see Sec. "Defining work offsets"). You can also view the coordinates of the selected offset here.

Mics. | Transfor-mation > | Work offset >

- Press the "Misc.", "Transformation", and "Work offset" softkeys.

- Select one of the work offsets or the standard offset.

-or-

- Enter the desired offset directly in the input field.

-or-

Work offset >

- Press the "Work offset" softkey.

The work offset list is displayed.

-and-

- Select a work offset.

To Program

-and-

➢ Press the "To Program" softkey.

The work offset is loaded into the parameterization screen form.

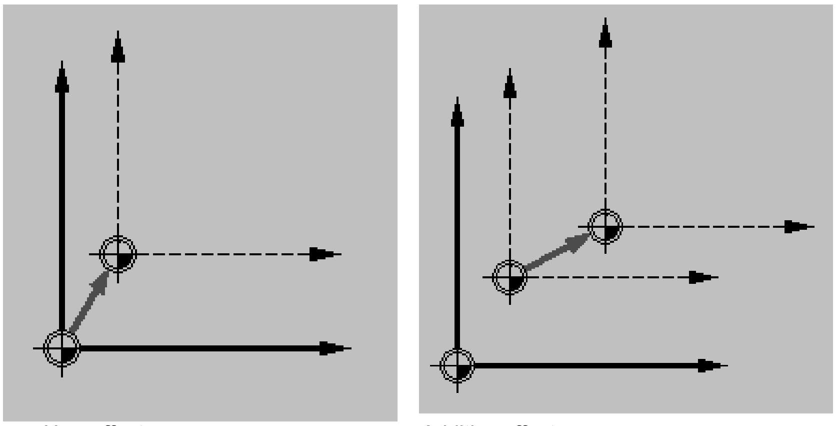
To deselect the work offsets, select the standard offset or enter zero in the field.

3.10.5 Defining coordinate transformations

To make programming easier, you can transform the coordinate system. Use this possibility, for example, to rotate the coordinate system.

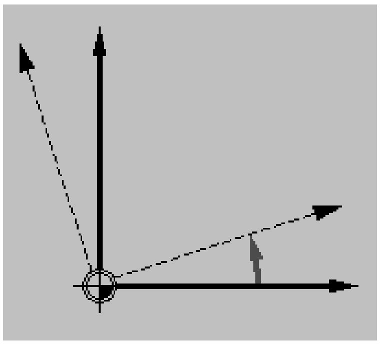
Coordinate transformations only apply in the current program.
You can define displacement, rotation, scaling or mirroring. You can select between a new or an additive coordinate transformation.
In the case of a new coordinate transformation, all previously defined coordinate transformations are deselected. An additive coordinate transformation acts in addition to the currently selected coordinate transformations.

- Offset
  For each axis, you can program an offset of the zero point.

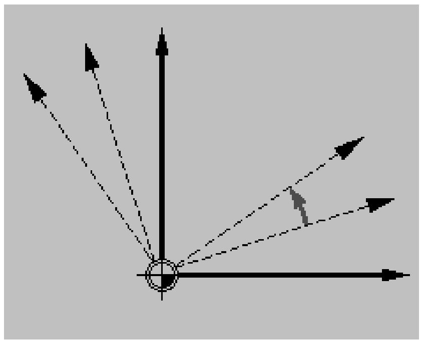


New offset *Additive offset*

- Rotation
  You can rotate every axis through a specific angle. A positive angle corresponds to counterclockwise rotation.



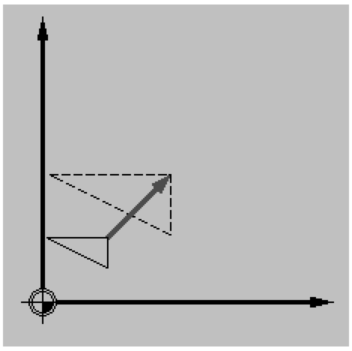
New rotation



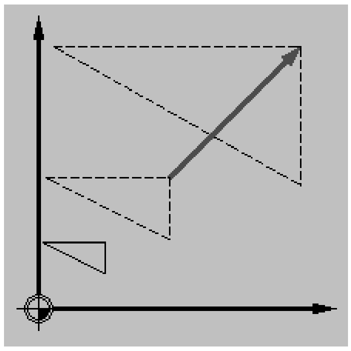
Additive rotation

- Scaling
  You can specify a scale factor for the active machining plane as well as for the tool axis. The programmed coordinates are then multiplied by this factor.

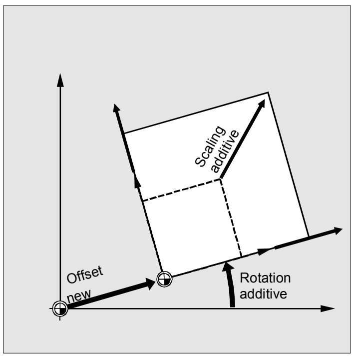
Note that the scaling always refers to the zero point of the workpiece. For example, if you increase the size of a pocket whose center point does not coincide with the zero point, scaling will shift the center of the pocket.



New scaling



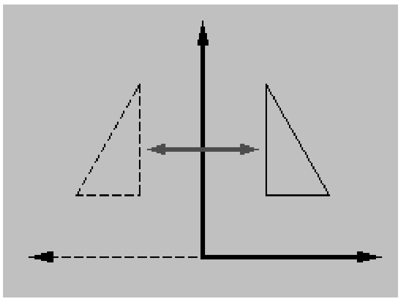
Additive scaling



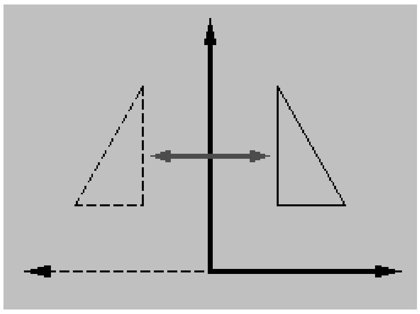
Offset, rotation, and scaling

- Mirroring
  Furthermore, you can mirror all axes.
  Enter the axis to be mirrored in each case.

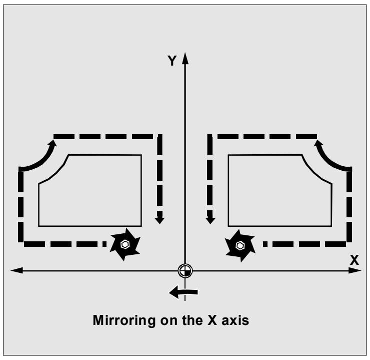
Note that with mirroring, the travel direction of the cutting tool (conventional/climb) is also mirrored.



New mirroring



Additive mirroring



Mirroring of the X axis

Mics. | Transfor-mation >

Offset > ... Mirroring >

- Press the "Misc." and "Transformation" softkeys.
- Select the coordinate transformation using the softkey.
- Select whether you want to program a new or an additive coordinate transformation.
- Enter the desired coordinates.

3.10.6 Cylinder surface transformation

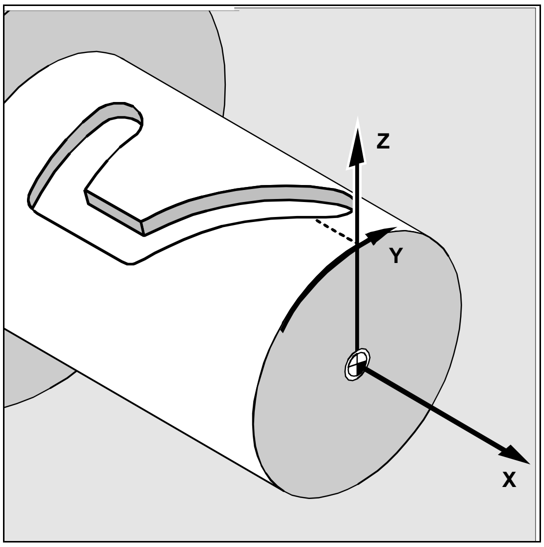
Function

The cylinder surface transformation is required in order to

- Longitudinal grooves on cylindrical bodies,
- Transverse grooves on cylindrical objects,
- Grooves with any path on cylindrical bodies.

The cylinder peripheral surface transformation is a software option.

The path of the grooves is programmed with reference to the unwrapped, level surface of the cylinder. The slot can be programmed as a line/circle contour, via drilling or milling cycles or with the contour milling function (free contour programming).



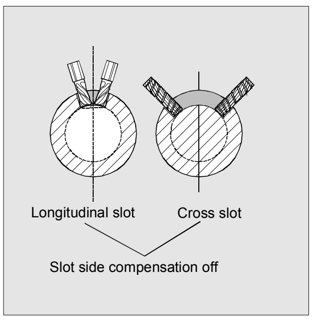
There are two variants of cylinder surface transformation, i.e.

- Slot side compensation OFF
- Slot side compensation on (path milling only)

Slot side compensation OFF

When slot side compensation is deactivated, any type of slot with parallel sides can be machined if the tool diameter equals the slot width.

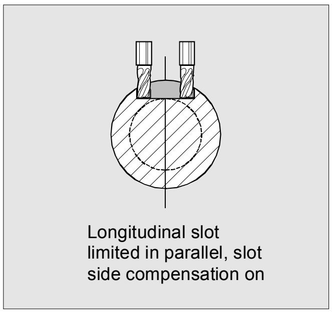
The slot sides are not parallel if the slot width is larger than the tool diameter.



The slot contour is programmed for machining purposes.

Slot side compensation ON

When slot side compensation is active, slots with parallel sides are machined even if the slot width is larger than the tool diameter.



The slot contour must not be programmed for machining purposes, but the imaginary center-point path of a bolt inserted in the slot; the bolt must be in contact with all sides of the slot. The slot width is determined by parameter D. (see also Sec. "Example 5: Slot side compensation".)

Programming

The basic programming procedure is as follows:

1. Select work offset for cylinder surface transformation (e.g. offset the zero point on the center point of the cylinder end face)
2. Position the Y axis (Y axis must be positioned prior to cylinder surface transformation because it is defined differently after transformation)
3. Activate cylinder surface transformation
4. Select work offset for machining on developed cylinder surface (e.g. shift zero point to the zero point on the workpiece drawing)
5. Program machining operation (e.g. enter contour and path milling)
6. Deactivate cylinder surface transformation

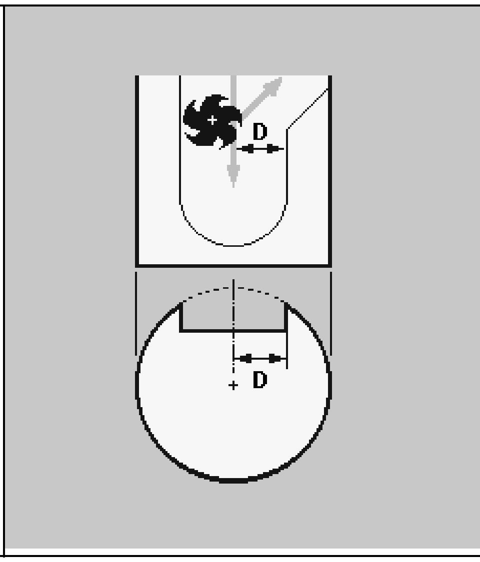
The programmed cylinder surface transformation is simulated only as a developed peripheral surface.

The work offsets active prior to selection of cylinder surface transformation are no longer active after the function has been deselected.

Select with softkey

Transfor-
mations >

Cylinder
surface >

Parameters	Description	Unit
Transformation	Activate/deactivate cylinder peripheral surface transformation (see also example below)	
∅	Cylinder diameter (only when transformation is active)	mm
Slot wall offset	Activate/deactivate slot side compensation (only when transformation is active)	
D	Offset to the programmed path (only with groove wall offset ON) 	mm

Options for free contour programming

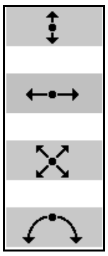
General information

For contours (e.g. slots) on a cylinder, lengths in the circumferential direction of the cylinder peripheral surface (e.g. Y axis) are often specified as angles.
Several options are available under the "Mill contour" function in free contour programming for this purpose.
Depending on the selected axis (selection is made via a display machine data), you can enter the length as an angle.

Start point

In the screen form for selecting the starting point, you can also activate or deactivate the cylinder surface transformation function via the "Alternat." softkey. When the function is active, you are offered the diameter ∅ of the cylinder.

Contour elements



Depending on the axis and the relevant element, angle parameters α, Iα or Yα, Jα are added to the "Horizontal/vertical/diagonal line" and "Arc" when the cylinder surface transformation function is active.

Notes
The dimensions of the developed surface are specified in mm in the graphics!

3.10.7 Swiveling

You can use swivel heads or swivel tables to create and/or machine inclined planes.

It is not necessary to program the swivel axes of the machine (A, B, C), since you can specify the rotations around the geometry axes (X, Y, Z) of the workpiece coordinate system directly as described in the relevant workpiece drawing.
The rotation of the workpiece coordinate system in the program is then automatically converted to a rotation for the relevant swivel axis of the machine during machining.

The swivel axes are always rotated to place the machining plane perpendicular to the tool axis for machining. During machining, the machining plane is permanently set.

When the coordinate system is swiveled, the previously set work offset is automatically converted for the swiveled state.

The main programming procedure is:

1. Swivel the coordinate system into the plane to be machined.
2. Program machining in the X/Y plane in the usual way.
3. Swivel the coordinate system back to its original position.

It is possible the software limit switches may be violated in the swiveled plane when approaching the programmed machining. In this case, ShopMill travels along the software limit switches above the retraction plane. In the event of violation below the retraction plane, the program is interrupted with an alarm for safety's sake. To avoid this, before swiveling, e.g. move the tool in the X/Y plane and position it as close as possible to the starting point of the machining operation or define the retraction plane closer to the workpiece.

In a swiveled plane the "Workpiece zero" function is operative but not the "Measure tool" function.
The swiveled coordinates are retained in reset state and even after power ON, i.e. you can still retract from an inclined hole by retracting in +Z direction.

Please refer to the machine manufacturer's instructions.

The following provides an explanation of the most important parameters for swiveling:

Retraction

Before swiveling the axes you can move the tool to a safe retraction position. The retraction methods available to you are defined in the "Retraction position" parameter during set-up of the swivel data set. The retraction mode is modal. On a tool change or after a block search, the retraction mode last set is used.

Please refer to the machine manufacturer's instructions.

Warning

You must select a retraction position that precludes collision between the tool and workpiece during swiveling.

Swiveling

Select whether to rotate the coordinate system, for example, to execute a further swivel in the new coordinate system or whether to really traverse the swivel axes. If you want to perform a machining operation in the swiveled plane, you will need to be able to move the swivel axes.

Swivel method

The coordinate system can be swiveled either axially or via solid or projection angles. The machine manufacturer determines when setting up the "Swivel" function which swivel methods are available.

Please refer to the machine manufacturer's instructions.

- In the case of axial swiveling, the coordinate system is rotated about each axis in turn, with each rotation starting from the previous rotation. The axis sequence can be selected freely.
- With the swiveling variant based on solid angles, the coordinate system is rotated first about the Z axis and then in negative direction about the Y axis. The second rotation starts from the first.
- When swiveling via projection angle, rotation is carried out about two axes simultaneously, i.e. you can view two angles at the same time. The third rotation is based on the first two.
  You can select any of the axes.
  This method is used, for example, for oblique drilling with angles dimensioned in the side elevations of the part drawing. The side elevations then correspond to the unrotated coordinate system.

The positive direction of each rotation for the different swivel methods is shown in the help displays.

Direction

In swivel systems with 2 rotary axes, a particular plane can be reached in two different ways. You can choose between these two different positions in the "Direction" parameter. The +/- corresponds to the larger or smaller value of a rotary axis. This may affect the working area.
When the swivel data set is set up, the entries in the "Direction" parameter determine for which rotary axis you can select each of the two settings.

Please refer to the machine manufacturer's instructions.

If one of the two positions cannot be reached for mechanical reasons, the alternative position is automatically selected irrespective of the setting of the "Direction" parameter.

Fixing the tool tip

To avoid collisions, you can use the 5-axis transformation (software option) to retain the position of the tool tip during swiveling. This function must be enabled in the "Follow-up tool" parameter when the "Swivel" function is set up.

Please refer to the machine manufacturer's instructions.

Mics. | Transfor-mation > | Swivel >

➢ Press the "Misc." and "Transformation" softkeys.

Delete values

➢ Press the "Delete values" softkey to restore the initial state, i.e. to zero the values.
This is done, for example, to swivel the coordinate system back to its original orientation.

Parameters	Description	Unit
TC	Name of the swivel data block 0: Removing the swivel head, deselecting the swivel data set No entry: No change to set swivel data block	
T	Tool name	
Move clear	No: Do not retract tool before swiveling Z: Move tool axis to retraction position before swiveling Z, X, Y: Move machining axis to retraction position before swiveling Tool max: Retract tool as far as the software limit switch in the tool direction Tool inc: Retract tool up to the incremental value entered in the tool direction	

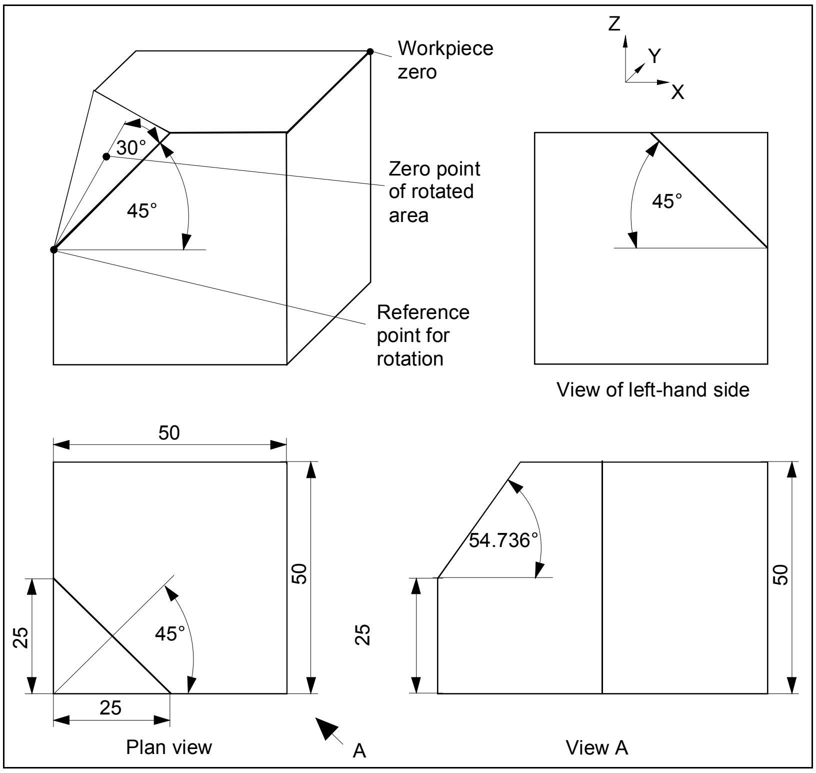
Swiveling	Yes: compute and swivel (swivel coordinate system and move swivel axes) No: only compute, don't swivel (only swivel coordinate system, don't move swivel axes)		
Transformation	Swiveling additive or new		
X0	Reference point for rotation		mm
Y0	Reference point for rotation		mm
Z0	Reference point for rotation		mm
Swivel method	Axial swiveling, or swiveling via solid or projection angle		
X	Axis angle (axial swivel)	The sequence of the axes can be altered as required with "Alternat."	Degr.
Y	Axis angle (axial swivel)		Degr.
Z	Axis angle (axial swivel)		Degr.
α	Angle of rotation in the XY plane about the Z axis (swiveling via solid angle)		Degr.
β	Angle of rotation in space about the Y axis (swiveling via solid angle)		Degr.
$X\alpha$	Axis angle (swiveling via projection angle)	The sequence of the axes can be altered as required with "Alternat."	Degr.
$Y\alpha$	Axis angle (swiveling via projection angle)		Degr.
$Z\beta$	Axis angle (swiveling via projection angle)		Degr.
X1	New zero point of rotated surface		mm
Y1	New zero point of rotated surface		mm
Z1	New zero point of rotated surface		mm
Direction	Preferred direction of rotation with 2 alternatives +: Larger angle of the axis on the scale of the swivel head / swivel table -: Smaller angle of the axis on the scale of the swivel head / swivel table		
Fix tool tip	Follow-up: The position of the tool tip is maintained during swiveling. Do not correct: The position of the tool tip is changes during swiveling.		

Other additive transformations can be added to the offsets before (X0, Y0, Z0) or after (X1, Y1, Z1) swiveling (see Sec. "Work offsets").

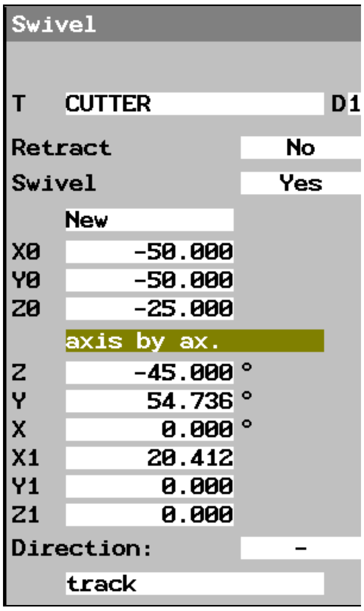
Programming example

You want to bevel a corner on a cube. The oblique surface is defined as the machining plane as follows:

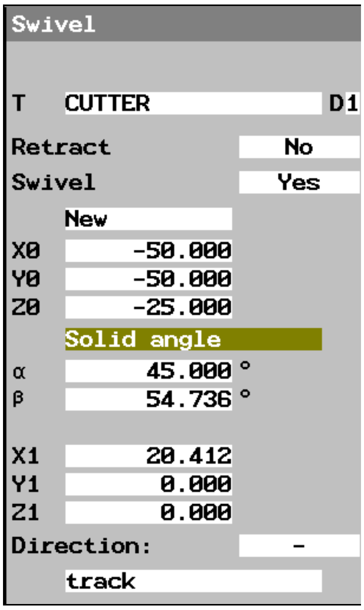
- With axial swiveling and swiveling using solid angles, the system of coordinates is rotated first in the XY plane in such a way that the upper edge of the inclined surface of the cube runs parallel to the X axis (rotate 45° about Z axis or α=45°). The system of coordinates is then tilted so that the inclined plane of the cube is in the XY plane (rotate -54.736° about Y axis -54.736° or β=54.736°).
- With the swiveling via projection angles options, the X and Y axes are rotated through 45° so that the inclined plane of the cube is in the XY plane. The Z axis is then rotated through 30° so that the X axis runs through the center point of the inclined surface (zero point of rotated surface).



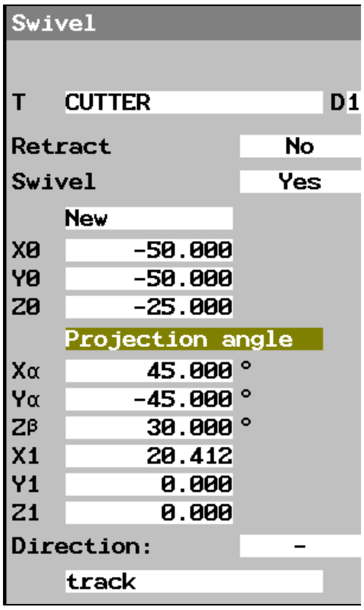
Workpiece machined by a swivel head



Swivel (axial)



Swivel (solid angle)



Swivel (projection angle)

3.10.8 Miscellaneous functions

You can e.g. position the spindle again between the individual machining steps or activate the coolant or stop machining.

The following functions are available:

- Spindle
  Determine direction of spindle rotation or spindle position (see Sec. "Starting, stopping, and positioning a spindle manually")
- Gear stage
  Set gear stage, if machine has gears

Please refer to the machine manufacturer's instructions.

- Miscellaneous M functions
  Machine functions, such as "Close door"; they are additionally provided by the machine manufacturer

Please refer to the machine manufacturer's instructions.

- Coolant
  Activate/deactivate coolant 1/2
- Tool-spec. fct 1 to 4
  Select tool-specific functions 1 to 4; they are additionally provided by the machine manufacturer

Please refer to the machine manufacturer's instructions.

- Dwell time
  Set time after which execution on the machine is continued
- Programmed stop
  Stop execution on the machine if the "Programmed Stop" softkey is also active (see Sec. "Program control").
- Stop
  Stop execution on the machine

Line Circle | Machine funct. >

➢ Press the "Line Circle" and "Machine func." softkeys.

➢ Enter the desired parameters.

Accept

➢ Press the "Accept" softkey.

3.11 Inserting G code into the sequential control program

You can program G code blocks in a sequential control program. You can also insert comments to explain the program.

You will find a detailed description of G code blocks to DIN 66025 in:

References: /PG/, Programming Guide Fundamentals SINUMERIK 840D/840Di/810D
/PGA/, Programming Guide Advanced SINUMERIK 840D/840Di/810D

You cannot insert G code blocks before the program header, after the end of the program or within a chained sequence of program blocks.

ShopMill does not display G code blocks in programming graphics.

Feed

After each ShopMill cycle (drilling, milling, profile milling), the feed type G94 (mm/min) is always active, irrespective of the feed type that was programmed in the ShopMill cycle. The feed value F is only active after a ShopMill cycle if G94 was programmed in the G94 ShopMill cycle.

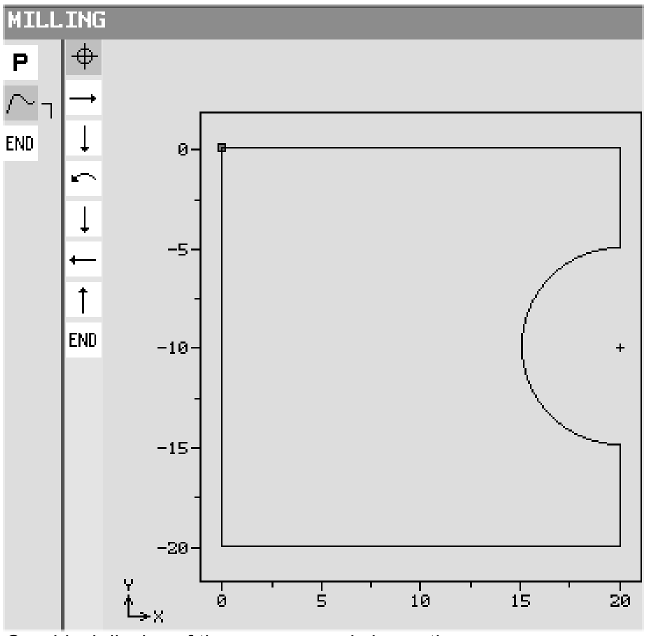
However, you should always program the feed type (G94 or G95) and the feed value (F) in the first G code block after a ShopMill cycle to avoid any unexpected types of motion.

FOR loop

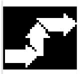
If you want to program a FOR loop in the sequential control program, you can use the global user variables (GUD7) _E_COUNTER [0] to _E_COUNTER [9] of type INT.

```
COUNTER_E
P    N5  COUNTER_E            Work offs 1 G54
T    N10 T=CUTTER_10 S1000U
G    N15
→    N20 RAPID X0 Y0 Z5
G    N25
→    N30 F200/min Z-5
G    N35 G64
G    N40
G    N45 FOR _E_COUNTER[0]=0 TO 3600
G    N50
G    N55   G1 X=_E_COUNTER[0]/20 Y=SIN(_E_COUNTER[0]/10)*70
G    N60 ENDFOR
→    N65 RAPID Z5
END      Program end          N=1
```

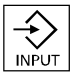
Example of loop programming (sine path))



Graphical display of the programmed sine path



- ➢ In the machining plan of a sequential control program, position the cursor on the program block after which you want to insert a G code block.



- ➢ Press the "Input" key.
- ➢ Enter the G code commands or comments.
  The comment must always start with a semicolon (;).

The newly created G code block is marked with a "G" in front of the block number in the machining plan.

G code in sequential control program

■

Programming with G Code

4.1 Creating a G code program

If you do not want to program with the ShopMill functions, you can also generate a G code program with G code commands in the ShopMill user interface.

You can program a G code command according to DIN 66025.
In addition, the parameter screens offer support for measuring and programming contours, drilling and milling cycles. G code is generated from the individual forms, which you can recompile back to the screen forms. The measuring cycle support function must be set up by the machine manufacturer.

Please refer to the machine manufacturer's instructions.

For a detailed description of G code commands to DIN 66025, and of cycles and measuring cycles, please refer to:

References:	/PG/,	Programming Guide Fundamentals SINUMERIK 840D/840Di/810D
	/PGA/,	Programming Guide Advanced SINUMERIK 840D/840Di/810D
	/PGZ/,	Cycles Programming Guide SINUMERIK 840D/840Di/810D
	/BNM/,	User Manual Measuring Cycles SINUMERIK 840D/840Di/810D

You can call up context-sensitive help, if you require more information on particular G code commands or cycle parameters on the PCU 50.

For a detailed description of the online help, please refer to:

References:	/BAD/,	Operator's Guide HMI Advanced SINUMERIK 840D/840Di/810D

Creating a G code program

Pro-gram

- Press the "Program" softkey.
- Select the directory in which you want to create a new program.

New | G code program

- Press the "New" and "G code program" softkeys.
- Enter a program name.
  Program names may be a maximum of 24 characters in length.
  You can use any letters, digits or the underscore symbol (_).
  ShopMill automatically replaces lower case with upper case.

OK -or- INPUT

➢ Press the "OK" softkey or the "Input" key.

The G code editor is opened.

➢ Enter the desired G code commands.

Calling a tool

Continue > Tools

➢ Select the "Continue" and "Tools" softkeys if you want to select a tool from the tool list.

-and-

➢ Place the cursor on the tool that you want to use for machining.

-and-

To program

➢ Press the "To program" softkey.

The selected tool is loaded into the G code editor.
Text such as the following is displayed at the current cursor position in the G code editor: T="MILL"

Unlike sequential control programming, the settings made in the tool management do not become active automatically when the tool is called.
This means that you must also program the tool change (M6), the direction of spindle rotation (M3/M4), the spindle speed (S...), the coolant (M7/M8) and any other tool-specific functions required.

Example:

```
...
T="MILL"            ;Call tool
M6                  ;Change tool
M7 M3 S2000         ;Deactivate coolant and spindle
...
```

Cycle support

Contour ... Mill

➢ Use the softkeys to select whether you want support for programming contours, drilling or milling cycles.

Create contour ...

➢ Select the cycle you want via the softkey.

➢ Enter the parameters.

OK

➢ Press the "OK" softkey.

The cycle is transferred to the editor as G code.

- Position the cursor on a cycle in the G code editor if you want to display the associated parameter screen form again.

Recompile

- Select the "Recompile" softkey.

The parameter screen for the selected cycle appears.

Edit

Select the "Edit" softkey if you want to go directly back to the G code editor from a parameter screen form.

Measuring cycle support

>

- Switch to the extended horizontal softkey bar.

Measure mill

- Press the "Measure mill" softkey.

Calibr. probe ...

- Select the required measuring cycle via the softkey.
- Enter the parameters.

OK

- Press the "OK" softkey.

The measuring cycle is transferred to the editor as G code.

- Position the cursor on a measuring cycle in the G code editor, if you want to display the associated parameter screen form again.

Recompile

- Select the "Recompile" softkey.

The parameter screen for the selected measuring cycle appears.

Edit

Select the "Edit" softkey if you want to return directly to the G code editor from a parameter screen form.

Online help (PCU 50)

- Place the cursor on a G code command in the G code editor or on an input field in a cycle support parameter screen form.

HELP

- Press the "Help" key.

The relevant help screen is displayed.

4.2 Running a G code program

During execution of a program, the workpiece is machined in accordance with the programming on the machine.
After the program is started in automatic mode, workpiece machining is performed automatically. You can, however, stop the program at any time and then resume execution later.

Execution of the program can be simulated graphically on the screen to enable you to check the programming result without moving the machine axes.
For more information about simulation, see Sec. "Simulation".

The following requirements must be met before executing a program on the machine:

- The measuring system of the control is synchronized with the machine.
- A program created in G code is available.
- The necessary tool offsets and work offsets have been entered.
- The necessary safety interlocks implemented by the machine manufacturer are activated.

When executing a G code program, the same functions are available as for executing a sequential control program (see Sec. "Machining a workpiece").

Simulating a G code program

Pro-gram -or- PROGRAM MANAGER

➢ Press the "Program" softkey or the "Program Manager" key.

➢ Position the cursor on the desired G code program.

INPUT -or- ▶

➢ Press the "Input" or "Cursor right" key.

The program is opened in the G code editor.

Simu-lation

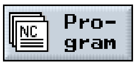
➢ Press the "Simulation" softkey.

Execution of the program will be displayed in full on the screen in graphical form.

Edit

Select the "Edit" softkey if you want to return directly to the G code editor from the simulation screen.

Running a G code program



-or- PROGRAM MANAGER

- Press the "Program" softkey or the "Program Manager" key.

-and-

- Position the cursor on the desired G code program.

-and-

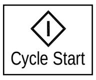
Execute

- Press the "Execute" softkey.

-or-

- Press the "Execute" softkey if you are in the "Program" operating area.

ShopMill automatically changes to "Machine Auto" operating mode and loads the G code program.



- Press the "Cycle Start" key.

Execution of the G code program starts on the machine.

4.3 G code editor

You use the G code editor when you want to change the sequence of program blocks within a G code program, delete program blocks or copy program blocks from one program to another.

When you want to change G code in a program that you are currently executing, you can only change the G code blocks that have not yet been executed. These blocks are highlighted.

The following functions are available in the G code editor:

- Select
  You can select any G code.
- Copy/paste
  You can copy and paste G code within a program or between different programs.
- Cut
  You can cut and therefore delete any G code. However, the G code remains in the buffer, so you can still paste it in somewhere else.
- Search/replace
  In a G code program, you can search for a specific character string and replace it with a different one.
- To start/end
  You can jump easily to the start or end of the G code program.
- Number
  If you insert a new or copied G code block between two existing G code blocks, ShopMill automatically assigns a new block number. This block number may be higher than the one in the following block. The "Renumber" function is used to renumber the G code blocks in ascending order.

The G code editor will be opened automatically if you write or open a G code program.

Selecting G code

➢ Place the cursor at the position in the program where you want your selection to start.

Mark

➢ Press the "Mark" softkey.

➢ Place the cursor at the position in the program where you want your selection to end.

The G code is selected.

Copying G code

➢ Select the G code that you want to copy.

Copy

➢ Press the "Copy" softkey.

The G code is stored in buffer memory and remains there even if you switch to another program.

Pasting G code

➢ Copy the G code that you want to insert.

Insert

➢ Press the "Insert" softkey.

The copied G code is pasted from buffer memory into the text in front of the cursor.

Cutting G code

➢ Select the G code that you want to cut.

Cut

➢ Press the "Cut" softkey.

The selected G code is removed and stored in buffer memory.

Finding G code

Search >

➢ Press the "Search" softkey.

A new vertical softkey bar appears.

➢ Enter the character string that you want to locate.

OK

➢ Press the "OK" softkey.

The G code program is searched for the character string in the forward direction. The character string is marked in the editor by the cursor.

Continue search

➢ Press the "Find next" softkey if you want to continue the search.

The next character string found is displayed.

Finding and replacing G code

Search >

➢ Press the "Search" softkey.

A new vertical softkey bar appears.

Search/ Replace

➢ Press the "Search/Replace" softkey.

➢ Enter the character string that you want to find and the characters that you want to insert in its place.

OK

➢ Press the "OK" softkey.

The G code program is searched for the character string in the forward direction. The character string is marked in the editor by the cursor.

Replace all

➢ Press the "Replace all" softkey if you want to replace the character string throughout the entire G code program.

-or-

Find next

➢ Press the "Find next" softkey if you want to continue the search without replacing the instance of the character string found.

-or-

Replace

➢ Press the "Replace" softkey if you want to replace the character string at this point in the G code program.

Jumping to start/end

Continue >

To start | To end

➢ Select the "Continue" and "To start" or "To end" softkeys.

The beginning or end of the G code program is displayed.

Renumber the G code blocks

Continue > | Re-number >

➢ Select the "Continue" and "Renumber" softkeys.

➢ Enter the number of the first block and the increment between block numbers (e.g. 1, 5, 10).

Accept

➢ Press the "Accept" softkey.

The blocks are renumbered.
You can cancel the numbering again by entering 0 for the increment or block number.

4.4 Arithmetic variables

Arithmetic variables (R variables) are variables that you can use within a G code program.
G code programs can read and write the variables. You can assign a value in the R variable list to variables that can be read.

Input and deletion of variables can be disabled via the keyswitch.

Displaying R variables

Tools WOs -or- OFFSET

- Press the "Tools WOs" softkey or the "Offset" key.

R vari.

- Press the "R vari." softkey.

The R variable list is opened.

Finding R variables

Search >

- Press the "Search" softkey.
- Enter the number of the variable you want to find.

Accept

- Press the "Accept" softkey.

The variable is displayed.

Editing R variables

- Place the cursor on the input field of the variable that you want to change.
- Enter the new value.

The new value of the variables is applied immediately.

Deleting R variables

- Place the cursor on the input field of the variable whose value you want to delete.

BACKSPACE

- Press the "Backspace" key.

The value of the variable is deleted.

4.5 ISO dialects

If ISO dialects are set up in ShopMill, you can also create and run ISO dialect programs.

Please refer to the machine manufacturer's instructions.

ISO dialect programs are not programs that were created with SIEMENS G code. See Section "Creating a G code program".

■

Simulation

5.1 General information

ShopMill provides various extensive and detailed simulation functions for displaying machining paths.

Please refer to the machine manufacturer's instructions.

Standard simulation

To simulate the machining process, the control system completely calculates the currently selected program and displays the result in graphical form.

You can select the following modes of representation for simulation:

- Plan view
- 3-plane view
- Volume model

The simulation function displays tools and workpiece contours in their correct proportions. Cylindrical die-sinking cutters, bevel cutters, bevel cutters with corner rounding and tapered die-sinking cutters are displayed as end milling tools.

The traverse paths for the tools are shown in color:
Red line = tool is moving at rapid traverse
Green line = tool is moving at machining feedrate

In all views, a clock is displayed during graphical processing. The displayed machining time (in hours/minutes/seconds) indicates the approximate time that would actually be required to execute the machining program on the machine (incl. tool change).
If a program is interrupted during simultaneous recording the clock stops.

In addition, the current axis coordinates and the program block that is currently executing are also displayed.
The active tool with the cutting edge number and feedrate are also displayed in the simulation.

Transformations

Transformations are displayed differently during simulation and simultaneous recording:

- Coordinate transformations (translation, scaling, …) are displayed as programmed.
- Cylinder surface transformations are displayed as a developed surface.
- After swivel transformation, the previous machining operations are deleted from the display and only machining of the swiveled plane is displayed (viewing angle perpendicular to the swiveled plane).
- Work offset (G54, …) do not alter the zero in the graphical display. With multiple clampings, machining of all the separate workpieces is drawn superimposed.

If you want to display a different portion of the workpiece from the one defined in ShopMill, you can define a new blank in the program (see Sec. "Changing program settings").

Quick display for mold making

Quick display of traverse paths is possible for large part programs. In this quick dashed-line drawing view, all programmed positions (even those resulting from work offsets) are shown as axis paths resulting from G1.

Please refer to the machine manufacturer's instructions.

Quick display for mold making is only available for the PCU 50.

5.2 Starting/stopping a program in standard simulation

Start simulation

Precondition

The required program

- sequential control program or
- G Code Program

has been selected and is in the program editor.

Simu-lation ... Standard

➢ Press the "Simulation" and "Standard" softkeys.

Details | Single Block

➢ Press the "Details" and "Single Block" softkeys, if you wish to execute the program block by block.

Execution of the program will be displayed on the screen in graphical form. The machine axes do not move.

In the case of sequential control programs, the dimensions of the blank for simulation are taken from the program header.

If a subroutine is called in the program, ShopMill evaluates the program header of the subroutine and uses the blank defined there for graphically displaying the part. The settings from the subroutine header remain effective, even after the subroutine has been executed. If you want to retain the blank used in the main program, delete the data relating to the blank in the subroutine header.

With G code programs, you must specify the dimensions of the blank or the selected viewport yourself.

Details Settings

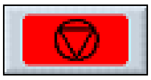
➢ With a G code program, select the "Details" and "Settings" softkeys and enter the dimensions of your choice (see also Sec. "Creating a new program; defining a blank").

These dimensions are stored for simulation of the next G code program. If you set the "Blank" parameter to "off", the dimensions will be deleted.

Feedrate override is also active during simulation.
0%: The simulation stops.
≥ 100%: The program is executed as fast as possible.
Feedrate override for simulation must be activated via a machine data code.

Please refer to the machine manufacturer's instructions.

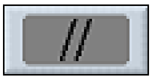
Stop simulation



➢ Press the "Stop" softkey.

Simulation is stopped.

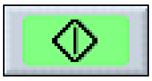
Aborting simulation



➢ Press the "Reset" softkey.

Simulation is aborted and the unmachined blank shape of the workpiece is displayed again.

Resuming simulation



➢ Press "Start" softkey.

Simulation is resumed.

Ending simulation

End

➢ Press the "End" softkey.

The machining plan or programming graphic for the program is displayed again.

5.3 Representation as a plan view

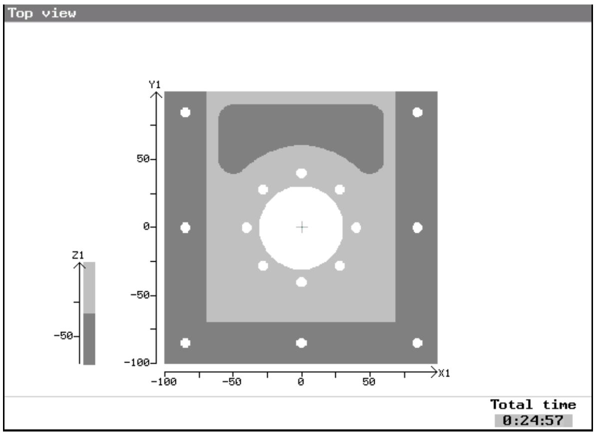
You can display the workpiece as a plan view by pressing this softkey.
A depth display indicates the current depth at which machining is currently taking place.
The rules for depth display in these graphics is:
"The deeper, the darker".

Display as a plan view

Plan view

➢ Press the "Plan view" softkey.

A plan view of the current workpiece is displayed.
Example of a plan view display of a workpiece:



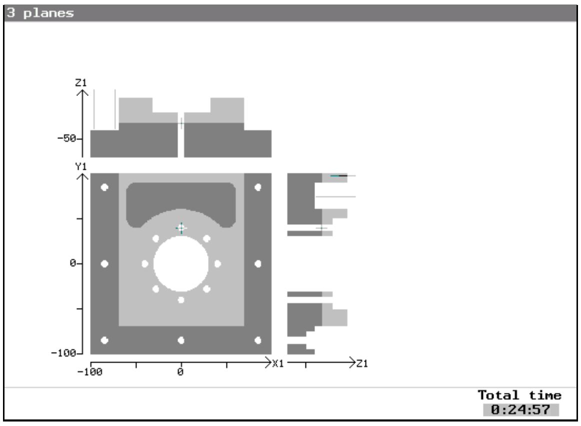
5.4 Representation as a 3-plane view

The process is represented as a plan view with 2 sections, similar to a technical drawing.
Functions for zooming viewports are provided in the 3-plane view.

Display as a 3-plane view

➢ Press the "3-Plane View" softkey.

A 3-plane view of the current workpiece is displayed.
Example of a 3-plane view of a workpiece:



Shifting section planes

The cross-hair can be positioned in the plan view to display the section plane in the relevant side view.

To reveal concealed contours, you can shift the section planes to any position you want in the 3-plane display. This way you can make hidden contours visible.

➢ Press a cursor key to move the section plane in the y plane.

-or-

➢ Press a cursor key to move the section plane in the x plane.

-or-

PAGE DOWN PAGE UP

➢ Press the "Page Down" or "Page Up" key to move the section plane in the y plane.

5.5 Enlarging a portion of the display

Functions for displaying a more detailed representation of a workpiece are available

- in the plan view and
- in the 3-plane display.

Details

➢ Press the "Details" softkey.

-and-

Zoom +

-or-

+

➢ Press the "Zoom +" softkey or the "+" key to enlarge the viewport.

-or

Zoom -

-or-

–

➢ Press the "Zoom -" softkey or the "-" key to reduce the viewport.

-or-

Auto Zoom

➢ Press the "Auto Zoom" softkey to fit the viewport to the size of the window automatically.

-or-

Back to original

➢ Press the "Back to original" softkey to restore the original size of the viewport.

-or-

▶ ◀ ▲ ▼

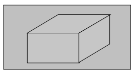
➢ Press a cursor key to move the viewport right, left, up, or down.

5.6 Three-dimensional display

The workpiece is displayed as a volume model. The simulation window displays the current machining status.
You can display concealed contours and views on the volume model by

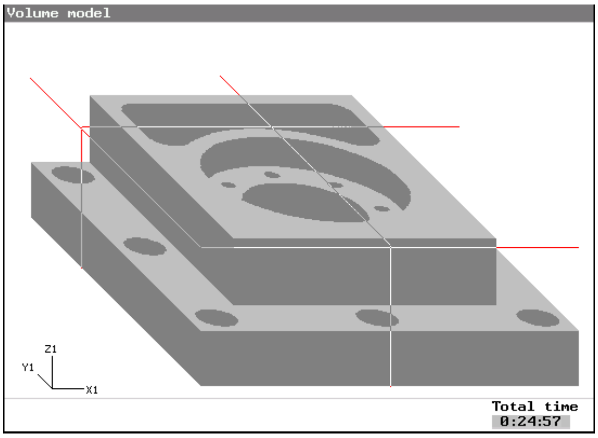
- changing the position about the vertical axis or
- cutting a section out of the volume model at the desired point.

Volume model



➢ Press the "Volume model".

The volume model of the current workpiece is displayed.
Example of a volume model:



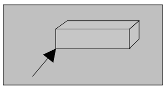
5.6.1 Changing the position of the viewport

You can select different views for viewing the volume module.

Details

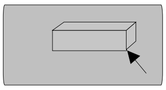
- Press the "Details" softkey.

-and-



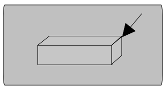
- Press this softkey to view the left side of the workpiece from the front.

-or-



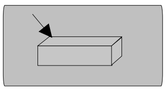
- Press this softkey to view the right side of the workpiece from the front.

-or-



- Press this softkey to view the right side of the workpiece from the back.

-or-

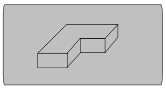


- Press this softkey to view the left side of the workpiece from the back.

5.6.2 Cutting a section out of the workpiece

You can cut a section out of the volume module to obtain certain views.

Precondition: You have selected one side of the workpiece.

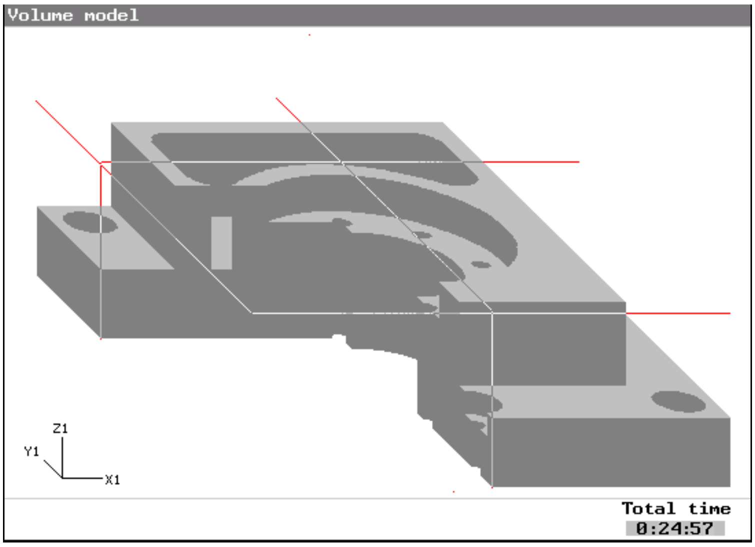


➢ Select the "Cut open" softkey.

Shifting section planes

To make concealed contours visible, shift the section planes using the cursor and "Paging" keys (see also Sec. "Representation in 3-plane view") to any position.

The new setting is displayed after a short update time.
Example of a section through a volume model:



5.7 Starting/stopping the quick display for mold making

Starting simulation

Precondition: A part program is selected in the Program Manager.

Simu-lation ... Mold Mak. G1 Blocks

➢ Press the "Simulation" and "Mold Making G1 Blocks" softkeys.

The program is shown in a two-line portion of the work window header. The first program block is highlighted.
Construction of the workpiece graphics starts.

Progress display: A message line below the visualized workpiece shows the percentage of the total program that is already shown in the graphics.

Ending simulation

End

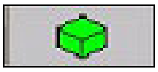
➢ Press the "End" softkey.

This takes you back to the Program Manager.
Changing operating areas interrupts graphics construction. If you return to the Program Manager operating area, it is resumed.

5.8 Views in the quick display

You can switch between the 2D- and 3D-views at any time. In the selected view, you can rotate the workpiece in any direction.

Selecting the 3D view

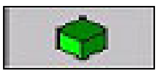


➢ Press the "3D-View" softkey.

Selecting 2D views

➢ Press the "X/Y-View" softkey.

- OR -



➢ Press the "X/Z-View" softkey.

- OR -

➢ Press the "Y/Z-View" softkey.

The workpiece is visualized in the selected view.

Changing the orientation in 3D

You can rotate the graphics in the X, Y, or Z axis direction.

Details Rotate

➢ Press the "Details" and "Rotate" softkeys.

Up

➢ Press the "Up" softkey.

- OR -

Down

➢ Press the "Down" softkey.

- OR -

Left

➢ Press the "Left" softkey.

- OR -

Right

➢ Press the "Right" softkey.

- OR -

▶ ◀ ▲ ▼

➢ Press one of the cursor keys.

You can see the result of the rotate command in the coordinate system in the lower left corner.

Accept

➢ Press the "Accept" softkey.

The commands are applied and the visualized workpiece is shown with its new axis orientation.

5.9 Zooming and panning the workpiece graphics

You can adjust the size of the displayed graphics to meet your requirements.

Zooming the view

Details | Enlarge Reduce

➢ Press the "Details" and "Enlarge Reduce" softkeys.

Enlarge -or- +

➢ Press the "Enlarge" softkey
or press the "+" key.

The graphics viewport is enlarged.

Reducing the view

Details | Enlarge Reduce

➢ Press the "Details" and "Enlarge Reduce" softkeys.

New softkeys appear on the vertical softkey bar.

Reduce -or- –

➢ Press the "Reduce" softkey
or press the "-" key.

The graphics viewport is reduced.

Automatic display size

Details | Enlarge Reduce

➢ Press the "Details" and "Enlarge Reduce" softkeys.

Autom. Size

➢ Press the "Autom. Size" softkey.

The viewport is fitted to the window size.
Automatic resizing takes account of the greatest extent of the workpiece in each axis.

Panning the graphics

Details | Pan

➢ Press the "Details" and "Pan" softkeys.

Up ... Right

➢ Press the "Up", "Down", "Left", or "Right" softkey.

- OR -

▶ ◀ ▲ ▼

➢ Press one of the cursor keys.

- OR -

Center

➢ Press the "Center" softkey.

The viewport is panned up, down, left, or right, or aligned in the center of the screen.

5.10 Distance measurement

It is possible to measure and display the direct path (spatial diagonal) between to points of the workpiece by marking two points in the graphics.

Measuring distance

Details | Distance

➢ Press the "Details" and "Distance" softkeys.

➢ Move the cross-hairs to the required position.

Mark Point A

➢ Press the "Mark Point A" softkey to define the first point.

Mark Point B

➢ Position the cursor on the second point and press the "Mark Point B" softkey.

The selected points are marked in the graphics.
The distance between the two points is calculated and output in the message line below the graphics display.
Repeat this process if you wish to measure more distances.

5.11 Search function

The Search function allows you to jump to the block of a selected position in the workpiece display.

Selecting a block in the graphics

Details | Search

➢ Press the "Details" and "Search" softkeys.

The mouse changes shape to a cross-hair.

Up ... Right

-and-

INPUT

➢ Press the "Up", "Down", "Left", or "Right" softkey to put the cross-hairs into the required position and confirm the point with the "Input" key.

- OR -

▶ ◀ ▲ ▼

-and-

INPUT

➢ Press one of the cursor keys to position the cross-hairs and confirm the point with the "Input" key.

- OR -

Block search

➢ Position the cross-hairs directly on the required point and press the "Block search" softkey.

The selected point is highlighted in color.
The block associated with the selected point is searched for and shown color-highlighted in the program section above the graphics display.
The **Edit** submenu provides another way of searching for certain blocks.

5.12 Editing part program blocks

During quick display, you are automatically in the G code editor. The program being visualized is open. There are various ways you can edit the part program shown here.

5.12.1 Selecting G blocks

There are various ways you can get to the block to be edited in the opened part program either directly or via a search function.

Searching via a string

Edit | Search

➢ Press the "Edit " and "Search" softkeys.

The "Search from cursor position" window opens.

➢ Enter a string in the "Search:" input field.

Search

➢ Press the "Search" softkey.

The search starts.
If a matching block is found, it is color-highlighted in the program section.

Searching via a block number

Edit | Go to

➢ Press the "Edit " and "Go to" softkeys.

The "Go to …" window opens.

OK

➢ Enter a G block in the "Block number …" input field and press the "OK" softkey.

The search starts.
If the matching block is found, it is displayed color-highlighted in the program section.

Jumping to start/end

Edit

➢ Press the "Edit" and "Beginning of program" or "End of program" softkeys.

Beginning of program | End of program

The first or last block of the opened part program is displayed color-highlighted in the program section.

Scrolling through the program

➢ Place the cursor in the program section.

▶ ◀ ▲ ▼

➢ Press one of the cursor keys.

You move up, down, left, or right in the part program.

Stopping a search

You can interrupt a search at any time.

Abort

➢ Press the "Abort" softkey.

5.12.2 Editing a G code program

Changing and saving G blocks

You can edit the selected block and then save it.

Edit

➢ Press the "Edit" softkey.

➢ Edit the selected block in the program section.

You are automatically in overwrite mode.
- OR -

Overwrite

➢ Press the "Overwrite" softkey.

Insert

The softkey changes to "Insert"
You can now insert blocks.

Save File

➢ Press the "Save File" softkey.

The changes are applied in the file.
The workpiece graphics are redrawn.

■

File Management

6.1 Program management with ShopMill

All programs that you have created in ShopMill for machining workpieces are stored in the main NC memory.
You can access these programs at any time via the Program Manager for execution, editing, copying, or renaming. Programs that you no longer require can be deleted to release their storage space.

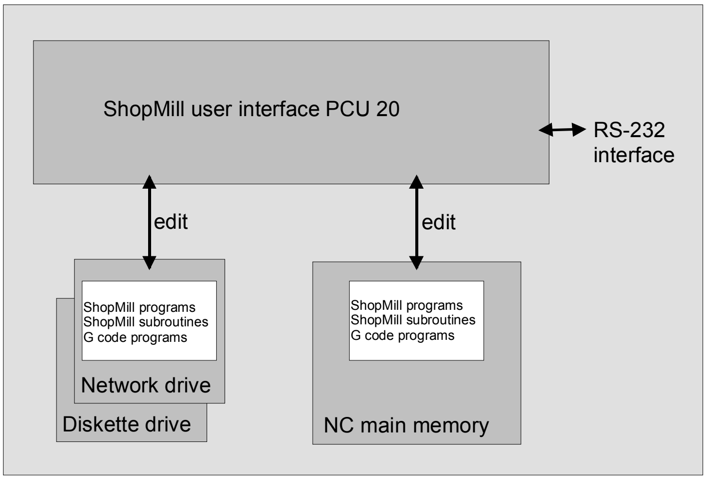
ShopMill offers several options for exchanging programs and data with other workstations:

- Own hard disk (PCU 50 only)
- RS-232 interface
- Floppy disk drive
- Network connection

The following Sections explain the alternate program management functions as used with PCU 20 or PCU 50.
Find out which PCU your ShopMill system is running on and then read either "Program management with PCU 20" or "Program management with PCU 50".

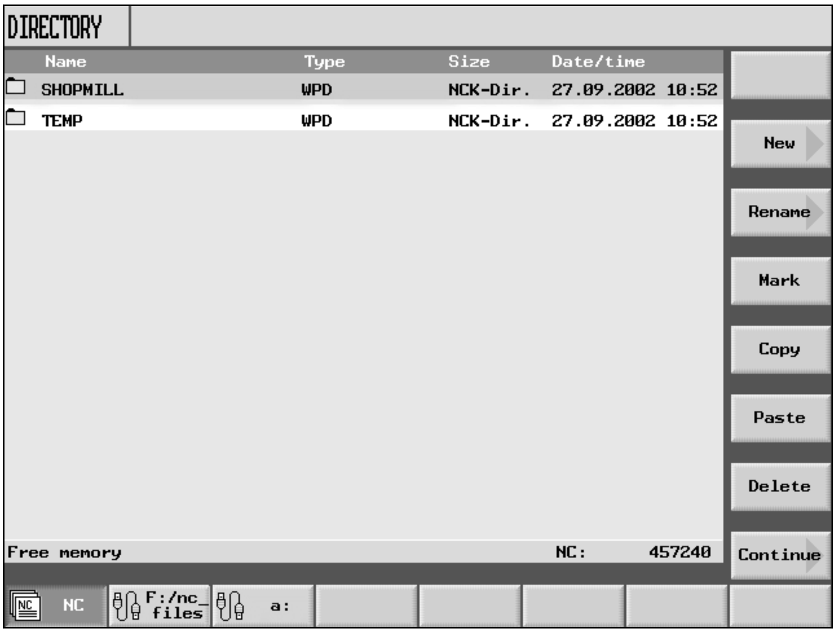
6.2 Program management with PCU 20

With the ShopMill variant with PCU 20, all programs and data are stored in the NC main memory.
You can exchange data and programs via a RS-232 interface.
It is also possible to display the directory tree for a floppy disk drive or network drive.



Data storage with PCU 20

You will find an overview of all directories and programs in the Program Manager.



PCU20 program manager

In the horizontal softkey bar, you can select the storage medium that contains the directories and programs that you want to display. In addition to the "NC" softkey, via which the NC main memory data can be displayed, a further 4 softkeys can also be assigned.

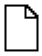
You can display the directories and programs on floppy disks and network drives:

Please refer to the machine manufacturer's instructions.

In the overview, the symbols in the left-hand column have the following meaning:

Directory

Program



Zero point/tool data

The directories and programs are always listed complete with the following information:

- Name
  The name can be up to 24 characters long. For data transfer to external systems, the name is truncated to 8 characters.
- Type
  Directory: WPD
  Program: MPF
  Zero point/tool data: INI
- Size (in bytes)
- Date/time (of creation or last change)

ShopMill stores the programs that are created internally for calculating the stock removal processes in the "TEMP" directory.

Information about memory allocation in the NC is displayed above the horizontal softkey bar.

Opening a directory

➢ Press the "Program" softkey or the "Program Manager" key.

The directory overview is displayed.

➢ Select the storage medium using the softkey.

➢ Place the cursor on the directory that you want to open.

➢ Press the "Input" or "Cursor right" key.

All the programs in this directory are displayed.

Returning to the next highest directory level

➢ Press the "Cursor left" key with the cursor in any line.

-or-

➢ Place the cursor on the Return line.

-and-

➢ Press the "Input" or "Cursor left" key.

The next highest directory level is displayed.

6.2.1 Opening a program

To view a program in more detail or modify it, you must display the machining plan for the program.

➢ Press the "Program" softkey.

The directory overview is displayed.

➢ Place the cursor on the program that you want to open.

➢ Press the "Input" or "Cursor right" key.

The selected program is opened in the "Program" operating area. The machining plan of the program is displayed.

6.2.2 Executing a program

You can select any program that is stored in your system at any time to machine workpieces automatically.

Pro-gram

- Open the Program Manager.
- Place the cursor on the program that you want to execute.

Execute

- Press the "Execute" softkey.

ShopMill now switches to "Machine Auto" operating mode and uploads the program.

Cycle Start

- Then press the "Cycle Start" key.

Workpiece machining is initiated (see also Sec. "Automatic mode".)

Ex-ecute

If the program is already open in the "Program" operating area, press the "Execute" softkey to load the program in "Machine Auto" mode. Then start machining of the workpiece by pressing the "Cycle Start" key.

6.2.3 Multiple clamping

The "Multiple clamping" function provides optimized tool change over several workpiece clampings. This shortens idle times because a tool performs all machining operations in all clampings before the next tool change is initiated.

You can use not only the surface clampings but also the "multiple clamping" function for rotating fixture plates. For this, the machine must have an additional rotary axis (e.g. A-axis) or a dividing unit.

Please refer to the machine manufacturer's instructions.

You can machine not only identical but also different workpieces with this function.

The "Multiple clamping for different programs" function is a software option.

ShopMill automatically generates a single program out of several programs. The tool sequence within a program remains unchanged. Cycles and subroutines are not opened, position patterns are processed as closed units.

The individual programs must meet the following requirements:

- Only sequential control programs (not G code programs)
- Programs must be executable
- Program for the 1st clamping must have been trial run
- No markers/repetitions, i.e. no branches in the program
- No inch/metric switchover
- No work offsets
- No coordinate transformation (translation, scaling, etc.)
- Contours must have unique names, i.e. the same contour name must not be called in several different programs.
- The "Starting point" parameter must not be set to "manual" in the stock removal cycle (contour milling).
- No modal settings, i.e. settings that are effective for all subsequent program blocks (only with multiple clamping for different programs)
- Max. of 50 contours per clamping
- Max. of 49 clampings

You can substitute subroutines for the markers or repetitions which may not be included in programs for multiple clampings.

Pro-gram

➢ Open the Program Manager.

Continue > | Multiple clamping

➢ Press the "Continue" and "Multiple clamping" softkeys.

➢ Specify the number of clampings and the number of the first work offset to be used.
The clampings are processed in ascending sequence from the start work offset. The work offsets are defined in the "Tools/Work Offsets" menu (see Sec. "Work offsets").

➢ Enter a name for the new, global program (XYZ.MPF).

OK

➢ Press the "OK" softkey.

A list is displayed in which the different programs must be assigned to the work offsets. Not all work offsets, i.e. clampings, must be assigned to programs, but at least two.

Program selection

➢ Press the "Program selection" softkey.

The program overview is displayed.

➢ Place the cursor on the required program.

OK

➢ Press the "OK" softkey.

The program is included in the assignment list.

➢ Repeat this process until a program is assigned to every required work offset.

On all clampings

➢ If you wish to execute the same program on all clampings, select "On all clampings" softkey.
You can assign different programs to individual work offsets first, and then assign one program to the remaining work offsets by selecting the "On all clampings" softkey.

Delete selection | Delete all

➢ Press the "Delete selection" or "Delete all" softkey if you want to clear individual or all programs from the assignment list.

Calculate program

➢ Press the "Calculate program" softkey when the assignment list is complete.

This optimizes the tool changes.
The global program is then renumbered. The number of the current clamping is specified every time the program switches from one clamping to another.
Apart from the global program (XYZ.MPF), the file XYZ_MCD.INI is also set up in which the assignment between work offsets and programs is stored. Both programs are stored in the directory that was previously selected in the Program Manager.

If you switch from the assignment list (without "Abort" or "Create program") to another function and then call the "Multiple clamping" function later on, the same assignment list is displayed again.

6.2.4 Running a G code program from floppy disk or network drive

If the capacity of your NC main memory is already stretched, you can also execute G code programs from a floppy disk or network drive. The entire G code program is not loaded into NC main memory before it is executed, but only the first part of it. Subsequent program blocks are then continuously reloaded as the first part is executed.

The G code program remains stored on the floppy disk/network drive when executed from there.

You cannot execute sequential control programs from floppy disk/network drive.

Pro-gram

➢ Open the Program Manager.

a: ...

➢ Select the floppy disk/network drive via the appropriate softkey.

➢ Place the cursor on the directory that contains the G code program you want to execute.

INPUT -or- ▶

➢ Press the "Input" or "Cursor right" key.

The directory opens.

➢ Place the cursor on the G code program you want to execute.

Continue > | Exec. from hard disk

➢ Select the "Continue" and "Exec. from hard disk" softkeys.

ShopMill switches to "Machine Auto" mode and uploads the G code program.

Cycle Start

➢ Press the "Cycle Start" key.

Workpiece machining is initiated (see also Sec. "Automatic mode"). The program contents are loaded continuously to the NC main memory while the program is being processed.

6.2.5 Creating a directory/program

Directory structures help you to manage your program and data transparently. You can create any number of subdirectories for this purpose in a directory.
You can also create programs in a subdirectory/directory and then create program blocks for the program (see Sec. "Programming with ShopMill").
The new program will be automatically stored in NC main memory for you to use.

Creating a directory

Pro-gram

➢ Open the Program Manager.

New > | Directory

➢ Press the "New" and "Directory" softkeys.

➢ Enter a new directory name.

OK

➢ Press the "OK" softkey.

The new directory is created.

Creating a program

Pro-gram

➢ Open the Program Manager.

➢ Place the cursor on the directory in which you want to create a new program.

INPUT -or- ▶

➢ Press the "Input" or "Cursor right" key.

New

➢ Press the "New" softkey.

ShopMill program

➢ Now press the "ShopMill program" softkey if you want to create a ShopMill program.
(See Sec. "Programming with ShopMill")

-or-

G code program

➢ Press the "G code program" softkey if you want to create a G code program
(See Sec. "Programming with G code")

6.2.6 Selecting multiple programs

You can mark several programs individually or in a block for subsequent copying, deleting, etc.

Selecting several programs as a block

Pro-gram

- Open the Program Manager.
- Place the cursor on the first program that you want to select.

Mark

- Press the "Mark" softkey.

▲ -or- ▼

- Expand the program selection area by pressing the cursor up or down key.

The entire block of programs is marked.

Selecting several programs individually

Pro-gram

- Open the Program Manager.
- Place the cursor on the first program that you want to select.

SELECT

- Press the "Select" key.

▲ -or- ▼

- Move the cursor to the next program that you want to select.

SELECT

- Press the "Select" key again.

The individually selected programs are marked.

6.2.7 Copying/renaming a directory or program

To create a new directory or program that is similar to an existing program, you can save time by copying the old directory or program and only changing selected programs or program blocks.
You can also use the copy and insert capabilities for directories and programs to exchange data with other ShopMill systems via diskette or the network drive.
You can also rename directories or programs.

It is not possible to rename a program when it is loaded in "Machine Auto" mode at the same time.

Copying a directory/program

Program

➢ Open the Program Manager.

➢ Place the cursor on the directory/program that you want to copy.

Copy

➢ Press the "Copy" softkey.

➢ Select the directory level in which you want to insert your copied directory/program.

Insert

➢ Press the "Insert" softkey.

The copied directory/program is inserted in the selected directory level. If a directory/program of the same name already exists in the directory level, a prompt asks whether you want to overwrite or insert it under a different name.

OK

➢ Press the "OK" softkey if you want to overwrite the directory/ program.

-or-

➢ Enter another name if you want to insert the program/directory under another name.

-and-

OK

➢ Press the "OK" softkey.

Renaming a directory/program

- Open the Program Manager.
- Place the cursor on the directory/program that you want to rename.

Rename

- Press the "Rename" softkey.
- Enter the name of the new directory or program in the "To:" field. The name must be unique, i.e. two directories or programs are not permitted to have the same name.

- Press the "OK" softkey.

The directory/program is renamed.

6.2.8 Deleting a directory/program

Delete programs or directories from time to time that you are no longer using to maintain the clarity of your data management system and to release NC main memory.
Back up this data beforehand on an external data medium if necessary (see Sec. "Importing/exporting a program via the RS-232 interface").

Please note that when you delete a directory, all programs, tool data and zero point data and subdirectories that this directory contains are deleted.

If you want to release space in NC main memory, delete the contents of the "TEMP" directory. ShopMill stores the programs that are created internally for calculating the stock removal processes in this directory.

- Open the Program Manager.
- Place the cursor on the directory/program that you want to delete.

Delete

- Press the "Delete" and "OK" softkeys.

The selected directory or program is deleted.

6.2.9 Running a program via the RS-232 interface

You can execute programs stored on external data media directly via the RS-232 interface, i.e. it is not necessary to read in these programs before you can machine a workpiece with them.
If a program needs more memory space for execution than is available in the NC main memory, for example, the program contents are continuously loaded via the RS-232 interface.

The RS-232 interface of the control and the external data medium must be compatible, i.e. you must make the same settings for each RS-232 interface.

NC Pro-gram

➢ Open the Program Manager.

Continue > | Execute RS-232

➢ Select the "Continue" and "Execute RS-232" softkeys.

RS-232 settings

➢ Press the "RS-232 settings" softkey to set up the interface.

➢ Enter the desired settings.

« Back

➢ Press the "Back" softkey.

The settings for the interface are saved.

➢ On the partner system, select the program that you want to execute.

➢ Start the transfer on the partner system.

Start

➢ Press "Start" softkey.

ShopMill switches to "Machine Auto" mode and uploads part of the program.

Cycle Start

➢ Then press the "Cycle Start" key.

Workpiece machining is initiated (see also Sec. "Automatic mode"). The program contents are loaded continuously to the NC main memory while the program is being processed. When the program has been executed via the RS-232 interface, the program remains stored on the external medium.

6.2.10 Importing/exporting a program via the RS-232 interface

Programs can be exchanged with other ShopMill stations via an external data storage system through the RS-232 interface.
The data export function can also be used to export data that is not currently required to release NC main memory. You can re-import the exported programs as they are needed.

When you import or export a program into/out of ShopMill, all ShopMill subroutines are transferred with it.
It is also possible to import or export more than one program in the same operation.

The RS-232 interface of the control and the external data medium must be compatible, i.e. you must make the same settings for each RS-232 interface.

Make sure that you set the correct file format (binary/PC, punched tape or punched tape/ISO format) on reading out. Otherwise the partner system will not be able to interpret the data.

Exporting a program

Pro-gram

➢ Open the Program Manager.

➢ Place the cursor on the program that you want to export.

Continue > | Read out

➢ Select the "Continue" and "Read out" softkeys.

RS-232 settings

➢ Press the "RS-232 settings" softkey to set up the interface.

➢ Enter the desired settings.

Back

➢ Press the "Back" softkey.

The settings for the interface are saved.

All files

➢ Press the "All files" softkey if you want to select all the programs displayed.

➢ Start the transfer on the partner system.

Start

➢ Press "Start" softkey.

The selected program and all its ShopMill subroutines are exported. The "Readout" window displays the name of the program that has just been exported and the number of bytes transferred.

Stop

- Press the "Stop" softkey if you want to interrupt data transfer.
- Then press the "Start" softkey again to restart data transfer.

Importing a program

NC Pro-gram

- Open the Program Manager.

Continue > | Read in

- Select the "Continue" and "Read in" softkeys.

RS-232 settings

- Press the "RS-232 settings" softkey to set up the interface.
- Enter the desired settings.

Back

- Press the "Back" softkey.

The settings for the interface are saved.

- On the partner system, select the programs that you want to read in.
- Start the transfer on the partner system.

Start

- Press "Start" softkey.

The "Read in" window displays the name of the program that has just been read in and the number of bytes transferred. The program is stored in the directory specified in the program header.

Stop

- Press the "Stop" softkey if you want to interrupt data transfer.
- Then press the "Start" softkey again to restart data transfer.

6.2.11 Displaying the error log

If errors occur during data transfer via the RS-232 interface, ShopMill records them in an error log.

Pro-gram

➢ Open the Program Manager.

Continue >

➢ Press the "Continue" softkey.

Read out -or- Read in

➢ Press the "Read in" or "Read out" softkey.

Error log

➢ Then press the "Error log" softkey.

The data transfer log is displayed.

6.2.12 Backing up/importing tool or zero point data

Apart from programs, you can also save tool data and zero point settings.
You can use this function, for example, to save the tool and zero point data for a specific sequential control program. If you want to execute this program at a later point in time, you will then have quick access to the relevant settings.

Even tool data that you have measured on an external tool setting station can be copied easily into the tool management system using this option. For further details, see:
References: /FBSP/, Description of Functions ShopMill

You can choose which data you want to back up:

- Tool data
- Magazine loading
- Zero points
- Basic zero point

You can also specify the amount of data to be backed up:

- Complete tool list or all zero points
- All tool data or zero points used in the program

You can only read out the magazine assignments if your system provides support for loading and unloading tool data to and from the tool-holding magazine (see Sec. "Loading/unloading tools").

Saving data

Pro-gram

- Open the Program Manager.
- Place the cursor on the program whose tool and zero point data you wish to back up.

Continue > | Back up data

- Select the "Continue" and "Back up data" softkeys.
- Select the data you want to back up.
- Change the suggested name if you want to.
  The name of the originally selected program with extension "..._TMZ" will be suggested as a name for your tool or zero point file.

OK

- Press the "OK" softkey.

The tool/zero point data will be set up in the same directory in which the selected program is stored.
If a tool/zero point file with the specified name already exists, this will now be overwritten with the new data.

Importing data

Pro-gram

- Open the Program Manager.
- Place the cursor on the tool/zero point data backup that you wish to re-import.

Execute -or- INPUT

- Select the "Execute" softkey or the "Input" key.

The window "Read in backup data" is opened.

- Select the data (tool offset data, magazine loading data, zero point data, basic work offsets) that you wish to import.

OK

- Press the "OK" softkey.

The data are read in.
Depending on which data you have selected, ShopMill will behave as follows:

All tool offset data
All data in the tool management system is deleted first. The backup data is then read in.

All tool offset data used in the program
If at least one of the tools to be read in already exists in the tool management system, you can choose between the following options.

Replace all

➢ Select the "Replace all" softkey to import all tool data. Any existing tools will now be overwritten without a warning prompt.

-or-

Replace none

➢ Select the "Replace none" softkey if you want to cancel the data import.

-or-

No

➢ Select the "No" softkey if you want to keep the old tool.
If the old tool is not at the saved magazine location, it is relocated there.

-or-

Yes

➢ Select the "Yes" softkey if you want to overwrite the old tool.

With the tool management option without loading/unloading, the old tool is deleted; the old tool is unloaded beforehand in the variant with loading/unloading.
If you change the tool name before importing it with "Yes", the tool will be added as an extra tool to the tool list.

Work offsets

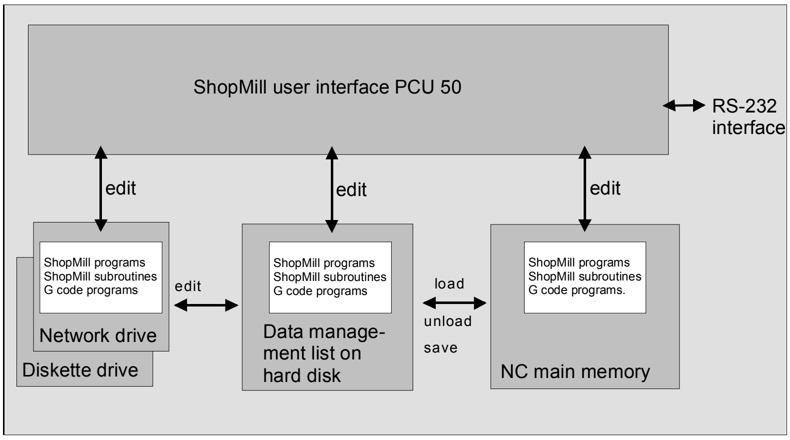
Existing work offsets are always overwritten when new offsets are imported.

Magazine loading

If magazine loading data are not imported at the same time, tools are entered without location number in the tool list.

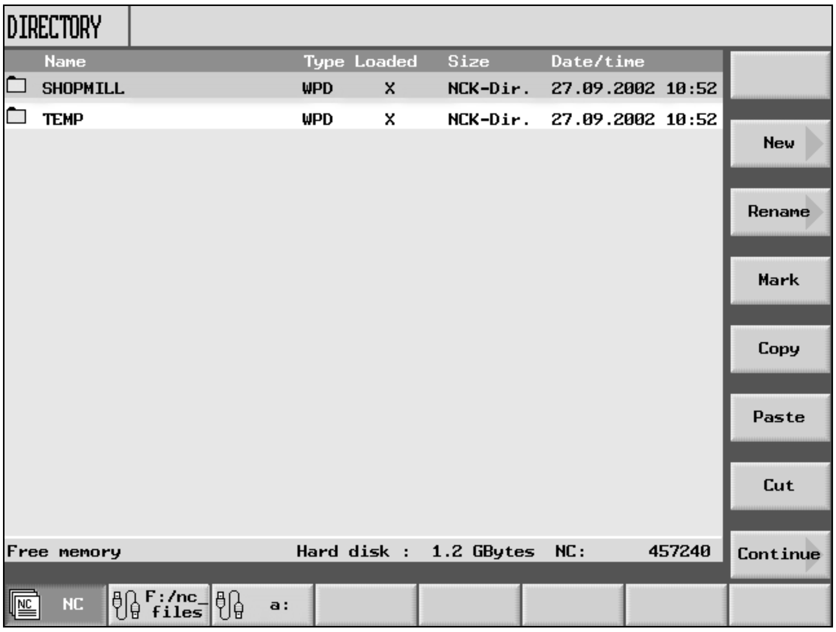
6.3 Program management with PCU 50

The ShopMill variant with PCU 50 has its own hard disk in addition to the NC main memory. This makes it possible to store all programs that are not currently required in the NC on the hard disk.
Display of the directory management of a floppy disk or network drive and programs is also possible, as are the import and export of data via an RS-232 interface.



Data storage with PCU 50

You will find an overview of all directories and programs in the Program Manager.



PCU 50 program manager

In the horizontal softkey bar, you can select the storage medium that contains the directories and programs that you want to display. In addition to the "NC" softkey, via which the data in the NC main memory and the data management directories on the hard disk can be displayed, a further 4 softkeys can also be assigned.

You can display the directories and programs of the following storage media:

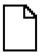
- Network drives (network card necessary)
- Floppy disk drive
- The hard disk as archive directory.

Please refer to the machine manufacturer's instructions.

In the overview, the symbols in the left-hand column have the following meaning:

Directory

Program



Zero point/tool data

The directories and programs are always listed complete with the following information:

- Name
  The name can be up to 24 characters long. For data transfer to external systems, the name is truncated to 8 characters.
- Type
  Directory: WPD
  Program: MPF
  Zero point/tool data: INI
- Loaded
  A cross in the "Loaded" column indicates whether the program is still in NC main memory (X) or whether it has been read out to hard disk (   ).
- Size (in bytes)
- Date/time (of creation or last change)

ShopMill stores the programs that are created internally for calculating the stock removal processes in the "TEMP" directory.

Information about memory allocation on the hard disk and in the NC is displayed above the horizontal softkey bar.

Opening a directory

Pro- gram -or- PROGRAM MANAGER

➢ Press the "Program" softkey or the "Program Manager" key.

The directory overview is displayed.

NC ...

➢ Select the storage medium using the softkey.

➢ Place the cursor on the directory that you want to open.

INPUT -or- ▶

➢ Press the "Input" or "Cursor right" key.

All the programs in this directory are displayed.

Returning to the next highest directory level

◀

➢ Press the "Cursor left" key with the cursor in any line.

-or-

..

➢ Place the cursor on the Return line.

-and-

INPUT -or- ◀

➢ Press the "Input" or "Cursor left" key.

The next highest directory level is displayed.

6.3.1 Opening a program

To view a program in more detail or modify it, you must display the machining plan for the program.

Pro- gram

➢ Press the "Program" softkey.

The directory overview is displayed.

➢ Place the cursor on the program that you want to open.

INPUT -or- ▶

➢ Press the "Input" or "Cursor right" key.

The selected program is opened in the "Program" operating area. The machining plan of the program is displayed.

6.3.2 Executing a program

You can select any program that is stored in your system at any time to machine workpieces automatically.

Pro-
gram

- Open the Program Manager.
- Place the cursor on the program that you want to execute.

Execute

- Press the "Execute" softkey.

ShopMill now switches to "Machine Auto" operating mode and uploads the program.

Cycle Start

- Then press the "Cycle Start" key.

Workpiece machining is initiated (see also Sec. "Automatic mode".)

Ex-
ecute

If the program is already open in the "Program" operating area, press the "Execute" softkey to load the program in "Machine Auto" mode. Then start machining of the workpiece by pressing the "Cycle Start" key.

6.3.3 Multiple clamping

The "Multiple clamping" function provides optimized tool change over several workpiece clampings. This shortens idle times because a tool performs all machining operations in all clampings before the next tool change is initiated.

You can use not only the surface clampings but also the "multiple clamping" function for rotating fixture plates. For this, the machine must have an additional rotary axis (e.g. A-axis) or a dividing unit.

Please refer to the machine manufacturer's instructions.

You can machine not only identical but also different workpieces with this function.

The "Multiple clamping for different programs" function is a software option.

ShopMill automatically generates a single program out of several programs. The tool sequence within a program remains unchanged. Cycles and subroutines are not opened, position patterns are processed as closed units.

The individual programs must meet the following requirements:

- Only sequential control programs (not G code programs)
- Programs must be executable
- Program for the 1st clamping must have been trial run
- No markers/repetitions, i.e. no branches in the program
- No inch/metric switchover
- No work offsets
- No coordinate transformation (translation, scaling, etc.)
- Contours must have unique names, i.e. the same contour name must not be called in several different programs.
- The "Starting point" parameter must not be set to "manual" in the stock removal cycle (contour milling).
- No modal settings, i.e. settings that are effective for all subsequent program blocks (only with multiple clamping for different programs)
- Max. of 50 contours per clamping
- Max. of 99 clampings

You can substitute subroutines for the markers or repetitions which may not be included in programs for multiple clampings.

Pro-gram

- Open the Program Manager.

Continue > | Multiple clamping

- Press the "Continue" and "Multiple clamping" softkeys.
- Specify the number of clampings and the number of the first work offset to be used.
  The clampings are processed in ascending sequence from the start work offset. The work offsets are defined in the "Tools/Work Offsets" menu (see Sec. "Work offsets").
- Enter a name for the new, global program (XYZ.MPF).

OK

- Press the "OK" softkey.

A list is displayed in which the different programs must be assigned to the work offsets. Not all work offsets, i.e. clampings, must be assigned to programs, but at least two.

Program selection

- Press the "Program selection" softkey.

The program overview is displayed.

- Place the cursor on the required program.

OK

- Press the "OK" softkey.

The program is included in the assignment list.

- Repeat this process until a program is assigned to every required work offset.

On all clampings

- If you wish to execute the same program on all clampings, select "On all clampings" softkey.
  You can assign different programs to individual work offsets first, and then assign one program to the remaining work offsets by selecting the "On all clampings" softkey.

Delete selection | Delete all

- Press the "Delete selection" or "Delete all" softkey if you want to clear individual or all programs from the assignment list.

Calculate program

- Press the "Calculate program" softkey when the assignment list is complete.

This optimizes the tool changes.
The global program is then renumbered. The number of the current clamping is specified every time the program switches from one clamping to another.

Apart from the global program (XYZ.MPF), the file XYZ_MCD.INI is also set up in which the assignment between work offsets and programs is stored. Both programs are stored in the directory that was previously selected in the Program Manager.
If you switch from the assignment list (without "Abort" or "Create program") to another function and then call the "Multiple clamping" function later on, the same assignment list is displayed again.

6.3.4 Loading/unloading a program

If you do not want to execute a program in the near future, you can unload it from NC main memory. The program is then stored on hard disk and NC main memory is free again.

As soon as you execute a program that was stored on hard disk, it is loaded into NC main memory again.
You can, however, also load one or more sequential control programs in the NC main memory memory without executing them immediately.

Programs that are in "Machine Auto" mode cannot be unloaded from NC main memory to the hard disk.

Unloading a program

Program

➢ Open the Program Manager.

➢ Place the cursor on the program that you want to unload from NC main memory.

Continue > | Manual Unload

➢ Press the "Continue" and "Manual Unload" softkeys.

The selected program is no longer marked with an "**X**" in the "Loaded" column.
In the line in which the available memory space is displayed, you can see that NC main memory has become free again.

Loading a program

Pro-gram

➢ Open the Program Manager.

➢ Place the cursor on the program that you want to load into NC main memory.

Continue > | Load manual

➢ Press the "Continue" and "Load manual" softkeys.

The selected program is now marked with an "**X**" in the "Loaded" column.

6.3.5 Executing a G code program from the hard disk, floppy disk or network drive

If the capacity of your NC main memory is already stretched, you can also execute G code programs from the hard disk or a floppy disk or network drive.
The entire G code program is not loaded into NC main memory before it is executed, but only the first part of it. Subsequent program blocks are then continuously reloaded as the first part is executed.

The G code program remains stored on the hard disk or floppy disk/network drive when executed from there.

You cannot execute sequential control programs from hard disk or floppy disk/network drive.

Running a G code program from the hard disk

Pro-gram

➢ Open the Program Manager.

➢ Place the cursor on the directory that contains the G code program that you want to execute from hard disk.

INPUT -or- ▶

➢ Press the "Input" or "Cursor right" key.

The program overview is displayed.

➢ Place the cursor on the G code program that you want to execute from hard disk (without "X").

Continue > | Exec. from hard disk

➢ Select the "Continue" and "Exec. from hard disk" softkeys.

ShopMill switches to "Machine Auto" mode and uploads the G code program.

Running a G code program from floppy disk or network drive

Pro-gram

➢ Open the Program Manager.

a: ...

➢ Select the floppy disk/network drive via the appropriate softkey.

➢ Place the cursor on the directory that contains the G code program you want to execute.

INPUT -or- ▶

➢ Press the "Input" or "Cursor right" key.

The directory opens.

➢ Place the cursor on the G code program you want to execute.

Continue > | Exec. from hard disk

➢ Select the "Continue" and "Exec. from hard disk" softkeys.

ShopMill switches to "Machine Auto" mode and uploads the G code program.

Cycle Start

➢ Press the "Cycle Start" key.

Workpiece machining is initiated (see also Sec. "Automatic mode"). The program contents are loaded continuously to the NC main memory while the program is being processed.

6.3.6 Creating a directory/program

Directory structures help you to manage your program and data transparently. You can create any number of subdirectories for this purpose in a directory.
You can also create programs in a subdirectory/directory and then create program blocks for the program (see Sec. "Programming with ShopMill").
The new program will be automatically stored in NC main memory for you to use.

Creating a directory

Pro- gram

- Open the Program Manager.

New | Directory

- Press the "New" and "Directory" softkeys.
- Enter a new directory name.

OK

- Press the "OK" softkey.

The new directory is created.

Creating a program

Pro- gram

- Open the Program Manager.
- Place the cursor on the directory in which you want to create a new program.

INPUT -or- ▶

- Press the "Input" or "Cursor right" key.

New

- Press the "New" softkey.

ShopMill program

- Now press the "ShopMill program" softkey if you want to create a ShopMill program.
(See Sec. "Programming with ShopMill")

-or-

G code program

- Press the "G code program" softkey if you want to create a G code program
(See Sec. "Programming with G code")

6.3.7 Selecting multiple programs

You can select several programs individually or in a block for subsequent copying, deleting, etc.

Selecting several programs as a block

Pro-gram

- Open the Program Manager.
- Place the cursor on the first program that you want to select.

Mark

- Press the "Mark" softkey.

▲ -or- ▼

- Expand the program selection area by pressing the cursor up or down key.

The entire block of programs is marked.

Selecting several programs individually

Pro-gram

- Open the Program Manager.
- Place the cursor on the first program that you want to select.

SELECT

- Press the "Select" key.

▲ -or- ▼

- Move the cursor to the next program that you want to select.

SELECT

- Press the "Select" key again.

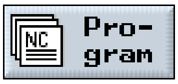
The individually selected programs are marked.

6.3.8 Copying/renaming/moving directories/programs

To create a new directory or program that is similar to an existing program, you can save time by copying the old directory or program and only changing selected programs or program blocks.
You can also move directories or programs or rename them.
You can also use the copy, cut and insert capabilities for directories and programs to exchange data with other ShopMill systems via diskette or the network drive.

It is not possible to rename a program when it is loaded in "Machine Auto" mode at the same time.

Copying a directory/program



➢ Open the Program Manager.

➢ Place the cursor on the directory/program that you want to copy.

Copy

➢ Press the "Copy" softkey.

➢ Select the directory level in which you want to insert your copied directory/program.

Insert

➢ Press the "Insert" softkey.

The copied directory/program is inserted in the selected directory level. If a directory/program of the same name already exists in the directory level, a prompt asks whether you want to overwrite or insert it under a different name.

➢ Press the "OK" softkey if you want to overwrite the directory/program.

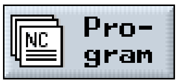
-or-

➢ Enter another name if you want to insert the program/directory under another name.

-and-

➢ Press the "OK" softkey.

Renaming a directory/program



- Open the Program Manager.
- Place the cursor on the directory/program that you want to rename.

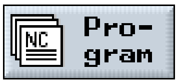
Rename

- Press the "Rename" softkey.
- Enter the name of the new directory or program in the "To:" field. The name must be unique, i.e. two directories or programs are not permitted to have the same name.

- Press the "OK" softkey.

The directory/program is renamed.

Moving a directory/program



- Open the Program Manager.
- Place the cursor on the directory/program that you want to move.

Cut

- Press the "Cut" softkey and then the "OK" softkey.

The selected directory/program is deleted at this point and stored in buffer memory.

- Select the directory level in which you want to insert the directory/program.

Insert

- Press the "Insert" softkey.

The directory/program is moved to the selected directory level.
If a directory/program of the same name already exists in this directory level, a prompt asks whether you want to overwrite or insert it under a different name.

- Press the "OK" softkey if you want to overwrite the directory/program.

-or-

- Enter another name if you want to insert the program/directory under another name.

-and-

- Press the "OK" softkey.

6.3.9 Deleting a directory/program

Delete programs or directories from time to time that you are no longer using to maintain a clearer overview of your data management. Back up this data beforehand on an external data medium if necessary (see Sec. "Importing/exporting a program via the RS-232 interface").

Please note that when you delete a directory, all programs, tool data and zero point data and subdirectories that this directory contains are deleted.

If you want to release space in NC main memory, delete the contents of the "TEMP" directory. ShopMill stores the programs that are created internally for calculating the stock removal processes in this directory.

Pro-
gram

- Open the Program Manager.
- Place the cursor on the directory/program that you want to delete.

Cut

OK

- Press the "Cut" and "OK" softkeys.

The selected directory or program is deleted.

6.3.10 Importing/exporting a program via the RS-232 interface

Programs can be exchanged with other ShopMill stations via an external data storage system through the RS-232 interface.
The data read out function can also be used to export data that is not currently required to release NC main memory or hard disk space.
You can re-import the exported programs as they are needed.

When you import or export a program into/out of ShopMill, all ShopMill subroutines are transferred with it.
It is also possible to import or export more than one program in the same operation.

The RS-232 interface of the control and the external data medium must be compatible, i.e. you must make the same settings for each RS-232 interface.

Make sure that you set the correct file format (binary/PC, punched tape or punched tape/ISO format) on reading out. Otherwise the partner system will not be able to interpret the data.

Exporting a program

Program	➢ Open the Program Manager.
	➢ Place the cursor on the program that you want to export.
Continue > / Read out	➢ Select the "Continue" and "Read out" softkeys.
RS-232 settings	➢ Press the "RS-232 settings" softkey to set up the interface.
	➢ Enter the desired settings.
Back	➢ Press the "Back" softkey.
	The settings for the interface are saved.
All files	➢ Press the "All files" softkey if you want to read out all the programs displayed.
	➢ Start the transfer on the partner system.
Start	➢ Press "Start" softkey.
	The selected program and all its ShopMill subroutines are read out.

Stop

The "Readout" window displays the name of the program that has just been read out and the number of bytes transferred.

- Press the "Stop" softkey if you want to interrupt data transfer.
- Then press the "Start" softkey again to restart data transfer.

Importing a program

Pro-gram

- Open the Program Manager.

Continue > | Read in

- Select the "Continue" and "Read in" softkeys.

RS-232 settings

- Press the "RS-232 settings" softkey to set up the interface.
- Enter the desired settings.

Back

- Press the "Back" softkey.

The settings for the interface are saved.

- On the partner system, select the programs that you want to read in.
- Start the transfer on the partner system.

Start

- Press "Start" softkey.

The "Read in" window displays the name of the program that has just been read in and the number of bytes transferred. The program is stored in the directory specified in the program header.

Stop

- Press the "Stop" softkey if you want to interrupt data transfer.
- Then press the "Start" softkey again to restart data transfer.

6.3.11 Displaying the error log

If errors occur during data transfer via the RS-232 interface, ShopMill records them in an error log.

Pro- gram

➢ Open the Program Manager.

Continue >

➢ Press the "Continue" softkey.

Read out -or- Read in

➢ Press the "Read out" or "Read in" softkey.

Error log

➢ Then press the "Error log" softkey.

The data transfer log is displayed.

6.3.12 Backing up/importing tool or zero point data

Apart from programs, you can also save tool data and zero point settings.
You can use this function, for example, to save the tool and zero point data for a specific sequential control program. If you want to execute this program at a later point in time, you will then have quick access to the relevant settings.

Even tool data that you have measured on an external tool setting station can be copied easily into the tool management system using this option. For further details, see:
References: /FBSP/, Description of Functions ShopMill

You can choose which data you want to back up:

- Tool data
- Magazine loading
- Zero points
- Basic zero point

You can also specify the amount of data to be backed up:

- Complete tool list or all zero points
- All tool data or zero points used in the program

You can only read out the magazine assignments if your system provides support for loading and unloading tool data to and from the tool-holding magazine (see Sec. "Loading/unloading tools").

Saving data

Pro-gram

- Open the Program Manager.
- Place the cursor on the program whose tool and zero point data you wish to back up.

Continue > | Back up data

- Select the "Continue" and "Back Up Data" softkeys.
- Select the data you want to back up.
- Change the suggested name if you want to.
  The name of the originally selected program with extension "..._TMZ" will be suggested as a name for your tool or zero point file.

OK

- Press the "OK" softkey.

The tool/zero point data will be set up in the same directory in which the selected program is stored.
If a tool/zero point file with the specified name already exists, this will now be overwritten with the new data.

Importing data

Pro-gram

- Open the Program Manager.
- Place the cursor on the tool/zero point data backup that you wish to re-import.

Execute -or- INPUT

- Select the "Execute" softkey or the "Input" key.

The window "Read in backup data" is opened.

- Select the data (tool offset data, magazine loading data, zero point data, basic work offsets) that you wish to read in.

OK

- Press the "OK" softkey.

The data are read in.
Depending on which data you have selected, ShopMill will behave as follows:

All tool offset data
All data in the tool management system is deleted first. The backup data is then read in.

All tool offset data used in the program
If at least one of the tools to be read in already exists in the tool management system, you can choose between the following options.

Replace all

➢ Select the "Replace all" softkey to import all tool data. Any existing tools will now be overwritten without a warning prompt.

-or-

Replace none

➢ Select the "Replace none" softkey if you want to cancel the data import.

-or-

No

➢ Select the "No" softkey if you want to keep the old tool.
If the old tool is not at the saved magazine location, it is relocated there.

-or-

Yes

➢ Select the "Yes" softkey if you want to overwrite the old tool.

With the tool management option without loading/unloading, the old tool is deleted; the old tool is unloaded beforehand in the variant with loading/unloading.
If you change the tool name before importing it with "Yes", the tool will be added as an extra tool to the tool list.

Work offsets
Existing work offsets are always overwritten when new offsets are imported.

Magazine loading
If magazine loading data are not imported at the same time, tools are entered in the tool list without a location number.

■

Mold Making

7.1 Requirements

This chapter only describes special aspects of mold making with ShopMill. A detailed description of the ShopMill functions is given in the previous chapters.

ShopMill can process not only sequential control programs but also G code mold-making programs. The precondition for this is optimization of the drives.

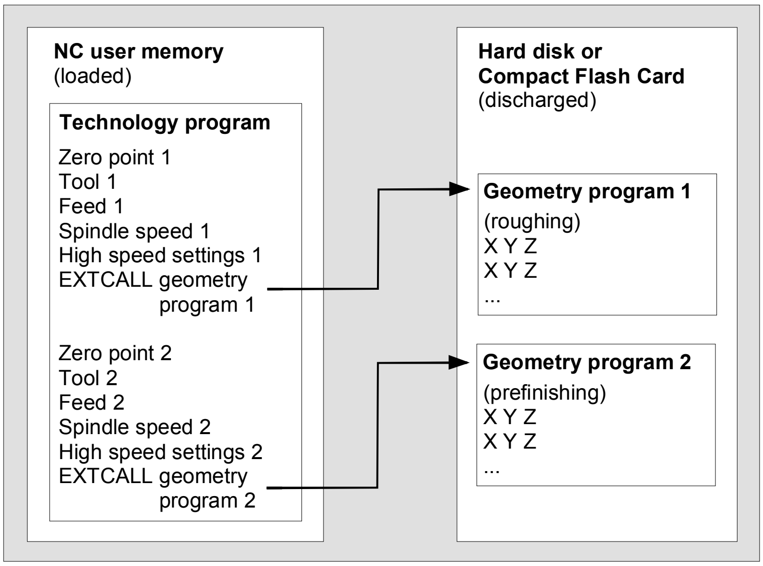
Please refer to the machine manufacturer's instructions.

Depending on the machine type, 3-axis mold making applications and also dynamic 5-axis machining operations can be performed.
You can also use the functions integrated into ShopMill to set up the machine, such as determining the zero of the workpiece or measuring tools for mold making programs.

Program structure and storage

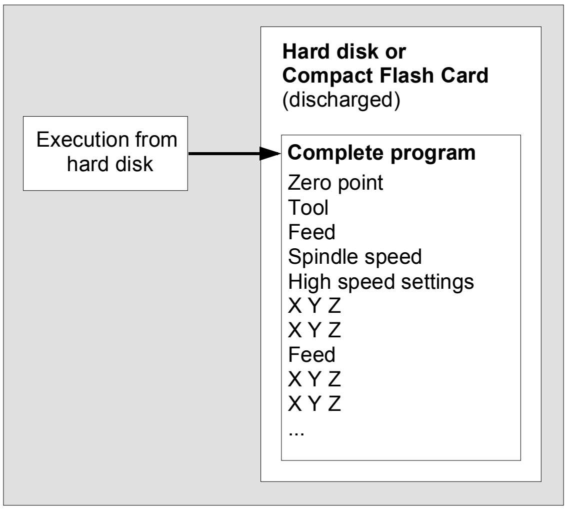
To achieve optimum velocity control for your mold-making programs, you should split the mold-making program into a central technology program and a separate geometry program rather than creating a single complete program.

- Technology program
  The technology program contains basic settings such as work offset, tool call-up, feed values, spindle speed, and control commands for velocity control. The technology program also calls the geometry programs as subroutines.
  You can create the technology program in ShopMill's G code editor.
- Geometry program
  The geometry programs of each type of operation (roughing, rough-finishing, and finishing) contain only the geometry values of the free-form surface to be machined.
  The geometry programs are created on an external CAM system in the form of G01 blocks.
  Depending on the application, the size of geometry programs ranges from 500 KB to 100 MB. Programs of this size can no longer be processed directly in the NC RAM. This means that the geometry programs must be saved either on the hard drive of the PCU 50 or on a Compact Flash card in the PCU 20.
  Storage on a network drive is only recommended if there is a point-to-point connection between the control and the server because only then is uninterrupted data transmission ensured. Storage on hard disk or compact flash card are preferable.



Program structure technology program with geometry programs

- Complete program
  Complete programs contain both the basic settings, such as work offset, tool call, etc. and the geometry values of the free-form surfaces to be machined. However, programming optimum velocity control is very complicated in a complete program.
  Complete programs are also created on external CAM systems. Because of their size, the complete programs are stored on the hard disk of the PCU 50 or on the compact flash card of the PCU 20.
  Storage on a network drive is also only recommended for a point-to-point connection between the control and server.



Program structure complete program

Data transfer

If you want to copy a geometry program or complete program from a network drive to the control, you must always use an Ethernet connection. The data transfer rate of the serial interface (RS232, V.24) is too low for the transfer of very large part programs.

7.2 Setting up the machine

7.2.1 Measuring the tool

The CAM system already takes the tool geometry into account when the geometry program is created. The tool path calculated refers either to the tool tip or the tool center. This means that, to determine the length of the tool, you must user the same reference point as the CAM system (tool tip or tool center).

If you use a ShopMill function to measure your tools, the tool length refers to the tool tip. On the other hand, if the tool center is taken into account in calculation of the tool path in the CAM system, you must subtract the radius of the tool from the tool length in the tool list.

Entry of the tool diameter in the tool list is not relevant to the processing of mold-making programs. However, you should still enter the tool diameter in the tool list for information.

7.3 Creating a program

7.3.1 Creating a program

For the technology program, you create a new G code program in the program manager and then process it in the G code editor. A sequential control program is not suitable for use as a technology program.

The geometry program or complete program is created on an external CAM system. For example, if you subsequently want to insert comments into the geometry program or change the tool name in the complete program, this can be done in ShopMill's G code editor.

7.3.2 Programming a tool

If you program a tool in the technology program, please note the following:
The geometry of the programmed tool must match the tool geometry used by the CAM system when the geometry program was created.

7.3.3 Programming the "High Speed Settings" cycle

Machining of free-form surfaces involves high velocity, precision, and surface quality requirements .
You can achieve optimum velocity control depending on the type of processing (roughing, rough-finishing, finishing) very simply with the "High Speed Settings" cycle.

You can call the cycle via the cycle support in the G code editor. The output tolerance of the postprocessor of the CAM system is usually entered in the "Tolerance" parameter.

Program the cycle before the geometry program call in the technology program.
More information on the cycle can be found in:
References: /PGZ/, Cycles Programming Guide
SINUMERIK 840D/840Di/810D

7.3.4 Subroutine call

The geometry program is called from the technology program as a subroutine. Because the geometry programs are not stored in the NC working memory but on the hard disk of the PCU 50 or on the compact flash card of the PCU 20 or on a network drive, you only have to call the subroutine with the G code command "EXTCALL".

PCU 50

The technology program and the geometry programs are in the same directory as the hard disk.

`EXTCALL "Geometry_program"`

Example: `EXTCALL "ROUGHING"`

PCU 20

The program syntax varies slightly depending on the storage location of the geometry program on the compact flash card.

- The geometry program is directly on the compact flash card.
  `EXTCALL ("C:\Geometry_program.mpf")`
  Example: `EXTCALL ("C:\Roughing.mpf")`
- The geometry program is in a directory on the compact flash card.
  `EXTCALL ("C:\Directory\Geometry_program.mpf")`
  Example: `EXTCALL ("C:\Mold\Roughing.mpf")`

Network drive

If the geometry program is on a network drive connected via the Ethernet, the program syntax is as follows.

`EXTCALL ("Path\Geometry_program.mpf")`

Example: `EXTCALL ("H:\Mold\Roughing.mpf")`

7.4 Executing a program

7.4.1 Selecting a program for execution

The technology program that is located in the NC working memory is selected for execution just like a normal G code program. The geometry program is then automatically selected via the G code command "EXTCALL".

A complete program, which is located either on the hard disk or the PCU 50 or on the compact flash card of the PCU 20 or on a network drive, is selected with the "Execute HD" sofkey in the program manager.
Processing via the V.24 interface on the PCU 20 is not recommended due to the low data transfer speed.

7.4.2 Starting execution at a specific point in the program

Technology program with geometry programs

To start execution of a certain program section in a geometry program, enter the destination in the search pointer.
Level 1 (technology program): Program line with required geometry program call
Level 2 (geometry program): Program line for starting machining

If the geometry program is on the compact flash card, you must not only specify the program name in the "Program" input field on level 2 but also the path. The path for the compact flash card is always "C:\", i.e. you enter the following in the input field:
`C:\Program_name`

Select the accelerated calculation method "External – without calculation".
The block search is performed in the technology program with calculation. All EXTCALL commands before the required geometry program are skipped. The block search in the required geometry program is performed without calculation.

This calculation method assumes that all machine functions, such as tool call, machining feedrate, spindle speed, etc. are contained in the technology program. The geometry program must only contain geometry values for the free-form surface.

Complete program

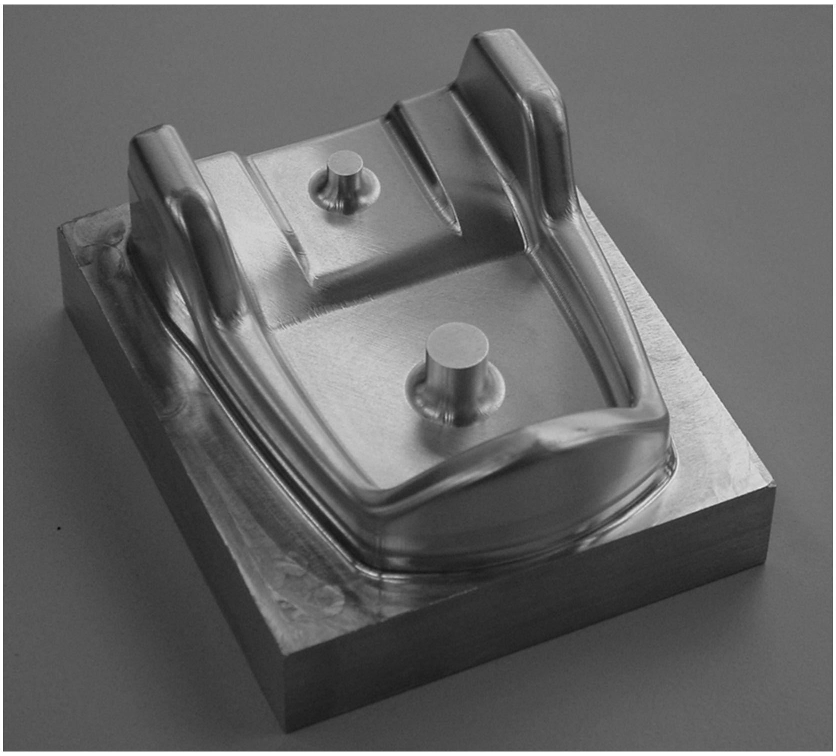
To start execution of a certain program section in a complete program, place the cursor directly on the required destination block (using "Search", if necessary).

When you then select a calculation method, consider:
The "External without calculation" method performs a search without considering the machine functions. That means execution of the program can only be started a points at which all relevant machine functions, such as feed, spindle speed, etc. are performed.
For safety reasons, you should therefore choose the "to contour" or "to end point" method. However, these calculation methods require more computation time.

7.5 Example

Workpiece

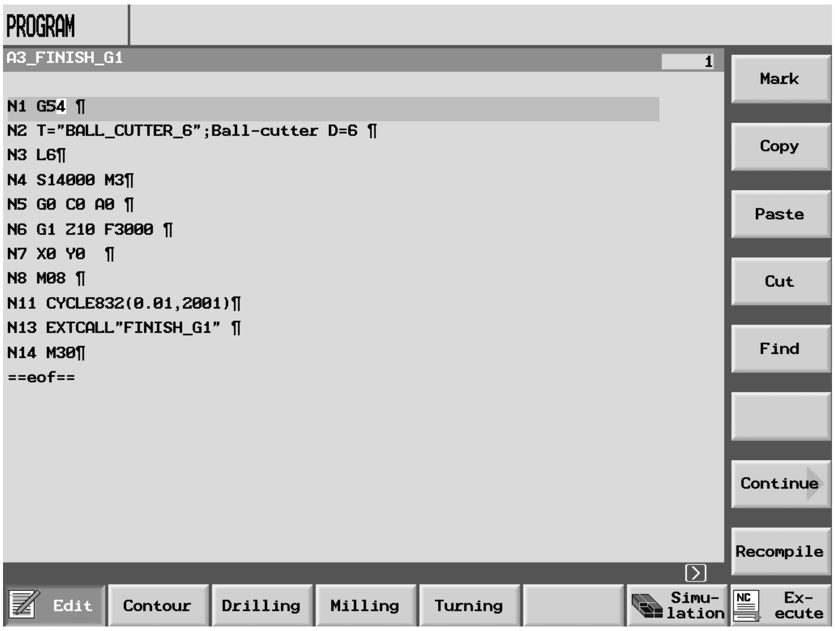
The task is to machine a mobile phone holder on a 3-axis machine.



Workpiece to be machined

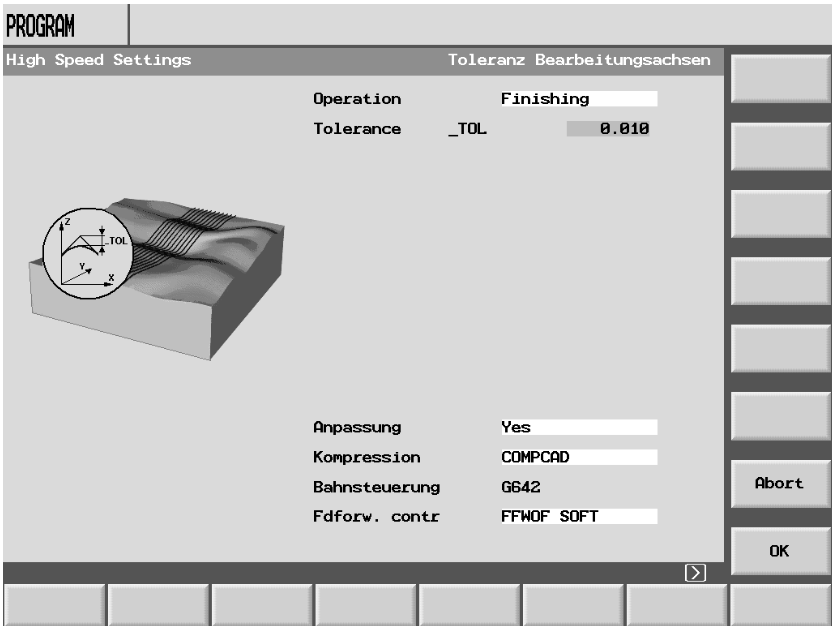
Program structure

The mold-making program is split into a technology program and a geometry program.

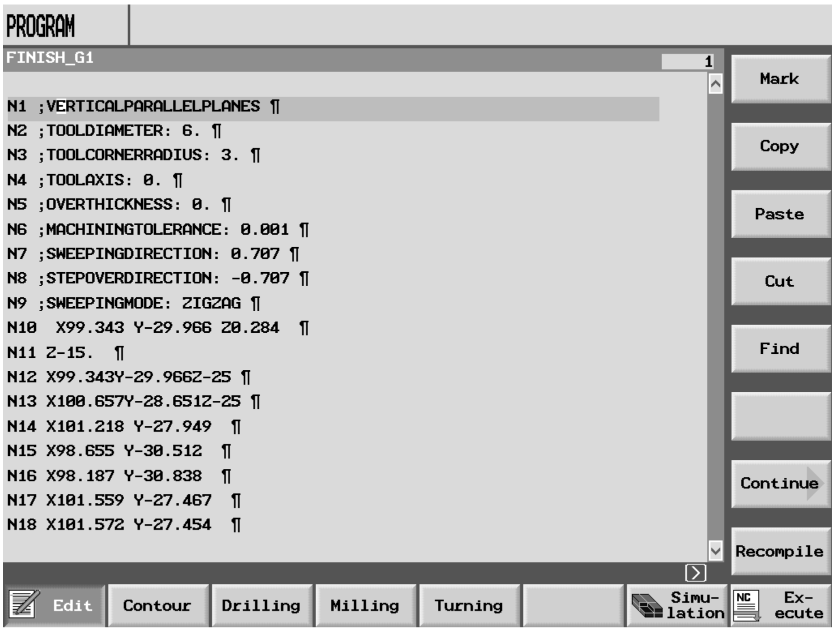


Technology program for the finishing operation

The "High Speed Settings" cycle is called in the technology program to achieve optimum velocity control.



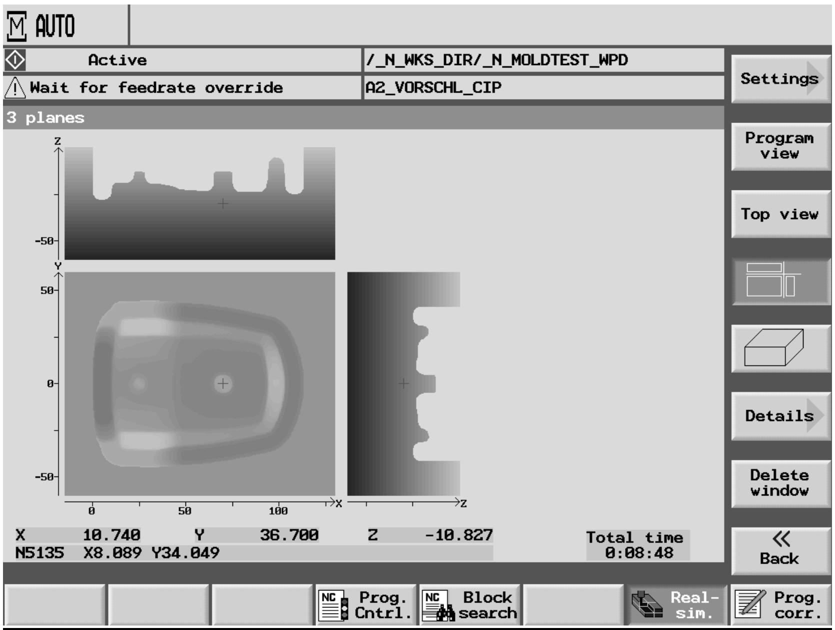
"High Speed Settings" cycle (CYCLE832)



Geometry program for the finishing operation

Simultaneous recording

During execution of the mold-making program, progress can be observed on screen.



Graphical display of the workpiece

■

Alarms and Messages

8.1 Cycle alarms and messages

8.1.1 Error handling in the cycles

If error conditions are detected in the cycles, an alarm is output and processing is aborted.
Alarms with numbers between 61000 and 62999 are output in the cycles.
The reset criteria for these number ranges are

- 61000 ... 61999 is NC-RESET
- 62000 ... 62999 is CANCEL

The text displayed with the alarm number provides an explanation of the cause of the error.

8.1.2 Overview of cycle alarms

The following table lists the alarms that might occur in the cycles with tips on how to remedy the errors that cause them.

Alarm number	Alarm text	Explanation, remedy
61000	"No tool offset active"	D-correction must be programmed before the cycle call
61001	"Thread pitch incorrectly defined"	Check parameters for thread size and check pitch information (contradict each other)
61002	"Machining type incorrectly defined"	The machining type parameter has been set to the wrong value and needs to be altered.
61003	"No feedrate programmed in cycle"	The parameter for feedrate has been incorrectly set and must be altered.
61006	"Tool radius too large"	Select smaller tool
61007	"Tool radius too small"	Select larger tool
61009	"Active tool number = 0"	Load the required tool
61010	"Finishing allowance too large"	Reduce the finishing allowance setting
61011	"Invalid scaling"	The active scaling factor is not permissible for this cycle.
61012	"Different scales in one plane"	Cycle execution only possible with uniform scaling in the plane
61013	"Basic settings have been altered, program cannot be executed"	Check and, if necessary, change the basic settings
61101	"Reference plane incorrectly defined"	Either different values must be entered for the reference plane and the retraction plane if they are relative values or an absolute value must be entered for the depth

61102	"No spindle direction programmed"	A spindle direction must be programmed
61103	"Number of holes is zero"	No value has been programmed for the number of holes
61104	"Contour violation of slots"	Errors in parameterization of milling pattern in those parameters which define the position of slots on a circle and their shape
61105	"Cutter radius too large"	The diameter of the milling cutter being used is too large for the figure that is to be machined; either a tool with a smaller radius must be used or the contour must be changed
61106	"Number or spacing of circle elements"	Parameterization error, programmed circle elements cannot be arranged around a full circle
61107	"First drilling depth incorrectly defined"	First drilling depth is inverted in relation to total drilling depth.
61108	"No valid settings for parameters _RAD1 and _DP1"	Parameters "Radius" and "Infeed depth per revolution" must be taken into account for insertion along helical path
61109	"Parameter _CDIR incorrectly defined"	Parameter defining milling direction is incorrectly defined
61110	"Finishing allowance on base > infeed depth"	Alter setting for depth infeed, if necessary.
61111	"Infeed width > tool diameter"	The programmed infeed width is greater than the diameter of the active tool. The infeed width must be reduced.
61112	"Tool radius negative"	The radius of the active tool is negative. This is illegal.
61113	"Parameter _CRAD for corner radius too large"	Reduce the parameter for corner radius
61114	"Direction of machining G41/G42 incorrectly defined"	Check the machining direction of tool radius compensation left/right and alter.
61115	"Approach or retract mode (line/circle/plane/ space) incorrectly defined"	An incorrect contour approach or retract mode was defined. Check the approach/retract mode or approach/retract strategy parameter.
61116	"Approach or retract path = 0"	The approach or retract path is set to zero, it must be increased.
61117	"Active tool radius <= 0"	The radius of the active tool is negative or zero. This is illegal.
61118	"Length or width = 0"	The length or width of the milling surface is illegal.
61119	"Nominal or core diameter incorrectly programmed"	Check thread geometry
61120	"Thread type internal, external not defined"	You must enter the internal, external thread type.

61121	"Number of teeth/cutting edge missing"	Enter the number of teeth/cutting edges for the active tool into the tool list.
61122	"Safety clearance in the plane incorrectly defined"	The safety clearance is negative or zero. This is illegal.
61124	"Infeed width is not programmed"	In active simulation without a tool, a value for the infeed width must always be programmed.
61125	"Technology selection in parameter _TECHNO incorrectly defined"	Check settings in machine data 9855 and 9856.
61126	"Thread length too short"	Check thread geometry.
61127	"Speed ratio of tapping axis incorrectly defined (machine data)"	Check settings in machine data 31050 and 31060.
61128	"Insertion angle = 0 for insertion via oscillation or helix"	Use larger insertion angle.
61180	"No name assigned to swivel data block although machine data $MN_MM_NUM_TOOL_CARRIER > 1"	Assign a unique name for the swivel data block.
61181	"NCK software version too old (no TOOLCARRIER functionality)"	Upgrade NCK software.
61182	"Name of swivel data record unknown"	Check the name of the swivel data block.
61183	"Retraction mode GUD7 _TC_FR outside value range 0..2"	Check installation and start-up of the swivel cycle CYCLE800.
61184	"No solution can be found with current angle inputs"	Check the angle entered for the swivelling of the machining plane.
61185	"No or incorrect (min > max) angle ranges declared for rotary axes"	Check installation and start-up of the swivel cycle CYCLE800.
61186	"Invalid rotary axis vectors"	Check installation and start-up of the swivel cycle CYCLE800.
61188	"No axis name declared for 1st rotary axis -> check CYCLE800 start-up"	Check installation and start-up of the swivel cycle CYCLE800.
61200	"Too many elements in machining block"	Revise machining block, if necessary deleting elements
61201	"Incorrect sequence in machining block"	Sort the machining block sequence.
61202	"Not a technology cycle"	Program technology block.

61203	"Not a position cycle"	Program positioning block.
61204	"Unknown technology cycle"	Delete and reprogram technology block.
61205	"Unknown position cycle"	Delete and reprogram positioning block.
61210	"Block search element not found"	Repeat block search.
61212	"Incorrect tool type"	Select a new tool type.
61213	"Circle radius too small"	Enter a larger value for the circle radius.
61214	"No pitch programmed"	The pitch must be programmed
61215	"Blank dimension incorrectly programmed"	Check dimensions of blank spigot. The blank spigot must be larger than the finished spigot.
61216	"Feed/tooth possible only for milling tools"	Alternatively, you can set another feed type
61217	"Cutting rate programmed for tool radius 0"	Enter a cutting rate setting.
61218	"Feed/tooth programmed, but number of teeth is zero"	Enter the number of teeth of the cutting tool in the "Tool list" menu.
61222	"Plane infeed greater than tool diameter"	Reduce plane infeed.
61223	"Approach path too small"	Enter a larger value for the approach path.
61224	"Retract path too small"	Enter a larger value for the retract path.
61225	"Swiveling data block unknown"	An attempt was made to access a swivel data block which has not been defined.
61226	"Swivel head cannot be replaced"	The parameter "Swivel data block" is set to "No". In spite of this, an attempt has been made to change the swivel head.
61230	"Tool probe diameter too small"	The tool probe is not correctly calibrated.
61231	"Sequential control program cannot be executed; not yet tested by ShopMill"	The program has to be simulated first in ShopMill or loaded into the operating mode "Machine auto" by ShopMill.
61232	"Magazine tool cannot be loaded"	Only manual tools may be loaded into a swivel head in which the tools can only be manually loaded.
61234	"ShopMill subroutine cannot be executed; not yet tested by ShopMill"	The subroutine has to be simulated first in ShopMill or loaded into the ShopMill operating mode "Machine auto".
61301	"Probe is not responding"	• Check probe connection • Set a longer measuring distance via MD 9752, 9753, 9754, 9755 • For edge measurements: Position probe closer to edge • For spigots/holes: Position roughly over the middle • Check value for spigot/hole diameter

61302	"Probe collision"	The measuring probe collided with an obstacle when being positioned. • Check spigot diameter (it may be too small) • Check measuring path (it may be too long)
61303	"Safe area exceeded"	Measuring result deviates greatly from specified value for the spigot/hole diameter. Check radius or diameter. Check measuring location (e.g. for inaccuracies caused by swarf)
61308	Check measuring path 2a	Measuring path = 0 entered Check MD 9752, 9753, 9754, 9755
61309	Check probe type	Probe type: 3D probe not active
61310	Scaling factor is active	Scaling factor = scaling is active
61311	No D number is active	No tool offset for the measuring probe (for workpiece measurement) or no tool offset for the active tool (for tool measurement) is selected.
61316	Center point and radius cannot be calculated	It is not possible to calculate a circle from the measured points.
61332	Alter the tool tip position	Tool tip is positioned below the probe surface (e.g. with a setting ring gauge or cube)
61338	Positioning speed is zero	Set corresponding feedrate (plane/infeed rate) via MD 9757 or 9758
61605	"Contour incorrectly programmed"	Check the contour.
61610	"No infeed depth programmed"	The infeed depth must be programmed
62100	"No drilling cycle active"	No modal drilling cycle has been called before the drilling pattern cycle.
62101	"Milling direction incorrect - G3 will be generated"	Climb or conventional milling programmed. However the spindle was not rotating when the cycle was called.
62103	"No finishing allowance programmed"	Program a finishing allowance.
61275	"Destination point violates software limit switch!"	Swiveling has placed the destination point outside the software limit switch. You may have to choose a different preferred direction for swiveling or place the retraction plane lower.
62180	"Set rotary axes ... "	Prompt to position rotary axes manually.

62181	"Set rotary axis ... "	Prompt to position rotary axis manually.
62182	"Attach inclinable head: ..."	Request to load a swivel head.
62183	"Remove inclinable head:..."	Prompt to remove inclinable head.
62184	"Replace inclinable head:..."	Prompt to replace inclinable head.
62185	"Angle adjusted to angular grid:..."	Indication that the desired angle cannot be set due to the Hirth tooth system. The displayed angle is set instead.

8.1.3 Messages in the cycles

The cycles output messages in the dialog line of the control. These messages do not interrupt execution.
They provide information about specific cycle behavior and how machining is progressing and are usually displayed for the duration of the machining operation or until the end of the cycle.

8.2 Alarms in ShopMill

8.2.1 Overview of alarms

If errors are detected in ShopMill, the system generates an alarm and aborts program execution, if necessary.
The text displayed with the alarm number provides an explanation of the cause of the error.

Overview of alarms

100000-100999	Basic system	
101000-101999	Diagnostics	
102000-102999	Services	
103000-103999	Machine	
104000-104999	Parameters	
105000-105999	Programming	
106000-106999	Spare	
107000-107999	OEM	
110000-110999		reserved
111000-112999	ShopMill	
120000-120999		reserved

Warning

Please check the situation in the plant on the basis of the description of the active alarm(s). Eliminate the cause/s of the alarm/s and acknowledge it/them as instructed. Failure to observe this warning will place your machine, workpiece, stored settings and possibly even your own safety at risk.

If you are working in CNC ISO mode, please refer to alarm descriptions in the following manual:
References: /DA/, Diagnostics Guide SINUMERIK 840D/840Di/810D

8.2.2 Selecting the alarm/message overview

Function

You can view alarms and messages and then acknowledge them.

Operating sequence

Alarm list or ALARM

The alarm/message overview displays all active alarms and messages with numbers, date, cancel criterion and explanation.
Clear the alarm by pressing the key that is displayed as a symbol:

Switch machine/control off and on again (main switch) or NCK Power ON

Reset

Press the "Reset" key

ALARM CANCEL

Press the "Alarm cancel" key

Cycle Start

Alarm is canceled with "Cycle Start"

Alarm is canceled with the "Return" key

8.2.3 Description of the alarms

112100	**Renumbering error. Initial state restored.**
Explanation	You have selected the "Renumber" softkey in the program editor. An error during renumbering has damaged the program in the memory. The initial program must now be reloaded to the memory.
Response	Alarm display Program has not been renumbered.
Remedy	Create space in the memory, e.g. by deleting an old program. Select "Renumber" softkey again.

112200	**Contour is step in current program sequence. Processing not enabled**
Explanation	The selected contour is an element of the program loaded under "Program".
Response	Alarm display The contour is an element from a loaded program and cannot be deleted or renamed.
Remedy	Remove contour from the loaded program.

112201	**Contour is step in current Automatic sequence. Processing not enabled**
Explanation	The selected contour is an element of the program loaded under "Machine Auto".
Response	Alarm display The contour is an element of a program loaded under "Machine Auto" and cannot be deleted or renamed. After program start, contours included in the current program cannot be altered under "Program" while the program is running.
Remedy	Stop program run and load program under "Program". Delete contour from program.

112210	**Tool axis cannot be reselected. Insufficient NC memory.**
Explanation	If you select another tool axis, you must generate a new NC program. You must save the old NC program first and then generate the new one. There is not sufficient NC memory available at this point to store the new program.
Response	Alarm display The new tool axis selection is not implemented.
Remedy	You must create free space in the NC memory corresponding to at least the space required by the new program (e.g. by deleting programs you no longer need).
112211	**System unable to process tool preselection. Insufficient NC memory.**
Explanation	Before a tool preselection can be processed, you must generate a new NC program. You must save the old NC program first and then generate the new one. There is not sufficient NC memory available at this point to store the new program.
Response	Alarm display The system does not process the preselected tool.
Remedy	You must create free space in the NC memory corresponding to at least the space required by the new program (e.g. by deleting programs you no longer need).
112300	**Tool management strategy 2 impossible. Magazine is not fully loaded**
Explanation	The magazine is not fully loaded with tools. In the magazine of tool management type 2, the number of tools specified in machine data 18082 has to be created.
Response	Power ON alarm
Remedy	Start-up: Set up correct selection of tools
112301	**Tool management strategy 2 impossible. Magazine is not sorted according to tool list**
Explanation	The magazine list is not sorted according to the tool list. The tools in the magazine of tool management concept 2 must be ordered according to T number.
Response	Power ON alarm
Remedy	Start-up: Define tools in magazine locations according to T number.

112320 **Detach manual tool: %n%1**
Explanation The operator is prompted to detach the specified manual tool.
Response Alarm display

Please refer to the machine manufacturer's instructions.

Remedy Detach manual tool.

Please refer to the machine manufacturer's instructions.

112321 **Attach manual tool: %n%1**
Explanation The operator is prompted to attach the specified manual tool.

Please refer to the machine manufacturer's instructions.

Response Alarm display

Please refer to the machine manufacturer's instructions.

Remedy Attach manual tool.

Please refer to the machine manufacturer's instructions.

112322 **Replace manual tool: %n%1 -> %2**
Explanation The operator is prompted to replace the specified manual tool with the new manual tool.
Response Alarm display

Please refer to the machine manufacturer's instructions.

Remedy Replace manual tool.

Please refer to the machine manufacturer's instructions.

112323 **Unload swivel head**
Explanation The operator is prompted to remove the specified swivel head from the spindle.
Response Alarm display

Please refer to the machine manufacturer's instructions.

Remedy Replace swivel head.

Please refer to the machine manufacturer's instructions.

112324 **Load swivel head**
Explanation The operator is prompted to load the specified swivel head into the spindle.
Response Alarm display

Please refer to the machine manufacturer's instructions.

Remedy Load swivel head.

Please refer to the machine manufacturer's instructions.

112325	**Replace swivel head**
Explanation	The operator is prompted to replace the specified swivel head in the spindle with the new swivel head.
Response	Alarm display
	Please refer to the machine manufacturer's instructions.
Remedy	Exchange swivel head.
	Please refer to the machine manufacturer's instructions.
112326	**Set swivel head**
Explanation	The operator is prompted to set the swivel head in accordance with the specified data.
Response	Alarm display
	Please refer to the machine manufacturer's instructions.
Remedy	Set swivel head.
	Please refer to the machine manufacturer's instructions.
112327	**Angle outside the permissible range**
Explanation	The programmed machining operation cannot be performed with the inclinable head.
Response	Alarm display
Remedy	Perform NC reset. Clamp the workpiece differently if appropriate.
112328	**Angle adapted to angle grid**
Explanation	Due to the angle grid, the swivel head could not be set exactly to the specified angle.
Response	Alarm display
Remedy	Machining can be continued with the specified values, but it will not correspond exactly to the programming.
112329	**Set swivel head/table**
Explanation	The operator is prompted to set the swivel head/table in accordance with the specified data.
Response	Alarm display
	Please refer to the machine manufacturer's instructions.
Remedy	Set swivel head/table.
	Please refer to the machine manufacturer's instructions.

112330	**Set swivel table**
Explanation	The operator is prompted to set the swivel table in accordance with the specified data.
Response	Alarm display
	Please refer to the machine manufacturer's instructions.
Remedy	Set swivel table.
	Please refer to the machine manufacturer's instructions.
112340	**Confirmation not possible because axes are not referenced!**
Explanation	User confirmation in Safety Integrated cannot be given until the reference point has been approached.
Response	Alarm display
Remedy	Approach reference point.
112350	**No swiveling data available**
Explanation	No swiveling data sets are available.
Response	Alarm display
Remedy	Set up the necessary swiveling data sets (see /FBSP/, ShopMill Description of Functions)
112360	**Step was not included in program chain because program is running**
Explanation	The program that you want to change is currently being executed in "Machine Auto" mode. You cannot change programs when they are being executed in "Machine Auto" mode.
Response	Alarm display
Remedy	End program execution in "Machine Auto" mode.
112400	**Does not exist in tool management**
Explanation	The tool specified in the program does not exist.
Response	Alarm display
Remedy	The tool must be created before the data backup.
112401	**Could not create tool**
Explanation	A tool could not be created when reading in tool data.
Response	Alarm display
Remedy	Check the tool management system.
112402	**Work offsets: Error when writing**
Explanation	Data was not written into the work offset.
Response	Alarm display
Remedy	Check work offset.

112420	**Error on inch/metric changeover! Check all data!**
Explanation	Data conversion not completed for inch/metric changeover.
Response	Alarm display NC Start disable
Remedy	Check the following data: • Display machine data: MD9655: $MM_CMM_CYC_PECKING_DIST MD9656: $MM_CMM_CYC_DRILL_RELEASE_DIST MD9658: $MM_CMM_CYC_MIN_COUNT_PO_TO_RAD MD9664: $MM_CMM_MAX_INP_FEED_P_MIN MD9665: $MM_CMM_MAX_INP_FEED_P_ROT MD9666: $MM_CMM_MAX_INP_FEED_P_TOOTH MD9670: $MM_CMM_START_RAD_CONTOUR_POCKET MD9752: $MM_CMM_MEASURING_DISTANCE MD9753: $MM_CMM_MEAS_DIST_MAN MD9754: $MM_CMM_MEAS_DIST_TOOL_LENGTH MD9755: $MM_CMM_MEAS_DIST_TOOL_RADIUS MD9756: $MM_CMM_MEASURING_FEED MD9757: $MM_CMM_FEED_WITH_COLL_CTRL MD9758: $MM_CMM_POS_FEED_WITH_COLL_CTRL MD9759: $MM_CMM_MAX_CIRC_SPEED_ROT_SP MD9761: $MM_CMM_MIN_FEED_ROT_SP MD9762: $MM_CMM_MEAS_TOL_ROT_SP MD9765: $MM_CMM_T_PROBE_DIAM_LENGTH_MEAS MD9766: $MM_CMM_T_PROBE_DIAM_RAD_MEAS MD9767: $MM_CMM_T_PROBE_DIST_RAD_MEAS MD10240: $MN_SCALING_SYSTEM_IS_METRIC MD20150 [12]: $MC_GCODE_RESET_VALUES • Tool data for various edges D: Length Z, radius R, wear lengths Z and R • Work offsets: Base offset Position in X, Y, Z and A, C (if available) Work offset • Settings in MANUAL operating mode: Retraction plane Safety clearance
Note	This alarm is only output on a hardware fault.

112500	**Error in the NC interpreter % module %1**
Explanation	The ShopMill program cannot be opened.
Response	Alarm displayed interpreter stopped
Remedy	Please note the error text and contact the Siemens A&D MC Hotline.

112502	**Not enough memory Program aborted in line %1**
Explanation	%1 = Line number Program contains too many program blocks
Response	Alarm display Program not loaded
Remedy	Modify program in operating area PROGRAMS CNC-ISO operator interface.
112503	**ShopMill XXXX**
Explanation	A system error has occurred.
Response	Alarm display
Remedy	Please note the error text and contact the Siemens A&D MC Hotline.
112504	**File does not exist or is incorrect: %1**
Explanation	%1 = Name of file/contour Program cannot interpret a program block with contour programming. Contour does not exist in directory.
Response	Alarm display NC Start disable
Remedy	Load contour in directory.
112505	**Error while trying to interpret contour %1**
Explanation	%1 = Name of contour Contour incorrect
Response	Alarm display NC Start disable
Remedy	Check machining sequence of contour
112506	**Maximum number of contour elements exceeded %1**
Explanation	%1 = Name of contour Max. permissible number of 50 contour elements exceeded during interpretation of machining sequence of a contour.
Response	Alarm display
Remedy	Check machining sequence of contour and change if necessary.
112541	**Program cannot be interpreted**
Explanation	The program cannot be interpreted as a sequential control program during loading, as the program header is missing.
Response	Alarm display NC Start disable
Remedy	-

112542 **GUD variable not available or too small in the field dimension: %1**

Explanation	The required GUD variable was not found during read or write access.
Response	Alarm display
Remedy	Include the right GUD variables.

112543 **Prog. was created with higher software version**

Explanation	The part program was created with a higher software version than the existing software version.
Response	Alarm display
Remedy	Delete the machining operation and, if necessary, program it elsewhere.

112544 **Program cannot be opened. It is already being edited.**

Explanation	The program is already opened in another editor (e.g. in HMI Advanced).
Response	Alarm display
Remedy	Close the program

112546 **Program cannot be opened. No read rights on file.**

Explanation	The file has no read rights for the current access level.
Response	Alarm display
Remedy	Set the read rights by means of keyswitch or password entry.

112550 **Sequential control programming not opened**

Explanation	The "sequential control programming" option has not been set.
Response	Alarm display The program is opened as G code.
Remedy	Buy option.

112604 **Connection to PLC broken**

Explanation	Acknowledgement to the PLC user program, that the connection with the PCU has been broken off.
Response	Alarm display ShopMill PLC is shut down
Remedy	Check PLC user program.

112605 **Asynchronous subroutine has not been executed**

Note	Input values could not be processed correctly by the NC.
Response	Alarm display
Remedy	Perform NC reset.

112611	**NC Start not possible: % Deselect SBL mode**
Explanation	A program was activated with block search, while at the same time single block was active.
Response	NC Start disable Interface signals are set Alarm display
Remedy	Deselect single block mode.

112650	**Unknown PLC error**
Explanation	An error unknown to the user interface has been output by the PLC.
Response	Alarm display NC Start disable
Remedy	Press Power ON, inform Siemens.

8.3 User data

User data are variables that are used internally both by ShopMill programs and G code programs. These user data can be displayed in a list.

The following types of variables are defined:

- Global User Data (GUD)
  Global user data is valid in all programs.
  The display of global user data (GUD) can be disabled via keyswitch or password.
- Local User Data (LUD)
  Local User Data is only valid in the program or subroutine in which it was defined.
  When executing the program, ShopMill displays the LUD between the current block and the end of the program. If you press the "Cycle Stop" key, the LUD list is updated. The values, however, are continuously updated.
- Global Program User Data (PUD)
  Global program user data is created from the local variables (LUD) defined in the main program.
  PUD is valid in all subroutines, where it can also be read and written.
  The local data is also displayed with the global program user data.
- Channel-specific user data
  Channel-specific user data is only applicable in one channel.

ShopMill does not display user data of type AXIS and FRAME.

For details of which variables ShopMill displays, please refer to the machine manufacturer's instructions.

Displaying user data

Tools WOs -or- OFFSET	➢ Press the "Tools WOs" softkey or the "Offset" key.
>	➢ Press the "Expansion" key.
DEF User data	➢ Press the "User data" softkey.
Global user data ... Program user data	➢ Activate one of the softkeys to choose the user data that you want to display.

GUD + or GUD –

➢ You can press the "GUD +" and "GUD -" softkeys to display the global and channel-specific user data from GUD 1 to GUD 9.

Searching for user data

Search

➢ Press the "Search" softkey.

➢ Enter the text string that you want to locate.
You can search for any character string.

Accept

➢ Press the "Accept" softkey.

The user data is displayed.

Find next

➢ Press the "Find next" softkey to continue the search, as necessary.

The next user data that contains the search string is displayed.

8.4 Version display

You can read the version of the ShopMill-PLC on the ShopMill boot screen.
The ShopMill and NCU versions are given in the CNC-ISO operator interface.

CNC ISO

➢ Switch to the CNC ISO operator interface.

Diagnosis | Service display

➢ Press the "Diagnosis" and "Service displays" softkeys.

Version | NCU version

➢ Press the "Version" and "NCU version" softkeys.

The NCU version is displayed at the top of the window that appears:
xx.yy.zz 810D or 840D

HMI version

➢ Press the "HMI version" softkey.

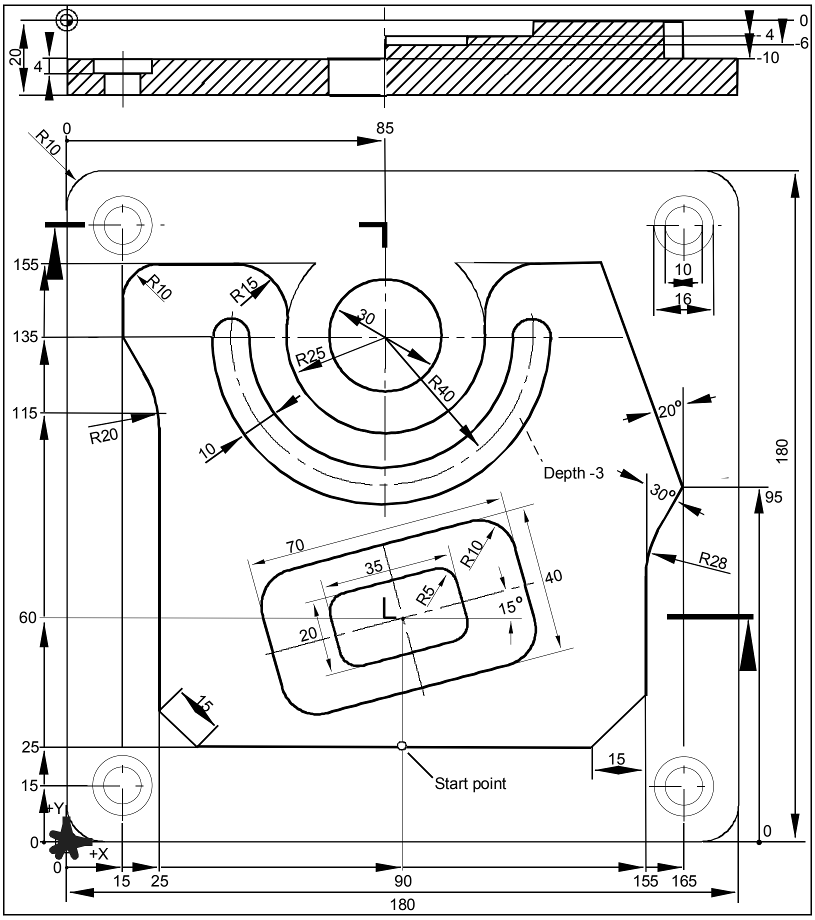
You can read the ShopMill version in the list that is displayed:
PCU 50: ShopMill........... V xx.yy.zz
PCU 20: cmm.dll............ V xx.yy.zz

■

Examples

9.1 Example 1: Machining with rectang./circ. pocket and circumf. slot

Workpiece drawing



Program Part_4

1. Program header

- Define the blank:

  X0 0 abs **Y0** 0 abs **Z0** 0 abs

  X1 180 abs **Y1** 180 abs **Z1** -20 abs

- Press the Accept softkey.

2. Face milling

- Select via the Mill- ing Face milling > softkeys and choose a machining strategy
- Example of technological data:

  T FACING TOOL **F** 0.1 mm/tooth **V** 1200 m/min

Machining	Roughing
X0	0 abs
Y0	0 abs
Z0	1 abs
X1	180 abs
Y1	180 abs

Z1	0 abs
DXY	80 %
DZ	0.5
UZ	0

- Press the Accept softkey.

3. Outside contour of workpiece

The outside contour can be defined as a rectangular spigot as shown **here**. It is, of course, also possible to use the contour milling function.

- Select via the Mill-ing / Pocket > / Rectang. pocket softkeys
- Assign technological parameters T, F and S accordingly and enter the following parameters:

Position of reference point	Bottom left
Machining	▽
Type of position	Single position
X0	0 abs
Y0	0 abs
Z0	0 abs
W	180 abs
L	180 abs
R	10 abs
α0	0 degrees
Z1	20 inc.
DZ	20
UXY	0
UZ	0
W1	185 (fictitious blank dimension)
L1	185 (fictitious blank dimension)

- Press the Accept softkey.

4. Outside contour of island

To machine the entire surface outside the island, define a contour pocket around the blank and then program the island. In this way, the entire surface area is machined and no residual material is left behind.

a) Outside contour of pocket

- Select via the Cont. mill. / New contour > softkeys
- Enter the contour name (here: Part_4_pocket) and confirm
- Fill out the start screen form for the contour

Tool axis Z
X -20 abs **Y** 0 abs

and confirm with Accept.

- Enter the following contour elements and confirm each one by pressing the Accept softkey:

1. ←•→ **X** 200 abs
2. ↕ **Y** 200 abs
3. ←•→ **X** -20 abs
4. Close contour

- Press the Accept softkey.

a) Outside contour of island

- Select via the Cont. mill. New contour > softkeys
- Enter the contour name (here: Part_4_Island) and confirm
- Fill out the start screen form for the contour

Tool axis Z

X 90 abs **Y** 25 abs

and confirm with Accept.

- Enter the following contour elements and confirm each one by pressing the Accept softkey:

1. ←•→ **X** 25 abs **FS** 15
2. ↕ **Y** 115 abs **R** 20
3. **X** 15 abs **Y** 135 abs
4. ↕ **Y** 155 abs **R** 10
5. ←•→ **X** 60 abs **R** 15
6. ↕ **Y** 135 abs **R** 20
7. Tangent to prec. elem. **Direction of rotation**
   R 25 **X** 110 abs
8. ↕ Tangent to prec. elem.
   Y 155 abs **R** 15
9. ←•→ **R** 0
10. **X** 165 abs **Y** 95 abs **α1** 290 deg. **R** 0
11. **X** 155 abs **α1** 240 deg. **R** 28
12. ↕ **FS** 0

13. **X** 140 abs **Y** 25 abs α**1** 225 deg. **R** 0

14. Close contour Accept

c) Mill/solid machine a contour

- Select via the Cont. mill. Solid > softkeys
- Assign technological parameters T, F and S accordingly (e.g. cutter diameter 10) and enter the following parameters:

Machining	▽
Z0	0 abs
Z1	10 inc.
DXY	4.5 mm
DZ	10
UXY	0 mm
UZ	0
Start point	Auto
Insertion	Center
FZ	0.1 mm/tooth
Retraction mode	Select the mode, e.g. to retraction plane

- Accept

Notes:

- When selecting the milling tool, please make sure that the tool diameter is large enough to cut the intended pocket. A message will be displayed if you make a mistake.
- If you want to finish cut the pocket, you must assign parameters UXY and UZ accordingly and add a second solid machining cycle for finishing.

5. Mill a rectangular pocket (large)

- Select via the Mill-ing Pocket > Rectang. pocket softkeys
- Example of technological data:
  T MILL10 **F** 0.1 mm/tooth **V** 200 m/min

Position of reference point	Center
Machining	▽
Type of position	Single position
X0	90 abs
Y0	60 abs
Z0	0 abs
W	40
L	70
R	10
α**0**	15
Z1	4 inc.
DXY	4.5 mm

DZ	4
UXY	0
UZ	0
Insertion	Helical
EP	2
ER	2
Remove stock	Complete machining

- Accept

6. Mill a rectangular pocket (small)

- Select via the Mill-ing | Pocket > | Rectang. pocket softkeys
- Enter parameters:

X0	90 abs
Y0	60 abs
Z0	-4 abs
W	20
L	35
R	5
α**0**	15
Z1	4 inc.
DXY	4.5 mm
DZ	2
UXY	0
UZ	0
Insertion	Oscillation
EW	10 degrees
Remove stock	Complete machining

- Accept

7. Mill a circumferential slot

- Select via the Mill-ing | Slot > | Circumfe-rential slot softkeys
- Example of technological data:

T MILL8	**F**0.5mm/tooth	**FZ** 0.02mm/tooth
V 150m/min		

Machining	▽
Full / pitch circle	Pitch circle
X0	85 abs
Y0	135 abs
Z0	0 abs
W	10
R	40
α**0**	180 degrees
α**1**	180 degrees

α2	0 degrees
N	1
Z1	3 inc.
DZ	3
UXY	0 mm

- Accept

8. Drill/center

- Select via the Drill-ing Centering > softkeys
- Assign technological parameters T, F and S accordingly and enter the following parameters:

Diameter/tip	Diameter
∅	16

- Accept

9. Drill/ream

- Select via the Drill-ing Drilling reaming > Drilling softkeys
- Assign technological parameters T, F and S accordingly (e.g. DRILL10) and enter the following parameters:

Diameter/tip	Tip
Z1	-25 abs
DT	0

- Accept

10. Positions

- Select via the Drill-ing Positions > softkeys
- Enter parameters:

	Rectangular
Z0	-10 abs
X0	15 abs
Y0	15 abs
X1	165 abs
Y1	15 abs

- Accept

11. Obstacle

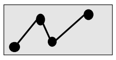
- Select via the Drill-ing Positions > Obstacle softkeys
- Enter parameters:

Z	2 abs

- Accept

Note:
If this obstacle cycle is not inserted, the drill will violate the right-hand corner of the island contour. Alternately, you could increase the safety clearance.

12.Positions

- Select via the [Drill-ing] [Positions >]  softkeys
- Enter parameters:

	Rectangular
Z0	-10 abs
X2	165 abs
Y2	165 abs
X3	15 abs
Y3	165 abs

- [Accept]

13.Mill a circular pocket

- Select via the [Mill-ing] [Pocket >] [Circular pocket] softkeys
- Example of technological data:

T MILL8 **F** 0.15 mm/tooth **V** 300 m/min

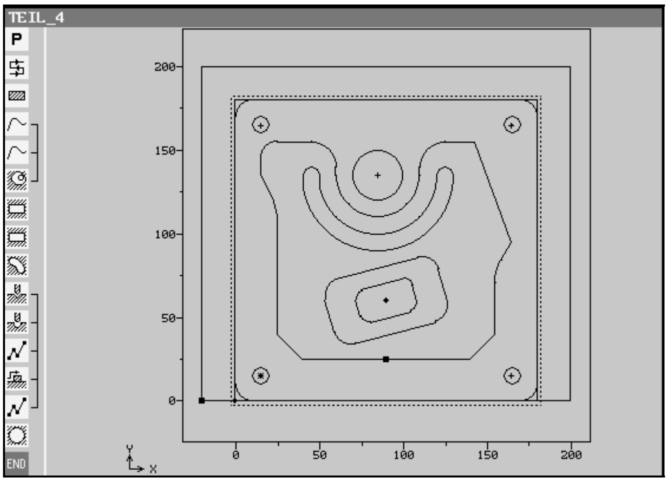
- Enter parameters:

Machining	▽
Type of position	Single position
X0	85 abs
Y0	135 abs
Z0	-6 abs
Diameter	30
Z1	15 inc.
DXY	4
DZ	5
UXY	0 mm
UZ	0
Insertion	Center
FZ	0.1 mm/tooth
Remove stock	Complete machining

- [Accept]

Result

- Programming graphics



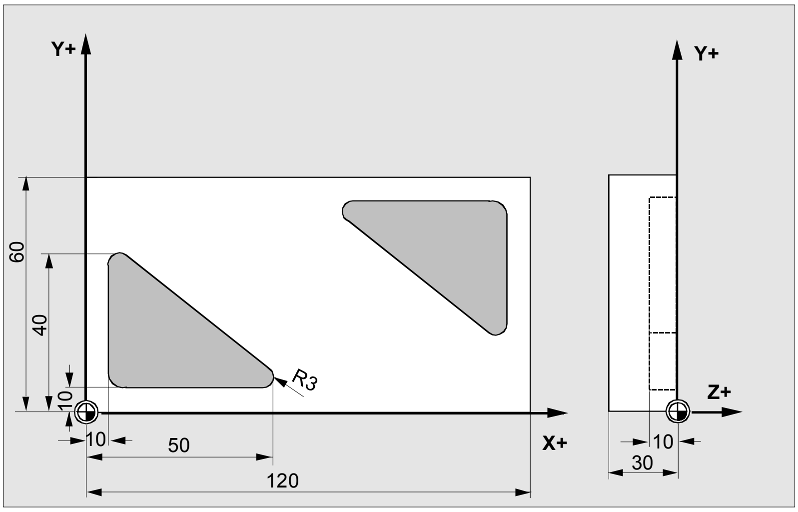
- ShopMill program representation

TEIL_4

P	N5 TEIL_4		
	N10 Face milling	▽	T=FRAESER60 F0.2/Z S400rev. X0=0 Y0=0
	N15 Rectang.spigot	▽	T=FRAESER60 F0.2/Z S500rev. X0=0 Y0=0
	N20 TEIL_4_TASCHE		
	N25 TEIL_4_INSEL		
	N30 Solid machin.	▽	T=FRAESER10 F0.2/Z S300rev. Z0=0
	N35 Rectang.pocket	▽	T=FRAESER10 F0.1/Z S200rev. X0=90 Y0=60
	N40 Rectang.pocket	▽	T=FRAESER10 F0.1/Z S200rev. X0=90 Y0=60
	N45 Circ.slot	▽	T=FRAESER8 F0.5/Z S150M X0=85 Y0=135
	N50 Centering		T=ZENTRIERER F300/min S300rev. ø16
	N55 DRILL		T=BOHRER10 F0.5/min S200M Z1=-25
	N60 001: Positions		Z0=-10 X0=15 Y0=15 X1=165 Y1=15
	N65 Obstacle		Z2
	N70 002: Positions		Z0=-10 X0=15 Y0=15 X1=165 Y1=15 X2=165
	N75 Circ. pocket	▽	T=FRAESER8 F0.15/Z S300M X0=85 Y0=135
END	N80 Program end		

9.2 Example 2: Translation and mirroring of a contour

Workshop drawing



In this example, the shapes shown recur in the same program. Both mirroring and translation operations are required. The shapes will be machined with a stock removal cycle.

Program Part_1

1. Program header
   - Define the blank:
     Corner point: **X0** 0 abs **Y0** 0 abs **Z0** 2 abs
     Dimensions: **L** 120 **W** 60 **H** -30
   - Press the Accept softkey.
2. Set start marker for repetition of the contour
   - Select via the Mics. Set marker > softkeys
   - Set start marking with "Marker1"
   - Accept
3. Define the contour
   - Select via the Cont. mill. New contour > softkeys
   - Enter the contour name (here: PART_1_3CORNER) and confirm
   - Fill out the start screen form for the contour
     Tool axis Z
     X 10 abs **Y** 10 abs
     and confirm with Accept.
   - Enter the following contour elements and confirm each one by pressing the Accept softkey:

1. X 60 abs R 3
2. X 10 abs Y 40 abs R 3
3. X 10 abs Y 10 abs R 3

- Press the Accept softkey.

4. Remove stock

- Select via the Cont. mill. | Remove stock > softkeys
- Assign technological parameters T, F and S accordingly (e.g. cutter diameter 3) and enter the following parameters:

Machining	▽
Z0	0 abs
Z1	10 inc.
DXY	1.5 mm
DZ	2
UXY	0.5
UZ	0.5
Start point	Auto
Insertion	Center
FZ	0.1 mm/tooth
Retraction mode	Select the mode, e.g. to retraction plane

- Accept

5. Set end marker for contour repetition

- Select via the Mics. | Set marker > softkeys
- Set end marking with "Marker2"
- Accept

6. Translate

- Select via the Mics. | Transfor- mations > | Offset > softkeys
- Set the following parameters:

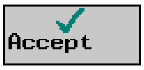
New/additive	New
X	120
Y	60
Z	0

- Accept

7. Mirror

- Select via the Mics. | Transfor- mations > | Mirror > softkeys
- Set the following parameters:

New/additive	Add
X	On
Y	On
Z	Off

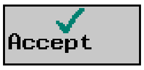
- 

8. Repetition of contour

- Select via the 

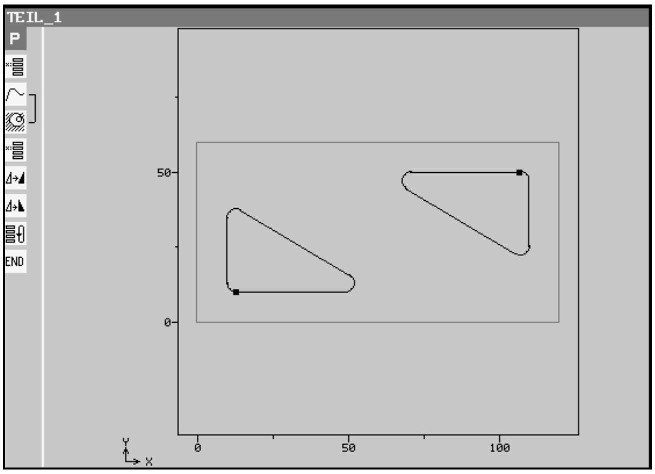
softkeys
- Set the following markers:

Start marker	Marker 1
End marker	Marker 2
Number of repetitions	1

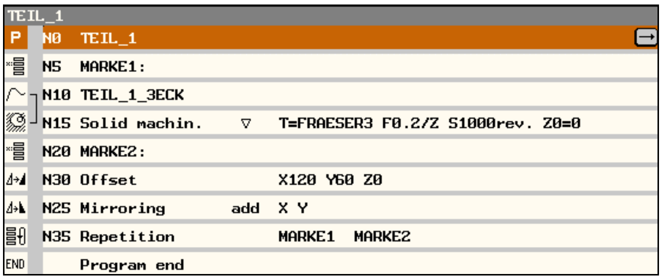
- 

Result

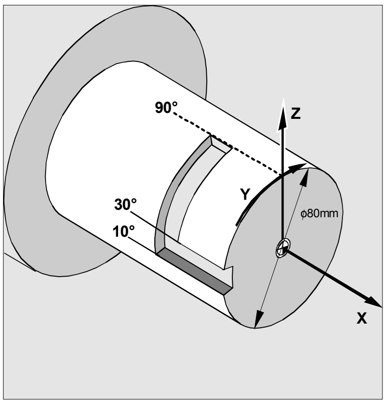
- Programming graphics

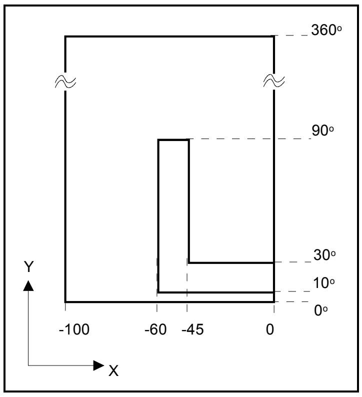


- ShopMill program representation



9.3 Example 3: Cylinder surface transformation





Requirements

- There is a rotary axis, e.g. axis A, and the transformation is configured via machine data.
- The reference points on the cylinder are predefined.
  Program the reference points X0, Y0, Z0 and the required work offset, for example, in "Machine Manual", "Workpiece zero" and "Edge". The work offset calculated from these is entered in the work offset list.

Program

1. Program header

- The blank dimensions correspond to the developed cylinder peripheral surface (L= ∅ x π).
  Define the blank:
  X0 0 abs **Y0** 0 abs **Z0** 40 abs
  X1 -100 abs **Y1** 251.327 abs **Z1** 20 abs **RP** 50
  Note: Y1 is calculated from diameter 80 multiplied by π (3.14...)
- Press the Accept softkey.

2. Activate the work offset in the program

Select work offset for cylinder surface transformation (e.g. offset the zero point on the center point of the cylinder end face).

- Select via the Mics. | Transfor-mations> | Work offset > softkeys
- Select the required work offset and then press the Accept softkey.

3. Position the Y axis

Position the tool in the Y axis over the center of the cylinder since the Y axis is not traversed after cylinder surface transformation is selected.

- Select via the Strai. Circle | Straight line softkeys
- Enter parameters:

X 10 abs **Y** 0 abs **Z** 50 abs **A** 0 abs
F *rapid traverse* mm/min **Radius compensation** off

- Press the Accept softkey.

4. Activate cylinder surface transformation
   - Select via the Mics. / Transfor-mations > / Cylinder surface > softkeys
   - Enter parameters:
     Transformation On
     ∅ 80
     Slot wall offset Off
   - Press the Accept softkey.

5. Activate the work offset in the program

   Define the work offset for the machining operation on the developed cylinder surface.
   - Select via the Mics. / Transfor-mations > / Work offset > softkeys
   - Select the required work offset and then press the Accept softkey.

6. Enter contour with contour calculator
   - Select via the Cont. mill. / New contour > softkeys
   - Enter the contour name and confirm
   - Fill out the contour start screen form
     Tool axis Z
     Cylinder surface yes
     ∅ 80
     X 0 **Yα** 10abs
     Note: Delete the **Y** value, then enter the **Yα** value (in this case 10°).
   - Enter the following contour elements and confirm each one by pressing the Accept softkey:
     1. **X** -60 abs
     2. **Yα** 90 abs
     3. **X** -45 abs
     4. **Yα** 30 abs
     5. **X** 0 abs
   - Press the Accept softkey.

7. Path milling

- Select via the "Cont. mill." "Path milling" softkeys
- Enter parameters

T CUTTER8 **F** 0.2 mm/tooth **S** 5000 rev/min

Radius compensation **Machining** ▽

Z0 40 abs **Z1** 10 inc. **DZ** 10

UZ0

UXY 0

Approach Straight line

Depth infeed

L1 2

FZ 0.1 mm/tooth

Retract Straight line

Retraction strategy

L2 2

Retraction mode To retraction plane

- Press the "Accept" softkey.

8. Deactivate cylinder surface transformation

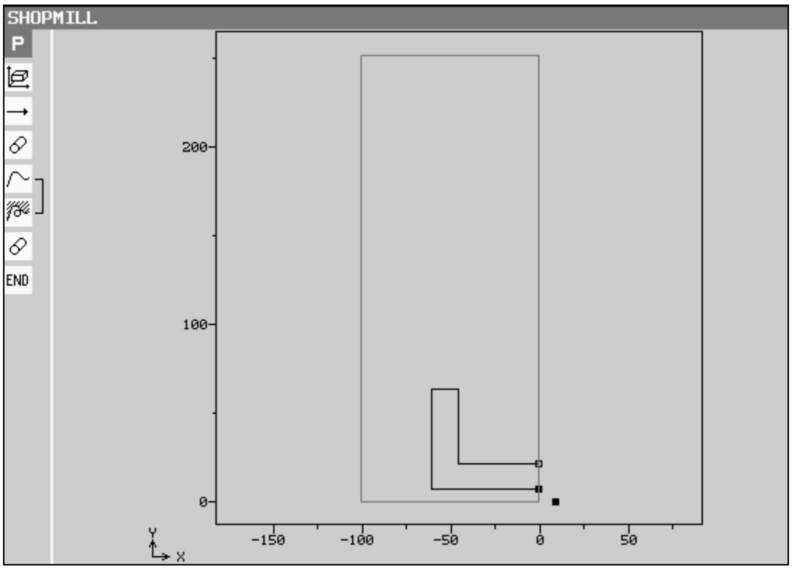
- Select via the "Mics." "Transfor-mations >" "Cylinder surface >" softkeys
- Enter parameters:

  Transformation Off

- Press the "Accept" softkey.

9. Result

- Programming graphics

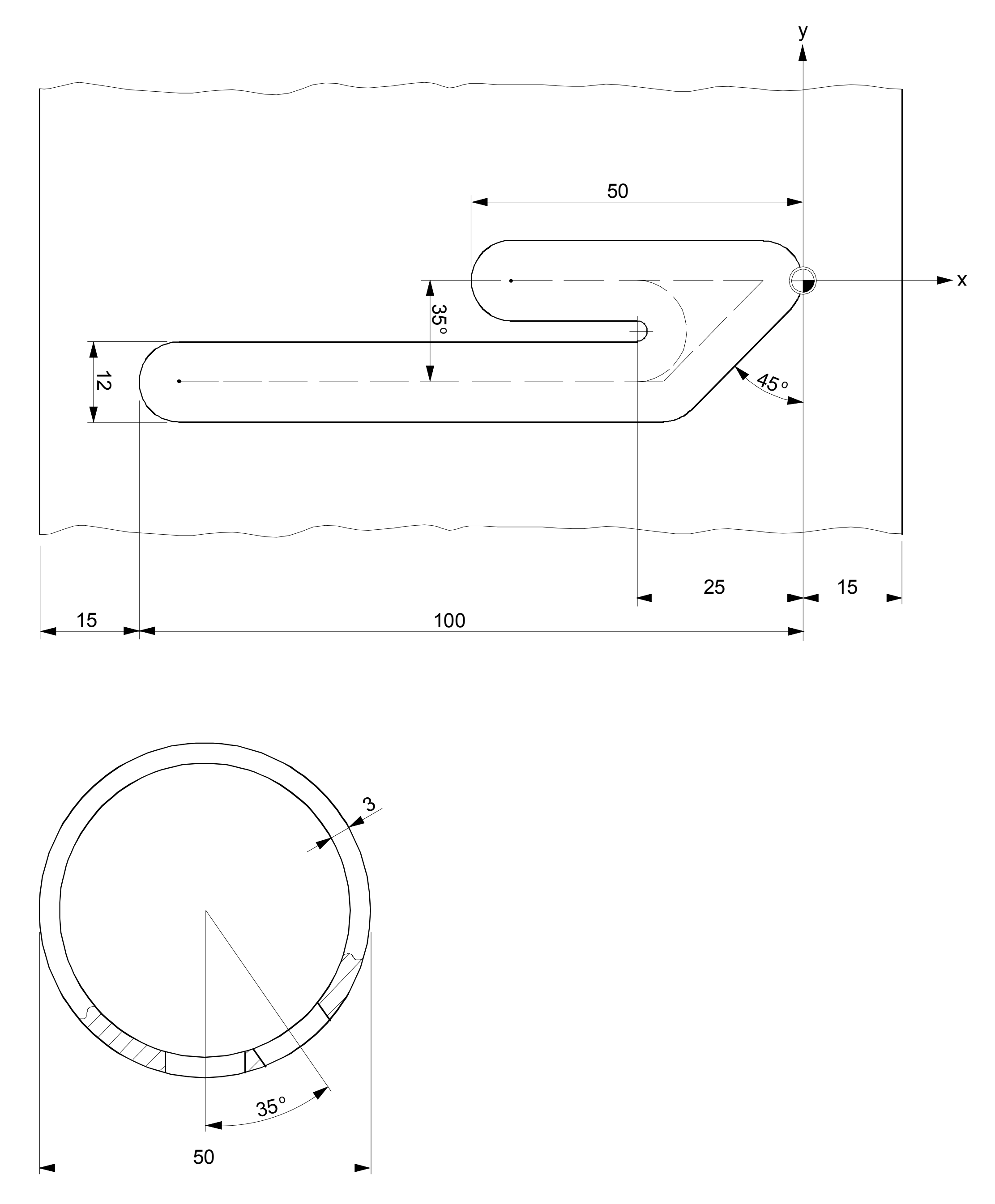


- ShopMill program representation

ZYLINDER	
P N5 ZYLINDER	
N10 Zero offset	1 G54
N15 RAPID X10 Y0 Z50	
N20 Cylind.surface	on None Groove wall compensation
N50 Zero offset	2 G55
N25 ZYLINDER_	
N30 Path milling ▽	T=CUTTER_8 F0.2/Z S5000rev. Z0=40
N35 Cylind.surface	off
END N40 Program end	

9.4 Example 4: Slot side compensation

A slot with parallel slot sides is milled in a pipe. In this instance, it is not the slot contour that is programmed, but the imaginary center-point path of a bolt inserted in the slot.



Requirements

- There is a rotary axis, e.g. axis A, and the transformation is configured via machine data.
- The reference points on the cylinder are predefined. Program the reference points X0, Y0, Z0 and the required work offset, for example, in "Machine Manual", "Workpiece zero", and "Edge".

The work offset calculated from these is entered in the work offset list.

Program

1. Program header

- The blank dimensions correspond to the developed cylinder peripheral surface.
  X0 0 abs **Y0** 0 abs **Z0** 25 abs
  X1 -130 abs **Y1** 157.08 abs **Z1** 22 abs
  RP 50 **SC** 1
  Note: Y1 is calculated according to equation: Y1 = $\varnothing \cdot \pi$
  In this case: Diameter 50 multiplied by 3.14...
- Press the "Accept" softkey.

2. Activate the work offset in the program

Select work offset for cylinder surface transformation (e.g. offset the zero point on the center point of the cylinder end face).

- Select via the "Mics." "Transfor-mations>" "Work offset >" softkeys
- Select the required work offset and then press the "Accept" softkey.

3. Position the Y axis

Position the tool in the Y axis over the center of the cylinder. The reason for this is that the Y axis is not traversed after cylinder surface transformation is selected.

- Select via the "Strai. Circle" "Straight line" softkeys
- Enter parameters:
  X 10 abs **Y** 0 abs **Z** 40 abs
  F *rapid traverse* mm/min **Radius compensation** off
- Press the "Accept" softkey.

4. Activate cylinder surface transformation

- Select via the "Mics." "Transfor-mations>" "Cylinder surface >" softkeys
- Enter parameters:
  Transformation On
  $\varnothing$ 50
  Slot wall offset On
  D 6
  Note: D is the distance from the imaginary center-point path to the slot wall.
- Press the "Accept" softkey.

5. Activate the work offset in the program

Define the work offset for the machining operation on the developed cylinder surface (shift zero point to the zero point on the workpiece drawing).

- Select via the "Mics." "Transfor-mations>" "Work offset >" softkeys
- Select the required work offset and then press the "Accept" softkey.

6. Enter contour with contour calculator

- Select via the "Cont. mill." "New contour" softkeys
- Enter the contour name (here: cylinder) and confirm
- Fill out the contour start screen form
  Tool axis Z
  Cylinder surface yes
  ∅ 50 **X** -25 abs **Yα** 0 abs
  Note: Delete the **Y** value, then enter the **Yα** value (in this case 0°).
- Press the "Accept" softkey.
- Enter the following contour elements and confirm each one by pressing the "Accept" softkey:

1. ←•→ **X** -44 abs
2. ←•→ **X** -25 abs
3. ↶↷ "All parameters" **Yα** -35 abs **I** 0 inc
   "Select dialog" (α2 tang.) "Accept dialog" **β2** 180°
4. ←•→
5. ←•→ **X** -94 abs
6. **X** -6 abs **Yα** 0 abs **α1** 45°
7. ←•→ **X** -25 abs

- Accept the contour by pressing the "Accept" softkey.

7. Path milling

- Select via the "Cont. mill." "Path milling" softkeys
- Enter parameters
  T CUTTER_8 **F** 0.2 mm/tooth **S** 5000 rev/min
  Radius compensation **Machining** ▽
  Z0 25 abs **Z1** 3 inc **DZ** 2
  UZ 0 **UXY** 0

Approach Quadrant
R1 1
FZ 0.1 mm/tooth

Retract Quadrant
R2 1
Retraction mode to retraction plane

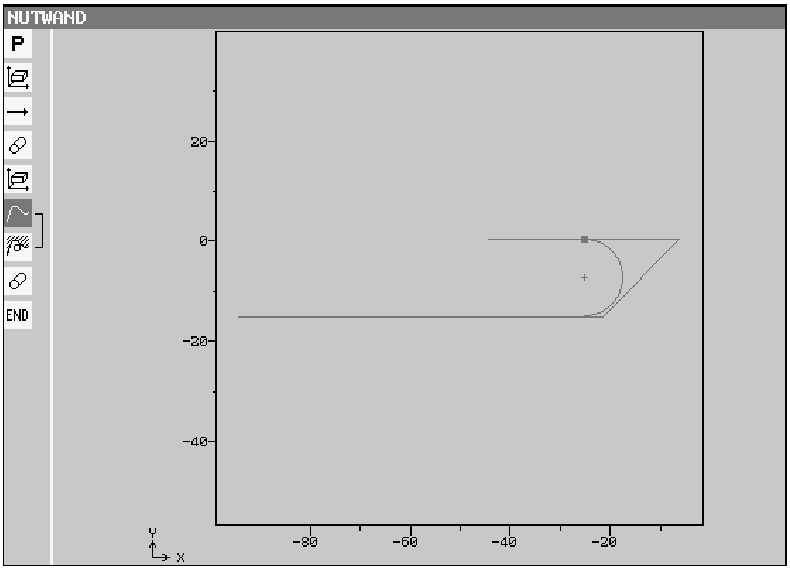
- Press the Accept softkey.

8. Deactivate cylinder surface transformation

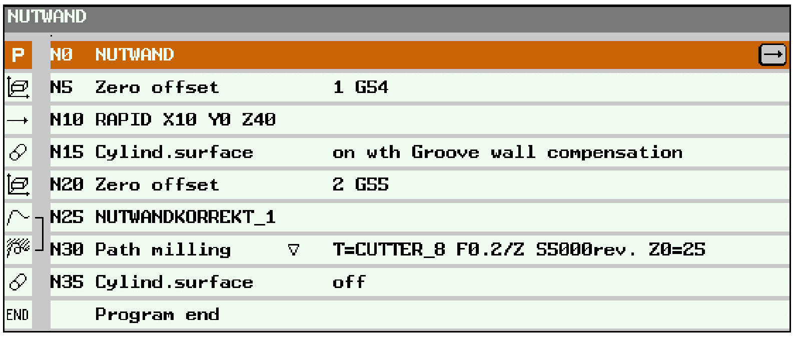
- Select via the Mics. / Transfor-mations > / Cylinder surface > softkeys
- Enter parameters:
Transformation Off
- Press the Accept softkey.

9. Result

- Programming graphics



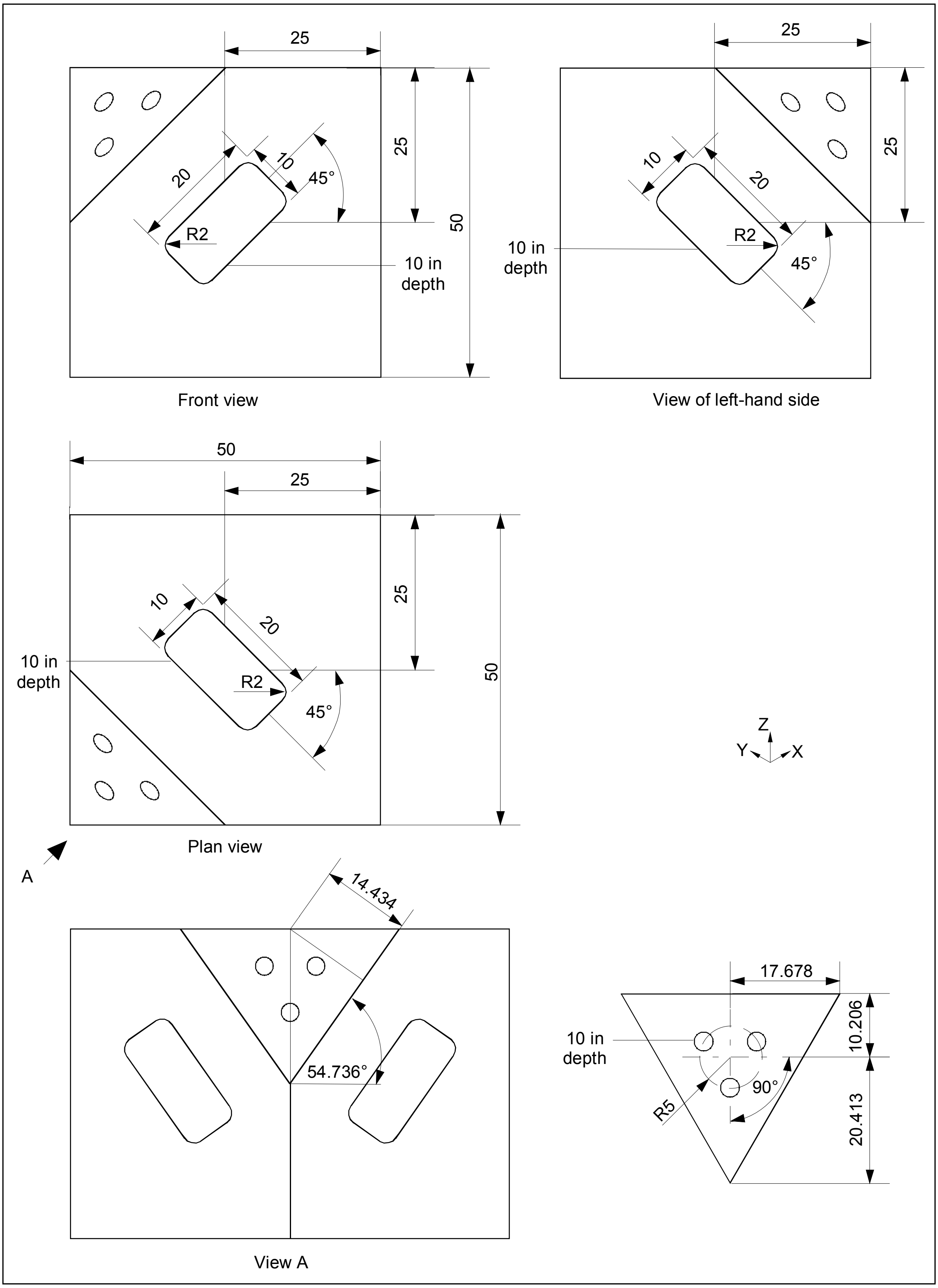
- ShopMill program representation

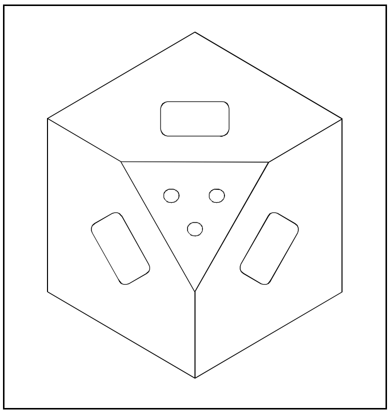


NUTWAND

P	N0	NUTWAND	
	N5	Zero offset	1 G54
	N10	RAPID X10 Y0 Z40	
	N15	Cylind.surface	on wth Groove wall compensation
	N20	Zero offset	2 G55
	N25	NUTWANDKORREKT_1	
	N30	Path milling ▽	T=CUTTER_8 F0.2/Z S5000rev. Z0=25
	N35	Cylind.surface	off
END		Program end	

9.5 Example 5: Swiveling





This example involves multiple swiveling of the machining plane.

Program example 4

1. Program header

- Define the blank:
  X0 0 abs **Y0** 0 abs **Z0** 0 abs
  X1 -50 abs **Y1** -50 abs **Z1** -50 abs
- Press the Accept softkey.

2. Rectangular pocket

- Select via the Mill-ing, Pocket >, Rectang. pocket softkeys
- Example of technological data:
  T MILL_4 **D** 1 **F** 0.1 mm/tooth **V** 200 m/min
- Enter the following parameters:

Position of reference point	Center
Machining type	Roughing
Type of position	Single position
X0	-25 abs
Y0	-25 abs
Z0	0 abs
W	10
L	20
R	2
α0	-45°
Z1	5 inc.
DXY	3 mm
DZ	2.5
UXY	0 mm
UZ	0
Insertion	Center
FZ	0.05 mm/tooth
Remove stock	Complete mach.

- Accept

3. Swiveling

- Select via the "Mics." "Transfor- mations >" "Swiveling >" softkeys
- Example of technological data:
  T MILL_4 **D** 1
- Enter the following parameters:

Retraction	Yes
Swiveling	Yes
Transformation	New
X0	0
Y0	-50
Z0	0
Swiveling	Axis by axis
X	90°
Y	0°
Z	0°
X1	0
Y1	0
Z1	0
Direction	-

- Accept

4. Rectangular pocket

- Select via the "Mill- ing" "Pocket >" "Rectang. pocket" softkeys
- Example of technological data:
  T MILL_4 **D** 1 **F** 0.1 mm/tooth **V** 200 m/min
- Enter the following parameters:

Position of reference point	Center
Machining type	Roughing
Type of position	Single position
X0	-25 abs
Y0	-25 abs
Z0	0 abs
W	10
L	20
R	2
α**0**	45°
Z1	5 inc.
DXY	3 mm
DZ	2.5
UXY	0 mm
UZ	0
Insertion	Center
FZ	0.05 mm/tooth
Remove stock	Complete mach.

- Accept

5. Swiveling

- Select via the Mics. | Transfor-mations > | Swiveling > softkeys
- Example of technological data:
  T MILL_4 **D** 1
- Enter the following parameters:

Retraction	Yes
Swiveling	Yes
Transformation	New
X0	-50
Y0	-50
Z0	0
Swiveling	Axis by axis
Z	-90°
X	90°
Y	0°
X1	0
Y1	0
Z1	0
Direction	-

- Accept

6. Rectangular pocket

- Select via the Mill-ing | Pocket > | Rectang. pocket softkeys
- Example of technological data:
  T MILL_4 **D** 1 **F** 0.1 mm/tooth **V** 200 m/min
- Enter the following parameters:

Position of reference point	Center
Machining type	Roughing
Type of position	Single position
X0	-25 abs
Y0	-25 abs
Z0	0 abs
W	10
L	20
R	2
α**0**	-45°
Z1	5 inc.
DXY	3 mm
DZ	2.5
UXY	0 mm
UZ	0
Insertion	Center
FZ	0.05 mm/tooth

Remove stock Complete mach.

- Accept

7. Setting

Define a different blank so that the visible section of the simulation shows the machining of the inclined plane:

- Select via the Mics. Settings softkeys
- Define the blank:

X0 -17.678 abs	**Y0** 10.206 abs	**Z0** 0 abs
X1 17.678 abs	**Y1** -20.413 abs	**Z1** -10 abs

- Press the Accept softkey.
- Accept

8. Swiveling

- Select via the Mics. Transfor-mations Swiveling > softkeys
- Example of technological data:
  T FACING TOOL **D** 1
- Enter the following parameters:

Retraction	Yes
Swiveling	Yes
Transformation	New
X0	-50
Y0	-50
Z0	-25
Swiveling	Axis by axis
Z	-45°
X	54.736°
Y	0°
X1	0
Y1	20.413
Z1	0
Direction	-

- Accept

9. Face milling

- Select via the Mill-ing Face milling > softkeys and choose a machining strategy
- Example of technological data:
  T FACING TOOL **D** 1 **F** 0.1 mm/tooth **V** 200 m/min
- Enter the following parameters:

Machining type	Roughing
X0	-17.678 abs

Y0	-20.413 abs
Z0	14.434 abs
X1	17.678 abs
Y1	10.206 abs
Z1	0 abs
DXY	80 %
DZ	2.5
UZ	0

- Accept

10. Boring

- Select via the Drill-ing / Drilling Reaming > / Drilling softkeys
- Example of technological data:
  T DRILL_3 **D** 1 **F** 0.1 mm/rev **S** 2000 rev/min
- Enter the following parameters:

Shank/tip	Shank
Z1	5 inc.
DT	0 s

- Accept

11. Position pattern

- Select via the Drill-ing / Positions > / softkeys
- Enter the following parameters:

Full circle/pitch circle	Full circle
Z0	0 abs
X0	0 abs
Y0	0 abs
α0	-90°
R	5
N	3
Positioning	Straight line

- Accept

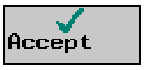
12. Swiveling

Return swivel head or swivel table to original position:

- Select via the Mics. / Transfor-mations / Swiveling > softkeys
- Example of technological data:
  T 0 **D** 1
- Enter the following parameters:

Retraction	Yes
Swiveling	Yes
Transformation	New
X0	0

Y0	0
Z0	0
Swiveling	Axis by axis
X	0°
Y	0°
Z	0°
X1	0
Y1	0
Z1	0
Direction	-

- 

Result

- ShopMill program representation

BEISPIEL4

P	N5 BEISPIEL4		
	N10 Rectang.pocket	▽	T=CUTTER_4 F0.1/Z V200M X0=-25 Y0=-25
	N15 Swivel		X90 Y0 Z0 T=CUTTER_4
	N20 Rectang.pocket	▽	T=CUTTER_4 F0.1/Z V200M X0=-25 Y0=-25
	N25 Swivel		Z-90 X90 Y0 T=CUTTER_4
	N30 Rectang.pocket	▽	T=CUTTER_4 F0.1/Z V200M X0=-25 Y0=-25
	N35 Setting		RP25 Blank
	N40 Swivel		Z-45 X54.736 Y0 T=CUTTER
	N45 Face milling	▽	T=CUTTER F0.1/Z V200M X0=-17.678
	N50 DRILL		T=DRILL F0.1/rev S2000rev. Z1=5inc
	N55 001: Hole full cir.		Z0=0 X0=0 Y0=0 R5 N3
	N60 Swivel		T=0
END	N65 Program end		

■

Appendix

A Abbreviations

ABS	Absolute dimensions
CNC	Computerized Numerical Control: Computerized numerical control
COM	Communication: Communication Component of NC control that performs and coordinates communication.
D	Cutting edge
DIN	Deutsche Industrie Norm (German Industry Standard)
DRF	Differential Resolver Function: Differential resolver function The function in conjunction with an electronic handwheel generates an incremental work offset in automatic mode.
DRY	Dry Run: Dry run feedrate
F	Feed
GUD	Global User Data: Global user data
HW	Hardware
INC	Increment
INC	Incremental dimensions
INI	Initializing Data: Initializing data
LED	Light Emitting Diode: Light emitting diode
M01	M function: Programmed stop
M17	M function: End of subprogram
MCS	Machine Coordinate System
MD	Machine data
MDA	Manual Data Automatic
MLFB	Machine-readable product designation
MPF	Main Program File: Main program

NC	Numerical Control: Numerical control The NC control comprises the components NCK, PLC, PCU and COM.
NCK	Numerical Control Kernel: Numerical control kernel Component of NC control that executes programs and basically coordinates movements for the machine tool.
OP	Operator Panel: Operator panel
PC	Personal Computer
PCU	Personal Computer Unit Component of NC control allowing communication between operator and machine.
PLC	Programmable Logic Control: Programmable logic control Component of NC control for processing machine tool control logic
PRT	Program Test
REF	Approaching a reference point
REPOS	Repositioning
ROV	Rapid override: Rapid override
RS-232-C	Serial interface
S	Spindle speed
SBL	Single Block: Single block
SI	Safety Integrated
SK	Softkey
SKP	SKiP: Skip block
SPF	Sub Program File: Subroutine
SW	Software
T	Tool
TMZ	Tool Magazine Zero
V	Cutting rate

WCS	Workpiece Coordinate System
WO	Zero offset
WPD	Workpiece Directory: Workpiece directory
WZ	Tool

B References

An overview of publications that is updated monthly is provided in a number of languages in the Internet at:

http://www.siemens.com/motioncontrol

via "Support" "Technical documentation" "Overview of publications"

General Documentation

/BU/ SINUMERIK & SIMODRIVE, Automation Systems for Machine Tools
Catalog NC 60

/IKPI/ Industrial Communication and Field Devices
Catalog IC PI

/ST7/ SIMATIC
Products for Totally Integrated Automation and Micro Automation
Catalog ST 70

/Z/ MOTION-CONNECT
Cable, Connectors & System Components for SIMATIC, SINUMERIK, MASTERDRIVES, and SIMOTION
Catalog NC Z

Safety Integrated
Application Manual
The Safety System for Industry

Electronic Documentation

/CD1/ The SINUMERIK System
DOC ON CD
(includes all SINUMERIK 840D/840Di/810D/802- and SIMODRIVE publications)

User Documentation

/AUK/ SINUMERIK 840D/810D
Short Guide **AutoTurn Operation**

/AUP/ SINUMERIK 840D/810D
Operator's Guide **AutoTurn Graphic Programming System**
Programming / Setup

/BA/ SINUMERIK 840D/810D
Operator's Guide **MMC**

/BAD/ SINUMERIK 840D/840Di/810D
Operator's Guide **HMI Advanced**

/BAH/ SINUMERIK 840D/840Di/810D
Operator's Guide **HT 6**

/BAK/ SINUMERIK 840D/840Di/810D
Short Guide Operation

/BAM/ SINUMERIK 810D/840D
Operation/Programming **ManualTurn**

/BAS/ SINUMERIK 840D/840Di/810D
Operation/Programming **ShopMill**

/BAT/ SINUMERIK 840D/810D
Operation/Programming **ShopTurn**

/BEM/ SINUMERIK 840D/810D
Operator's Guide **HMI Embedded**

/BNM/ SINUMERIK 840D/840Di/810D
User's Guide **Measuring Cycles**

/BTDI/ SINUMERIK 840D/840Di/810D
Motion Control Information System (MCIS)
User's Guide **Tool Data Information**

/CAD/ SINUMERIK 840D/840Di/810D
Operator's Guide **CAD Reader**
(part of the online help)

/DA/ SINUMERIK 840D/840Di/810D
Diagnostics Guide

/KAM/ SINUMERIK 840D/810D
Short Guide **ManualTurn**

/KAS/ SINUMERIK 840D/810D
Short Guide **ShopMill**

/KAT/ SINUMERIK 840D/810D
Short Guide **ShopTurn**

/PG/ SINUMERIK 840D/840Di/810D
Programming Guide **Fundamentals**

/PGA/ SINUMERIK 840D/840Di/810D
Programming Guide **Advanced**

/PGA1/ SINUMERIK 840D/840Di/810D
List Manual **System Variables**

/PGK/ SINUMERIK 840D/840Di/810D
Short Guide **Programming**

/PGM/ SINUMERIK 840D/840Di/810D
Programming Guide **ISO Milling**

/PGT/ SINUMERIK 840D/840Di/810D
Programming Guide **ISO Turning**

/PGZ/ SINUMERIK 840D/840Di/810D
Programming Guide **Cycles**

/PI/ PCIN 4.4
Software for Data Transfer to/from **MMC Modules**
Order number: 6FX2060-4AA00-4XB0 (English, German, French)
Order from: WK Fürth

/SYI/ SINUMERIK 840Di
System Overview

Manufacturer/Service Documentation

a) Lists

/LIS/ SINUMERIK 840D/840Di/810D
SIMODRIVE 611D
Lists

b) Hardware

/ASAL/ SIMODRIVE 611, MASTERDRIVES VC/MC
Planning Guide General Information for **Asynchronous Motors**

/APH2/ SIMODRIVE 611
Planning Guide **Asynchronous Motors 1PH2**

/APH4/ SIMODRIVE 611
Planning Guide **Asynchronous Motors 1PH4**

/APH7S/ SIMODRIVE 611
Planning Guide **Asynchronous Motors 1PH7**

/APH7M/ MASTERDRIVES MC
Planning Guide **Asynchronous Motors 1PH7**

/APL6/ MASTERDRIVES VC/MC
Planning Guide **Asynchronous Motors 1PL6**

/BH/ SINUMERIK 840D/840Di/810D
Operator Components Manual

/BHA/ SIMODRIVE Sensor
User Guide (HW) **Absolute Position Sensor with Profibus DP**

/EMV/ SINUMERIK, SIROTEC, SIMODRIVE, SIMOTION
Planning Guide **EMC Installation Guideline**

The up-to-date declaration of conformity can be viewed on the Internet at
http://www4.ad.siemens.de

Please enter the ID no.: 15257461 in the "Search" field (top right) and click "go".

/GHA/ SINUMERIK/SIMOTION
ADI4 - Analog Drive Interface for 4 Axes
Manual

/PFK6/ SIMODRIVE 611, MASTERDRIVES MC
Planning Guide **1FK6 Three-Phase AC Servomotors**

/PFK7/ SIMODRIVE 611, MASTERDRIVES MC
Planning Guide **1FK7 Three-Phase AC Servomotors**

/PFS6/ MASTERDRIVES MC
Planning Guide **1FS6 Three-Phase AC Servomotors**

/PFT5/ SIMODRIVE 611
Planning Guide **1FT5 Three-Phase AC Servomotors**

/PFT6/	SIMODRIVE 611, MASTERDRIVES MC Planning Guide **Synchronous Servomotors 1FT6**
/PFU/	SINAMICS, MASTERDRIVES MICROMASTER **SIEMOSYN Motors 1FU8**
/PHC/	SINUMERIK 810D **Configuring** Manual **(HW)**
/PHD/	SINUMERIK 840D **Configuring** Manual **(HW)**
/PJAL/	SIMODRIVE 611, MASTERDRIVES MC Planning Guide **Three-Phase Servomotors** **General Part for 1FT / 1FK Motors**
/PJAS/	SIMODRIVE 611, MASTERDRIVES VC/MC Planning Guide Asynchronous Motors Contents: **General Part, 1PH2, 1PH4, 1PH7, 1PL6**
/PJFE/	SIMODRIVE Planning Guide **1FE1 Built-In Synchronous Motors** Three-Phase AC Motors for Main Spindle Drives
/PJF1/	SIMODRIVE Installation Guide **1FE1 051.-1FE1 147**. **Built-In Synchronous Motors** AC Motors for Main Spindle Drives
/PJLM/	SIMODRIVE Planning Guide **1FN1, 1FN3 Linear Motors** ALL General Information about Linear Motor 1FN1 1FN1 Three-Phase AC Linear Motor 1FN3 1FN3 Three-Phase AC Linear Motor CON Connections
/PJM2/	SIMODRIVE 611, MASTERDRIVES MC Planning Guide **Servomotors** Contents: General Part, 1FT5, 1FT6, 1FK6, 1FK7, 1FS6
/PJTM/	SIMODRIVE Planning Guide **1FW6 Built-In Torque Motors 1FW6**
/PJU/	SIMODRIVE 611 Planning Guide **Converters**
/PKTM/	MASTERDRIVES Planning Guide **Torque Motors 1FW3**
/PMH/	SIMODRIVE Sensor Configuring/Installation Guide **Hollow-Shaft Measuring System SIMAG H**
/PMH2/	SIMODRIVE Sensor Configuring/Installation Guide **Hollow-Shaft Measuring System SIMAG H2**

/PMHS/ SIMODRIVE
Installation Guide **Measuring System for Main Spindle Drives**
SIZAG2 Toothed-Wheel Encoder

/PMS/ SIMODRIVE
Planning Guide
ECO Motor Spindle for Main Spindle Drives

/PPH/ SIMODRIVE
Planning Guide **1PH2 / 1PH4 / 1PH7 Motors**
AC Induction Motors for Main Spindle Drives

/PPM/ SIMODRIVE
Planning Guide
Hollow-Shaft Motors for **1PM4 and 1PM6** Main Spindle Drives

c) Software

/FB1/ SINUMERIK 840D/840Di/810D/FM-NC
Description of Functions **Basic Machine (Part 1)**
(the individual sections are listed below)

A2 Various Interface Signals
A3 Axis Monitoring, Protection Zones
B1 Continuous-Path Mode, Exact Stop and Look Ahead
B2 Acceleration
D1 Diagnostic Tools
D2 Interactive Programming
F1 Traverse to Fixed Stop
G2 Velocities, Setpoint/Actual Value Systems, Closed-Loop Control
H2 Output of Auxiliary Functions to PLC
K1 Mode Group, Channel, Program Operation Mode
K2 Axes, Coordinate Systems, Frames,
Actual-Value System for Workpiece, External Zero Offset
K4 Communication
N2 EMERGENCY STOP
P1 Traverse Axes
P3 Basic PLC Program
R1 Reference Point Approach
S1 Spindles
V1 Feeds
W1 Tool Offset

/FB2/ SINUMERIK 840D/840Di/810D
Description of Functions **Extended Functions (Part 2)**
including FM-NC: Turning, Stepper Motor
(the various manuals are listed below)

A4	Digital and Analog NCK I/Os
B3	Several Operator Panels and NCUs
B4	Operation via PG/PC
F3	Remote Diagnostics
H1	JOG with/without Handwheel
K3	Compensations
K5	Mode Groups, Channels, Axis Replacement
L1	FM NC Local Bus
M1	Kinematic Transformation
M5	Measurement
N3	Software Cams, Position Switching Signals
N4	Punching and Nibbling
P2	Positioning Axes
P5	Oscillation
R2	Rotary Axes
S3	Synchronous Spindle
S5	Synchronized Actions (SW 3 and lower, higher /FBSY/)
S6	Stepper Motor Control
S7	Memory Configuration
T1	Indexing Axes
W3	Tool Change
W4	Grinding

/FB3/ SINUMERIK 840D/840Di/810D
Description of Functions **Special Functions (Part 3)**
(the various sections are listed below)

F2	3-Axis to 5-Axis Transformation
G1	Gantry Axes
G3	Clock Times
K6	Contour Tunnel Monitoring
M3	Coupled Motion and Leading Value Coupling
S8	Constant Workpiece Speed for Centerless Grinding
T3	Tangential Control
TE0	Installation and Activation of Compile Cycles
TE1	Clearance Control
TE2	Analog Axes
TE3	Master-Slave for Drives
TE4	Transformation Package Handling
TE5	Setpoint Exchange
TE6	MCS Coupling
TE7	Retrace Support
TE8	Pulse-Independent Path-Synchronized Switching Signal Output
V2	Preprocessing
W5	3D Tool Radius Compensation

/FBA/ SIMODRIVE 611D/SINUMERIK 840D/810D
Description of Functions **Drive Functions**
(the individual sections are listed below)

DB1 Operating Messages/Alarm Reactions
DD1 Diagnostic Functions
DD2 Speed Control Loop
DE1 Extended Drive Functions
DF1 Enable Commands
DG1 Encoder Parameterization
DL1 Linear Motor MD
DM1 Calculating Motor/Power Section Parameters and Controller Data
DS1 Current Control Loop
DÜ1 Monitors/Limitations

/FBAN/ SINUMERIK 840D/SIMODRIVE 611 DIGITAL
Description of Functions **ANA MODULE**

/FBD/ SINUMERIK 840D
Description of Functions **Digitizing**

DI1 Start-Up
DI2 Scan with Tactile Sensor (scancad scan)
DI3 Scan with Laser (scancad laser)
DI4 Milling Program Generation (scancad mill)

/FBDM/ SINUMERIK 840D/840Di/810D
Description of Functions DNC **NC Program Management**
DNC Machines

/FBDN/ SINUMERIK 840D/840Di/810D
Motion Control Information System (MCIS)
Description of Functions **DNC NC Program Management**

DN1 DNC Plant / DNC Cell
DN2 DNC IFC SINUMERIK, NC Data Transfer via Network

/FBFA/ SINUMERIK 840D/840Di/810D
Description of Functions **ISO Dialects for SINUMERIK**

/FBFE/ SINUMERIK 840D/810D
Motion Control Information System (MCIS)
Description of Functions **Remote Diagnosis**

FE1 Remote Diagnosis ReachOut
FE3 RCS Host/RCS Viewer (pcAnywhere)

/FBH/ SINUMERIK 840D/840Di/810D
HMI Configuring Package
included with the software

Part 1 User's Guide
Part 2 Description of Functions

/FBH1/ SINUMERIK 840D/840Di/810D
HMI Configuring Package
ProTool/Pro Option SINUMERIK
included with the software

/FBHL/ SINUMERIK 840D/SIMODRIVE 611 digital
Description of Functions **HLA Module**

/FBIC/ SINUMERIK 840D/840Di/810D
Motion Control Information System (MCIS)
Description of Functions **TDI Ident Connection**

/FBMA/ SINUMERIK 840D/810D
Description of Functions **ManualTurn**

/FBO/ SINUMERIK 840D/810D
Description of Functions Configuring **OP 030 Operator Interface**
(the various sections are listed below)

BA Operator's Guide
EU Development Environment (Configuring Package)
PSE Introduction to Configuring of Operator Interface
(IK Screen Kit: Software Update and Configuration)

/FBP/ SINUMERIK 840D
Description of Functions **C-PLC Programming**

/FBR/ SINUMERIK 840D/840Di/810D
Description of Functions **RPC SINUMERIK Computer Link**

NFL Host Computer Interface
NPL PLC/NCK Interface

/FBSI/ SINUMERIK 840D/SIMODRIVE
Description of Functions SINUMERIK **Safety Integrated**

/FBSP SINUMERIK 840D/840Di/810D
Description of Functions **ShopMill**

/FBST/ SIMATIC
Description of Functions **FM STEPDRIVE/SIMOSTEP**

/FBSY/ SINUMERIK 840D/810D
Description of Functions **Synchronized Actions**

/FBT/ SINUMERIK 840D/810D
Description of Functions **ShopTurn**

/FBTC/ SINUMERIK 840D/810D
IT Solutions
Description of Functions **Tool Data Communication SinTDC**

/FBTD/ SINUMERIK 840D/810D
IT solutions
Description of Functions **Tool Information System (SinTDI)**
with Online Help

/FBTP/ SINUMERIK 840D/840Di/810D
Motion Control Information System (MCIS)
Description of Functions **Preventive Maintenance TPM**

/FBU/ SIMODRIVE 611 universal/universal E
Description of Functions **Closed-Loop Control Component**
for Speed Control and Positioning

/FBU2/ SIMODRIVE 611 **universal**
Installation Guide
(enclosed with SIMODRIVE 611 universal)

/FBW/ SINUMERIK 840D/810D
Description of Functions **Tool Management**

/HBA/ SINUMERIK 840D/840Di/810D
Manual **@Event**

/HBI/ SINUMERIK 840Di
Manual **SINUMERIK 840Di**

/INC/ SINUMERIK 840D/840Di/810D
System Description **Commissioning Tool SINUMERIK SinuCOM NC**
part of the Online Help for the Startup Tool

/PJE/ SINUMERIK 840D/810D
Description of Functions **Configuring Package HMI Embedded**
Software Update, Configuration, Installation

/PS/ SINUMERIK 840D/810D
Planning Guide **Configuring Syntax**
This document is supplied with the software and is available as a PDF.

/POS1/ SIMODRIVE **POSMO A**
User Guide's **Distributed Positioning Motor on PROFIBUS DP**

/POS2/ SIMODRIVE **POSMO A**
Installation Guide (included in every POSMO A)

/POS3/ SIMODRIVE POSMO SI/CD/CA
User Guide's **Distributed Servo Drive Systems**

/POS4/ SIMODRIVE **POSMO SI**

/POS5/ SIMODRIVE **POSMO CD/CA**
Installation Guide (included in every POSMO CD/CA)

/S7H/ SIMATIC S7-300
Installation Manual **Technological Functions**
Order number: 6ES7398-8AA03-8BA0
- Reference Manual: CPU Data (HW Description)
- Reference Manual: Module Data

/S7HT/ SIMATIC S7-300
Manual **STEP 7, Fundamentals, V. 3.1**

/S7HR/ SIMATIC S7-300
Manual **STEP 7, Reference Manuals, V. 3.1**

/S7S/ SIMATIC S7-300
FM 353 Positioning Module **for Stepper Drive**

/S7L/ SIMATIC S7-300
Positioning Module **FM 354 for Servo Drive**
Order with the configuring package

/S7M/ SIMATIC S7-300
FM 357.2 Multimodule for Servo and Stepper Drives
Order with the configuring package

/SP/ SIMODRIVE 611-A/611-D
SimoPro 3.1
Program for Configuring Machine-Tool Drives

d) Installation and Start-up

/BS/ SIMODRIVE 611 analog
Description **Start-Up Software for Main Spindle and Asynchronous Motor Modules Version 3.20**

/IAA/ SIMODRIVE 611A
Installation and Start-Up Guide

/IAC/ SINUMERIK 810D
Installation and Start-Up Guide
(including description of SIMODRIVE 611D start-up software)

/IAD/ SINUMERIK 840D/SIMODRIVE 611D
Installation & Start-Up Guide
(including description of SIMODRIVE 611D start-up software)

/IAM/ SINUMERIK 840D/840Di/810D
Installation and Start-Up Guide **HMI/MMC**

AE1 Updates/Supplements
BE1 Expanding the Operator Interface
HE1 Online Help
IM2 Starting Up HMI Embedded
IM4 Starting Up HMI Advanced
TX1 Creating Foreign Language Texts with Windows 95 / NT
TX2 Creating Foreign Language Texts with Windows 2000 / XP

C Index

To SIEMENS AG A&D MC BMS P.O. Box 3180 D-91050 Erlangen, Germany Phone: +49 (0) 180 5050-222 [Hotline] Fax: +49 (0) 9131 98-2176 [Documentation] E-mail: motioncontrol.docu@erlf.siemens.de	Suggestions Corrections For Publication/Manual: SINUMERIK 840D/840Di/810D ShopMill User Documentation
From Name Company/Dept. Address: Zip code: City: Phone: / Fax: /	Operation/Programming Order No.: 6FC5298-6AD10-0BP3 10.04 Edition Should you come across any printing errors when reading this publication, please notify us on this sheet. Suggestions for improvement are also welcome.

Suggestions and/or corrections

Overview of SINUMERIK 840D/840Di/810D Documentation (10.2004)

General Documentation

- *SINUMERIK* 840D/810D — Brochure
- *SINUMERIK* 840D/840Di/810D/ — Catalog Ordering Info. NC 60 *)
- *Safety Integrated* Application Manual — **Safety Integrated** Application Manual

User Documentation

- *SINUMERIK* 840D/810D/FM-NC — **AutoTurn**
  – Short Guide
  – Programming/Setup
- *SINUMERIK* 840D/840Di/810D — Operator's Guide **– HT 6**
- *SINUMERIK* 840D/840Di/810D — **Diagnostics Guide *)**
- *SINUMERIK* 840D/840Di/810D — **Operator's Guide *)**
  – Short Guide
  – HMI Embedded
  – HMI Advanced

User Documentation

- *SINUMERIK* 840D/840Di/810D — **Programming Guide**
  – Short Guide
  – Fundamentals *)
  – Advanced *)
  – Cycles
  – Measuring Cycles
  – ISO Turning/Milling
  – Lists **System Variables**
- *SINUMERIK* 840D/810D — Operator's Guide
  – ManualTurn
  – Short Guide ManualTurn
  – ShopMill
  – Short Guide ShopMill
  – ShopTurn
  – Short Guide ShopTurn
- *SINUMERIK* 840Di — **System Overview**

Manufacturer/Service Documentation

- *SINUMERIK* — **Configuring** (HW) *)
  – 810D
  – 840D
- *SINUMERIK* 840D/840Di/810D — **Operator Components** (HW) *)
- *SINUMERIK* 840D/840Di/810D — Description of Functions
  – ManualTurn
  – ShopMill
  – ShopTurn

Manufacturer/Service Documentation

- *SINUMERIK* 611D 840D/810D — Description of Functions **Synchronized Actions**
- *SINUMERIK SIMODRIVE* 840D/840Di/810D — Description of Functions **Drive Functions *)**
- *SINUMERIK* 840D/840Di/810D — **Description of Functions**
  – Basic Machine *)
  – Extended Functions
  – Special Functions
- *SINUMERIK* 840D/810D — Description of Functions **Tool Management**
- *SINUMERIK* 840D/810D — **Configuring Kit** HMI Embedded
- *SINUMERIK* 840D/810D — Description of Functions Operator Interface **OP 030**
- *SINUMERIK* 840D/810D — **MCIS**
  – Computer Link
  – Tool Data Inform. Syst.
  – NC Data Management
  – NC Data Transfer
  – Tool Data Communication

Manufacturer/Service Documentation

- *SINUMERIK SIMODRIVE* — Description of Functions SINUMERIK **Safety Integrated**
- *SINUMERIK* 840D — Description of Functions **Digitizing**
- *SINUMERIK SIMODRIVE* — **Installation & Start-Up Guide *)**
  – 810D
  – 840D/611D
  – HMI
- *SINUMERIK SIMODRIVE* 840D/840Di 810D 611D — **– Lists *)**
- *SINUMERIK SIMODRIVE* 840D 611D — Description of Functions **Linear Motor**
- *SINUMERIK SIMODRIVE* 840D 611D — Description of Functions
  – Hydraulics Module
  – Analog Module
- *MOTION CONTROL SYSTEMS* — **EMC** Guidelines

Electronic Documentation

- *SINUMERIK SIMODRIVE* 840D/840Di/810D 611, Motors — **DOC ON CD *)** The SINUMERIK System

Manufacturer/Service Documentation

- *SINUMERIK* 840D/840Di/810D — Description of Functions **ISO Dialects** for SINUMERIK
- *SINUMERIK* 840Di — **Manual** (HW + Installation and Start-Up)
- *SINUMERIK* 840D/810D — Description of Functions **Remote Diagnosis**
- *SINUMERIK* 840D/840Di/810D — **Manual @ Event**

*) These documents are a minimum requirement

Siemens AG
Automation & Drives
Motion Control Systems
P.O. Box 3180, D – 91050 Erlangen
Germany

www.siemens.com/motioncontrol

Order No. 6FC5 298-6AD10-0BP3

Printed in Germany