SIEMENS

User Guide

My SINUMERIK Operate

Contents

1 Introduction 7

2 SINUMERIK Operate 9

2.1 Uniform user interface for turning and milling 10
2.2 Help screens, animated elements, graphic view, tooltips, help 14
2.3 TSM mode/zero offsets 16
2.4 Programming 17
2.5 Tool management 21
2.6 Zero offset 24
2.7 Program management 25
2.8 Workpiece visualization (simulation and mold making view) 29
2.9 CNC operation in Automatic mode (AUTO) 31
2.10 Collision Avoidance 34

3 Turning technology 37

3.1 Setting up 38
3.1.1 Setting the zero point 38
3.1.2 Workpiece zero 39
3.1.3 Tool measurement 39
3.2 Manual Machine 42
3.3 Programming 44
3.3.1 ShopTurn 44
3.3.2 programGUIDE 46
3.4 Drilling 48
3.4.1 Overview of drilling cycles 48
3.4.2 Drilling centered/off-centered with ShopTurn 51
3.4.3 Drilling with programGUIDE 53
3.5 Turning 55
3.5.1 Overview of the turning cycles 55

3.6 Contour turning 58
3.6.1 Overview of the turning cycles for the contour machining 58
3.6.2 Contour turning with ShopTurn machining step programming 59
3.6.3 Contour turning with programGUIDE 61
3.7 Milling 66
3.7.1 Overview of the milling cycles 66
3.8 Contour milling 69
3.8.1 Overview of the milling cycles for the contour machining 69
3.8.2 Contour milling with ShopTurn machining step programming 71
3.8.3 Contour milling with programGUIDE 73
3.9 Counterspindle 76
3.10 Measuring in the Automatic mode - in-process measurement 77
3.10.1 Measure workpiece 77
3.10.2 Example 78

4 Milling technology 81

4.1 Setting up 82
4.1.1 Setting the zero point 82
4.1.2 Workpiece zero 83
4.1.3 Tool measurement 84
4.1.4 Face milling 85
4.1.5 Swiveling in JOG 86
4.2 Manual Machine 87
4.3 Programming 88
4.3.1 ShopMill machining step programming 88
4.3.2 programGUIDE 90
4.4 Drilling cycles 92
4.5 Milling cycles 95
4.6 Contour milling 99
4.6.1 Overview of the milling cycles for the contour machining 99
4.6.2 Contour milling with ShopMill machining step programming 101
4.6.3 Contour milling with programGUIDE 103
4.7 Miscellaneous 106
4.8 Measuring in the Automatic mode – in-process measurement 111
4.8.1 Measure workpiece 111
4.8.2 Measure tool 113

5 Multitasking (with SINUMERIK 840D sl) 115

5.1 Tool management 116
5.2 Turning-milling technology with ShopTurn and programGUIDE 117
5.2.1 Swivel axis in TSM mode 117
5.2.2 Turning with ShopTurn and B axis 117
5.2.3 Turning with programGUIDE and B axis 118
5.2.4 Milling with ShopTurn and B axis 118
5.2.5 Milling with programGUIDE and B axis 120
5.3 Milling-turning technology with programGUIDE 122
5.3.1 Align turning tool 122
5.3.2 Overview of the turning cycles in the milling technology 123
5.3.3 Contour turning in the milling technology 126
5.3.3.1 Overview of the contour turning cycles in programGUIDE 126
5.3.3.2 Contour turning with programGUIDE in the milling technology ... 128
5.4 Multi-channel machining 132
5.4.1 Machine basic screen 132
5.4.2 programSync multi-channel 133
5.4.3 Multi-channel program data 134
5.4.4 Dual editor 134
5.4.5 Time synchronization 135
5.4.6 Synchronous view 136
5.4.7 Simulation 136
5.4.8 Determining the machining time, optimization 137
5.4.9 Program control 138

6 Appendix 141

6.1 G-Code 141
6.2 Shortcuts 142
6.3 Further information 145

7 Index 147

1 Introduction

The SINUMERIK Operate user interface provides a clear and intuitive user and programming interface. This provides a uniform look & feel not only for turning and milling but also the connection of machining step and high-level language programming under a single system user interface. SINUMERIK Operate V4.5 also brings new, powerful functions.

This user guide provides you with an overview of the range of functions of SINUMERIK Operate and gives you useful tips and tricks for your daily work. In addition to the SINUMERIK Operate chapter – with general operating tips – the other chapters give practical expert knowledge for the milling, turning and working with multitasking machines. The appendix contains an overview of the SINUMERIK Operate shortcuts and an overview of G code commands.

2 SINUMERIK Operate

SINUMERIK Operate has a uniform operating and programming interface with powerful functions for turning and milling. The functions and operating instructions described in this chapter therefore apply irrespective of whether you work on a turning or a milling machine or at a machining center.

2.1 Uniform user interface for turning and milling

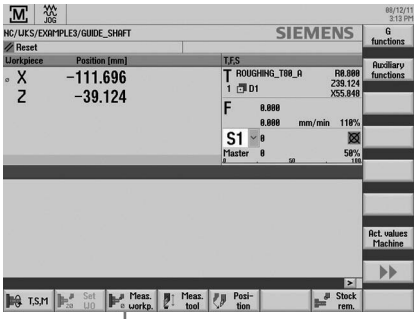
SINUMERIK Operate has a uniform operating and programming interface with powerful functions for turning and milling. The functions and operating instructions described in this chapter therefore apply irrespective of whether you work on a turning or a milling machine or at a machining center.

- HMI-Advanced, ShopMill and ShopTurn combined under a single interface
- Intuitive and clear operation and programming, including animated elements
- Representation in the modern Windows style
- New powerful functions ...
  - Setup, programming, tool and program management
  - for complete machining
  - Multi-channel capability with ShopTurn for multi-channel machines, among other things, synchronization of programs with programSYNC and much more...
- CNC programming for the highest level of productivity using programGUIDE
- Machining step programming for the shortest programming time with ShopMill and ShopTurn

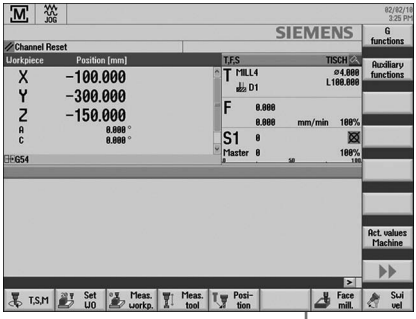
The following overview provides an initial introduction to the look & feel of SINUMERIK Operate:

Setting-up operation

A single user interface for almost all SINUMERIK controllers ...



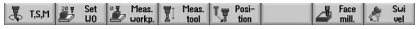
Turning



Milling

SINUMERIK Operate universal

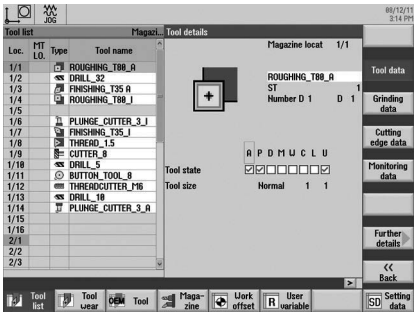
SINUMERIK Operate turning



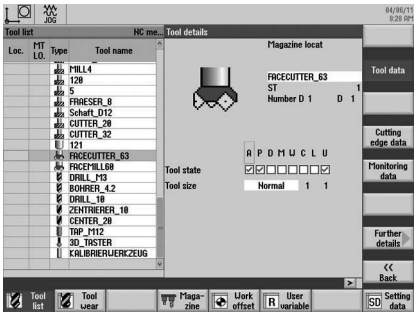
SINUMERIK Operate milling

Tool management

Efficient management of the tool data including all details and sister tool handling ...



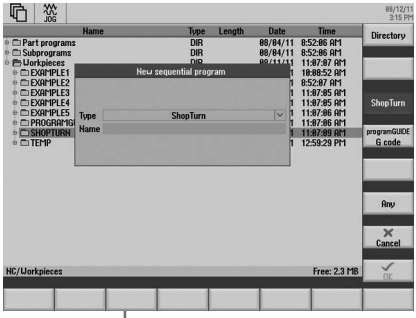
Turning



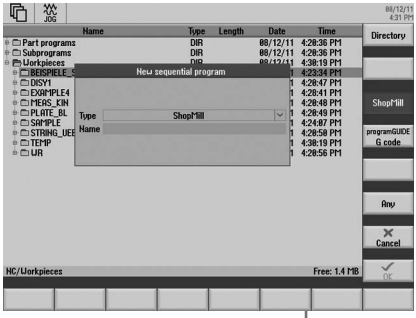
Milling

Program manager

Time savings thanks to user-friendly data transmission and simple program handling ...

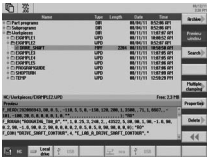


Turning

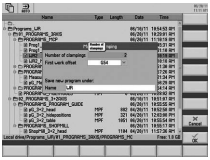


Milling

Backing up setup data ...



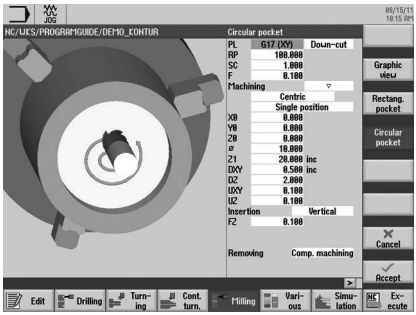
Preview window ...



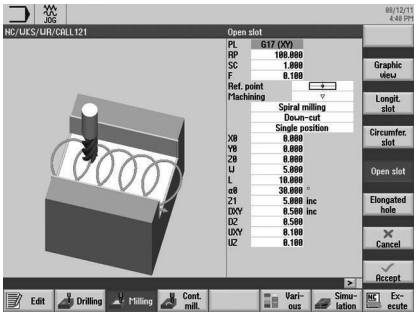
Multiple clamping for ShopMill ...

programGUIDE

Maximum productivity and flexibility in the programming combined with innovative technology and machining cycles ...



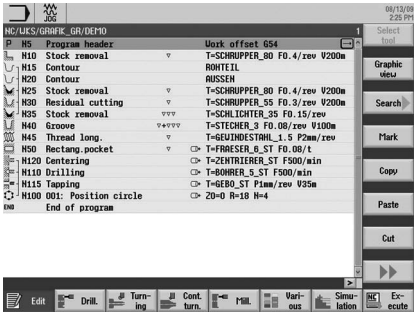
Turning



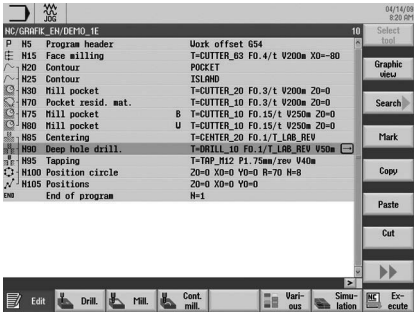
Milling

ShopTurn/ShopMill

In addition to programGUIDE, ShopTurn/ShopMill offers the unique machining step programming to achieve the shortest programming times in the single-part production.

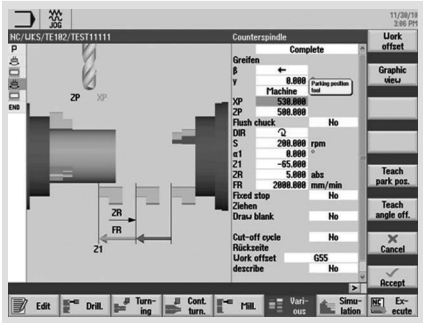


Turning



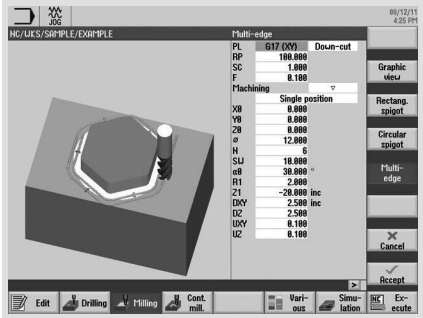
Milling

2.2 Help screens, animated elements, graphic view, tooltips, help



Help screens ...

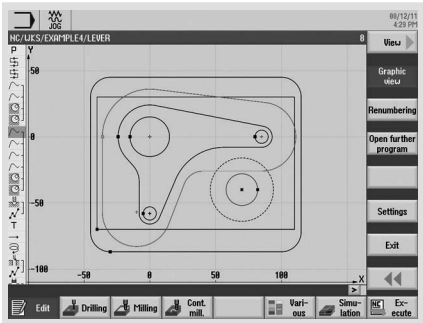
A help screen is displayed for each machining cycle. The current parameter is color-highlighted in the help screen.



Animated elements ...

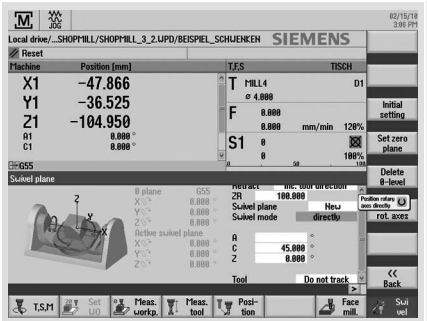
Animated elements help you for the parameterization of cycles with intuitive animation of the motion sequences.

The animation starts automatically after a few seconds.



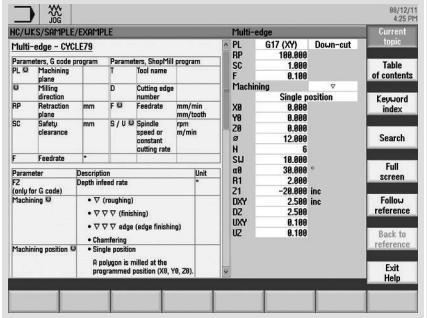
Graphic view

The “Graphic view” softkey can be used to change to the individual cycle screens or to call the broken-line graphics for the complete workpiece.



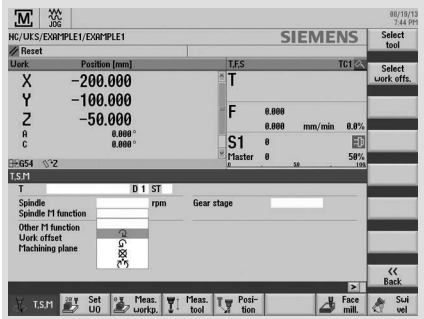
A tooltip is displayed for each input field. Selection fields are indicated with the additional symbol in the tooltip.

If an entered value is not permitted, the tooltip and the input field are displayed in red font and with a red background, respectively.



The “Help” key fetches the context-sensitive help that can be used in all screens (also in the machine data area).

2.3 TSM mode/zero offsets



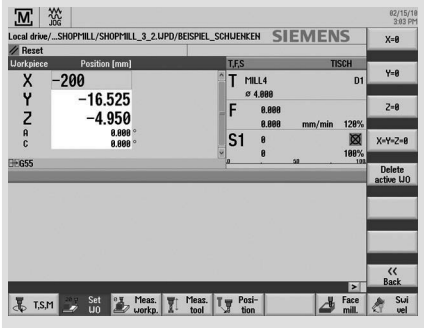
T,S,M or T,S,M

In TSM cycle (manual mode), you can, among other things,

- perform a tool change (T) also with spare tool, with direct access from the tool table,
- define the spindle speed and direction (S),
- enter M functions (M),
- activate zero offsets,
- select the machining plane,
- specify the gear stage.

SELECT

For selection fields, such as Spindle M function or Machining plane, you can use the "Select" key to make a selection; clicking any button opens the list.



Set WO or Set WO

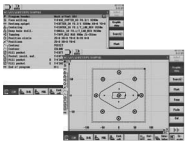
In the "Set ZO" dialog box, you can write direct position values for individual axes in the actual value display.

The difference between the position value in the machine coordinate system (MCS) and the workpiece coordinate system (WCS) is saved in the active zero offset.

Note: More information about setup functions in JOG and for in-process measurements can be found in the appropriate chapters for turning or milling.

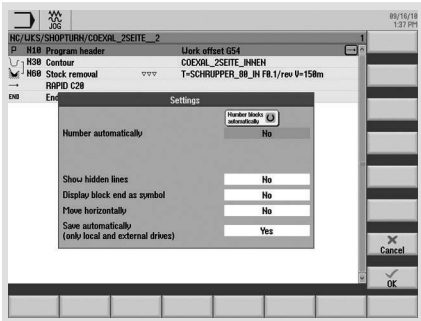
2.4 Programming

SINUMERIK Operate provides the appropriate programming method for every field of application.

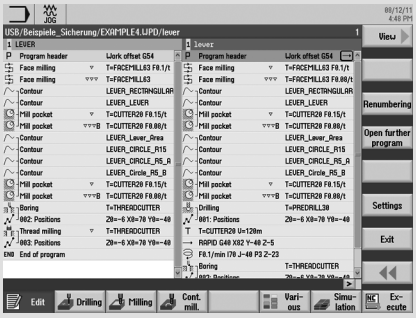
ShopMill/ShopTurn	programGUIDE	programSYNC	ISO-Code
workstep programming	DIN/ISO & SINUMERIK high-level language	multi channel programming	with cycle support
			
• Designed for shortest programming time • Tailored for single parts and small batch sizes	• Designed for maximal flexibility and shortest machining time • Tailored for medium to large batch sizes	• Increased productivity for multi-channel machines • Suitable for single-piece small-batch and batch production	• Designed for maximal ISO-code compatibility • Focused on medium to large batch sizes

The user and programming interface is uniform for all cycle screen forms irrespective of whether you use programGUIDE or ShopMill/ShopTurn. The only difference, the tool call and the technological information are directly integrated in the cycle for ShopMill/ShopTurn. More detailed information on the cycles is contained in the turning and milling sections.

Tips for working with the editors and the cycle screens, such as how to structure programs using blocks, follow.



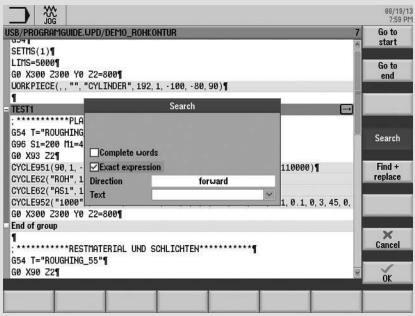
You can configure the editor for ShopMill/ ShopTurn or programGUIDE. Among other things, you can set whether, for example, block numbers should be displayed.



If, for example, you want to compare programs or transfer program sections from one program to another program, you can open two programs concurrently.

You can use the “Next window” key to the switch between the windows.

Click the “Close” softkey to close the second program.



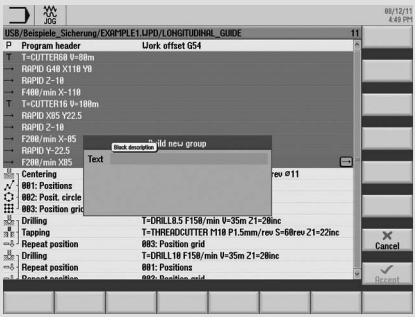
Edit Search

Click the “Find” softkey to start the search in the editors.

➡ ***Ctrl+F shortcut***

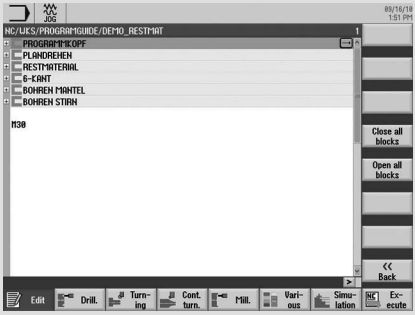
Edit Mark Search

If you select a line before calling the search screen, the selected text will be transferred to the search field.



Edit Build group

To structure large programs and individual machining steps, you can form program blocks in ShopMill/ShopTurn and in programGUIDE. The lines selected in the editor are grouped to form a program block.



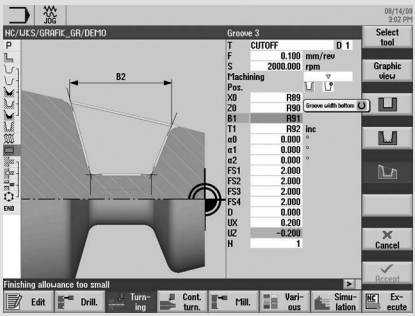
Edit View

Open all blocks or Close all blocks

You can open and close the program blocks of the program.

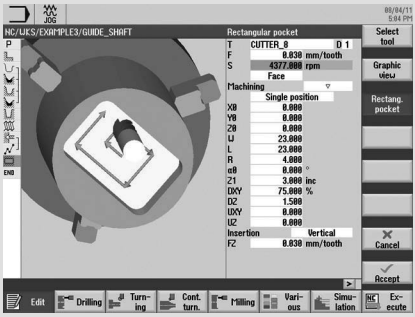
Edit ▶ or ◀

You can use the arrow keys to open and close individual blocks.



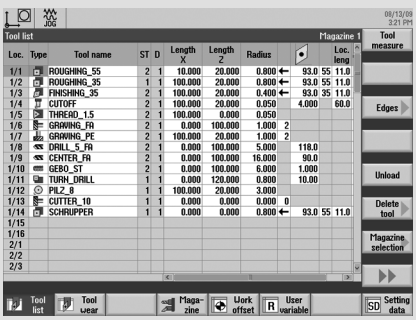
You can use parameters and variables to define the ShopMill/ShopTurn or programGUIDE cycles.

For more information about defining variables and the use of parameters, consult the documentation.



In ShopMill/ShopTurn, the feedrate/speed values are converted automatically by pressing the “SELECT” button.

2.5 Tool management

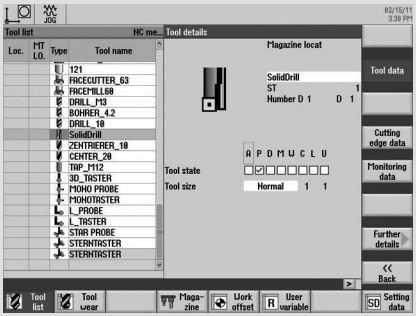


Tools with the same name are created as sister tools. In the ST column (ST = Sister tool), the sister tools are identified with an incrementing number.

Tool list or Tool list

Tools with their complete operating data can be managed in the tool list. This includes:

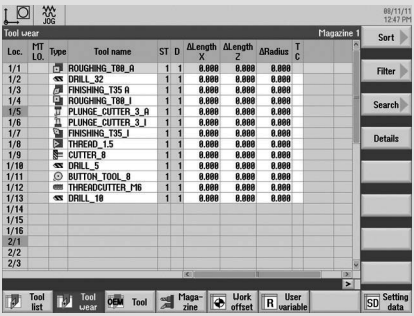
- The tool type (displayed as an icon)
- The unique tool name (max. 24 characters)
- Number of cutting edges per tool (max. 9)
- Tool length and diameter and/or cutting plate geometry
- Nose angle for drills or number of teeth for milling tools
- Direction of spindle rotation and coolant (level 1 and 2)



Tool list or Tool list

Details

You can view the tool details, such as tool status, using the “Details” softkey. The tool is displayed graphically.



Tool wear or Tool wear

The tool wear list contains all the information required during operation.

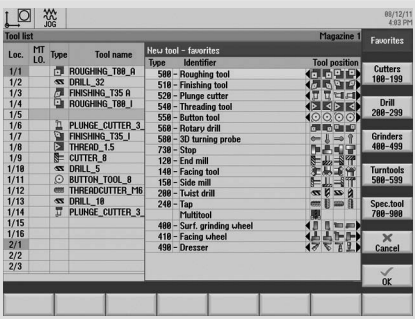
You can automatically monitor the tools' working times via the workpiece count, tool life or wear.

In addition, you can disable tools when you no longer wish to use them.

The following overview shows the used symbols and explains the tool status:

Icon/Designation	Meaning
Red "X"	The tool is disabled.
Yellow triangle pointing downward	The prewarning limit has been reached.
Yellow triangle pointing upward	The tool is in a special state. Place the cursor on the marked tool. A tooltip provides a short description.
Green border	The tool is preselected.

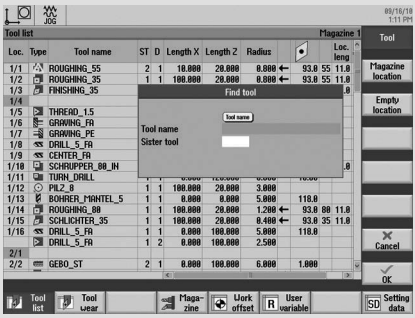
Magazine/location number	Meaning
Green double arrow	The magazine location is positioned at the change position.
Gray double arrow (configurable)	The magazine location is positioned at the loading position.
Red "X"	The magazine location is disabled.



or

When you create new tools, you can select the relevant tool from clearly arranged tables for the associated tool types. Among other things, the favorites are grouped in a separate table.

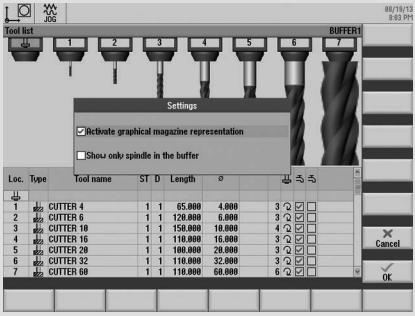
Multitools can also be created when a milling spindle or a dual tool holder is used.



or

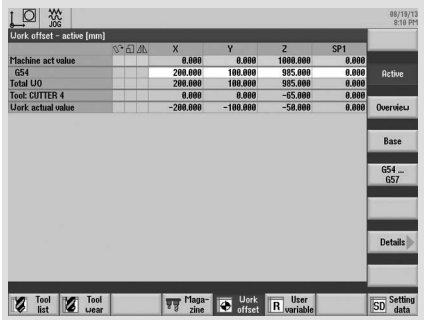
Click the “Find” softkey to open the tool search. You can search for tools, magazine locations or empty locations.

➡ ***Ctrl+F shortcut***



You can select in the settings whether, for example, the tools should be represented graphically in the magazine.

2.6 Zero offset

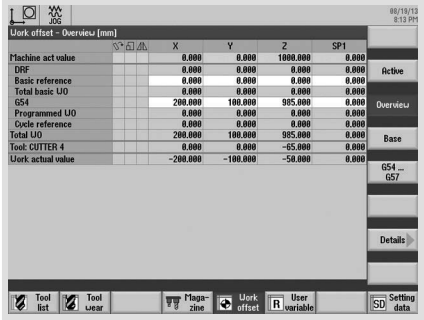


Work offset | Active

This view lists all zero offsets and frames active at runtime as well as the actual values of the Machine Coordinate System (MCS) and Workpiece Coordinate System (WCS).

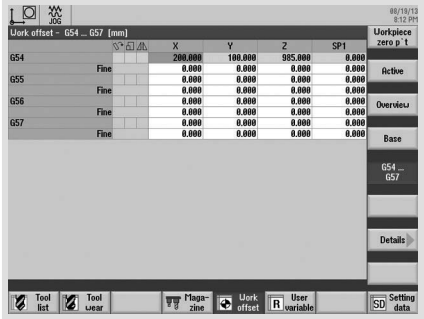
Rotations, scaling and mirroring also shown as icons

- Active zero offsets can be edited in the Reset status.



Work offset | Overview

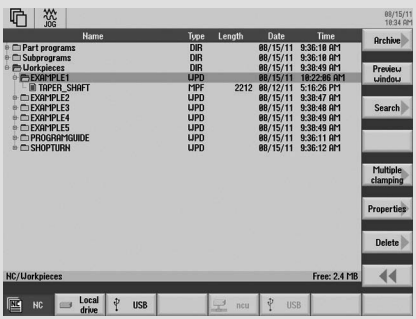
This view shows an overview of all offsets.



Work offset | G54... G57

All settable offsets, divided into coarse and fine offsets, are displayed in the “Zero Offset - G54..G599” window. Rotation, scaling and mirroring are displayed.

2.7 Program management



You can use the program manager to manage folders and programs in a similar manner as the Windows Explorer.

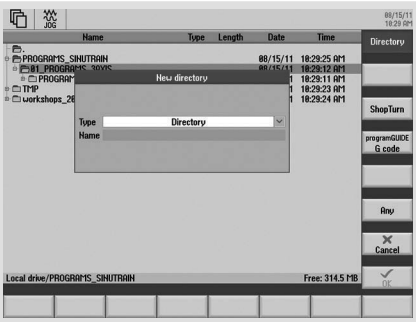
Functions, such as copy and select, can be found on the vertical softkey bar.

Active programs have a green background.

The storage capacity of the external memory media or the NC memory is displayed at the lower right.

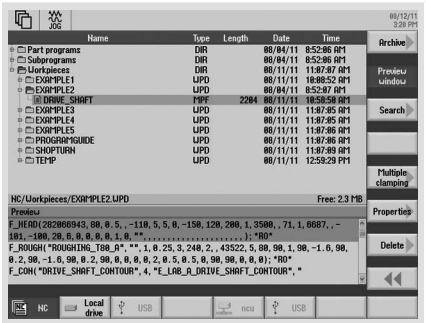
You can directly access all connected drives/ network drives.

➡ ***The shortcuts for program management can be found in the appendix.***

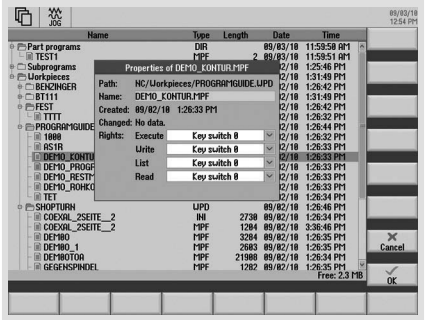


You can create folders and programs in a tree structure, on local drives and on the NC. The program names of the part programs have a maximum length of 24 characters.

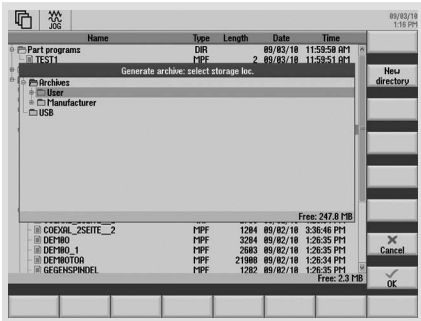
You can manage subdirectories as well as save and display files of any type (e.g. *.png, *.pdf, *.ini) on external storage media and on local drives. You can also manage subdirectories on the NC.



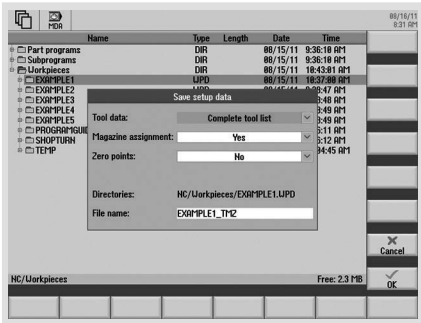
You use the “Preview window” softkey to activate the display of the preview window.



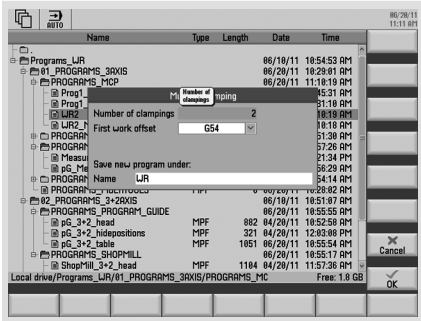
You can rename programs and folders in the “Properties” dialog. You can also define access rights for programs and folders, such as according to service, user, etc.



To backup and archive programs or program directories, you can create archives on drives/ network drives.

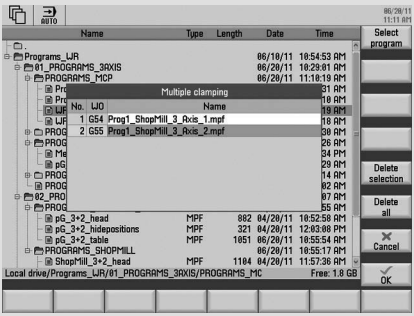


A data backup for the complete setup data, such as zero offsets, tool data, etc., can be performed for ShopMill/ShopTurn or programGUIDE programs. This allows a simple restore of workpiece setups.



You can use the “Multiple clamping” function to group the machining programs for the respective clampings to form a complete program. The machining programs for the clampings can be identical or differ, as necessary.

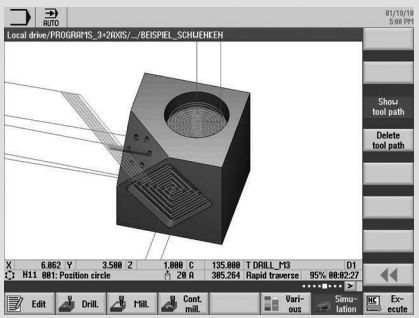
The machining steps are sorted in this program so that the number of tool changes (and thus the idle times) is reduced to a minimum.



In this view, you assign the associated zero offsets and programs to the clampings.

2.8 Workpiece visualization (simulation and mold making view)

To increase the programming reliability and as control capability, SINUMERIK Operate provides a user-friendly simulation (2D and 3D) as well as the fast display for mold making applications.



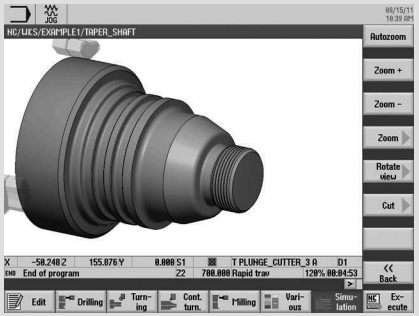
- The machining time is calculated automatically (display at the lower right).

Simu-lation or Simu-lation

The simulation offers the following different views:

- 3D simulation with 3-level view and volume model of the finished part.

➡ ***The shortcuts for operating the simulation are contained in the appendix.***

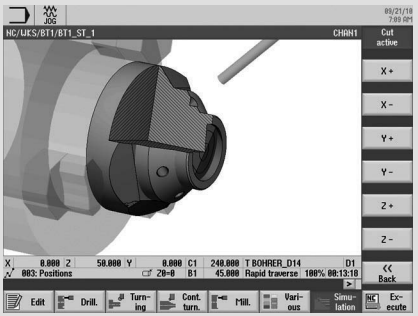


Simu-lation or Simu-lation

Details

- Details zoom, magnifying glass for enlarging a workpiece detail, rotate view.

➡ ***The shortcuts for operating the simulation are contained in the appendix.***

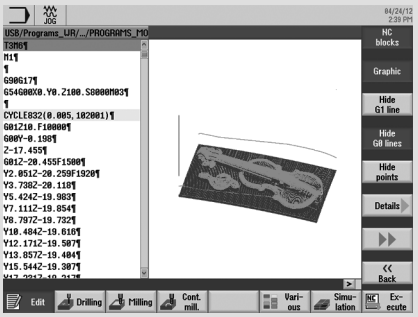


Simu-lation or Simu-lation

Details Cut

- 3D cross-section view details.

➡ ***The shortcuts for operating the simulation are contained in the appendix.***



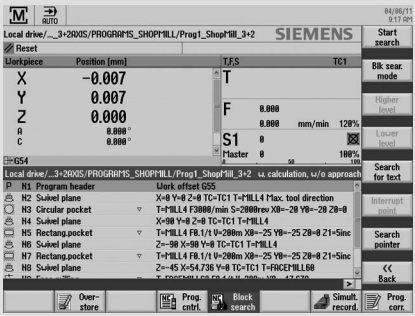
Edit Mold mak. view

For especially large part programs, the mold-making view can be used in edit mode.

G0 and G1 lines as well as the display and hiding of points can be used for the representation.

Only G0/G1 blocks can be represented in the mold-making view (no cycle processing). No syntax check is performed.

2.9 CNC operation in Automatic mode (AUTO)



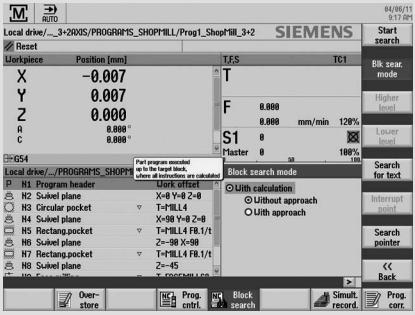
Requirement: The program is selected and the controller is in Reset status.

Block search

You can perform a block search in the Reset machine status. The program data is prepared so that all relevant parameters (tool, zero offsets, M functions, etc.) are available when the program is entered.

The following search variants are available:

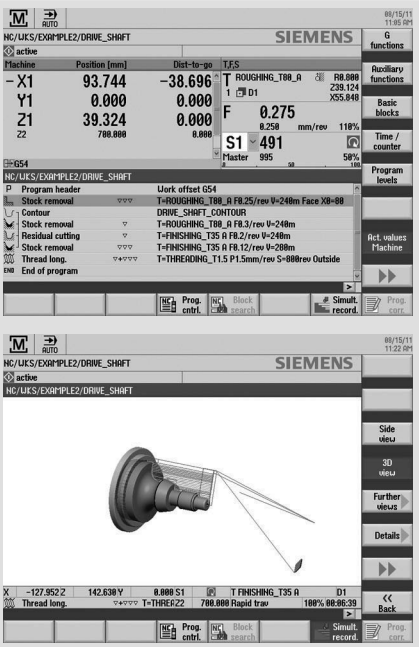
- Targeted at the interruption point, even after power off, is also possible
- To any CNC block in the DIN/ISO programs
- To any subroutine levels in DIN/ISO programs
- In ShopMill/ShopTurn machining step programs
- In the position patterns for the ShopMill/ShopTurn machining step programming
- Accelerated block search in large mold making programs



Block search **Blk sear. mode**

The block search can be customized:

- With calculation/without calculation
- With approach/without approach

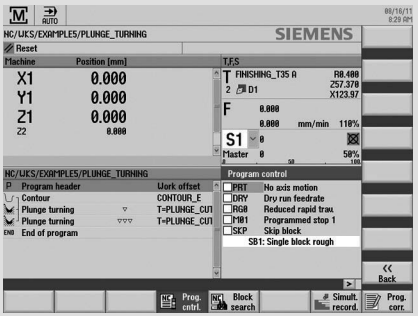


or

If required, you can activate the simultaneous recording while machining the workpiece.

You can select various display formats for the simultaneous recording.

➡ ***The shortcuts for the simultaneous recording are described in the appendix.***

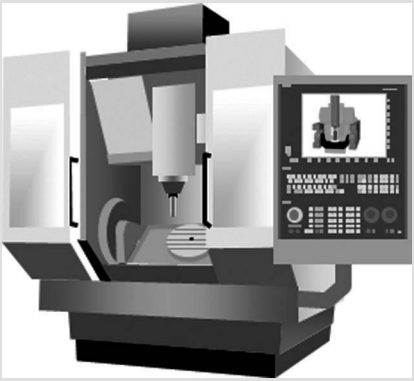


Ex-ecute Prog. cntrl.

You can influence the execution of a program in AUTO and MDA mode:

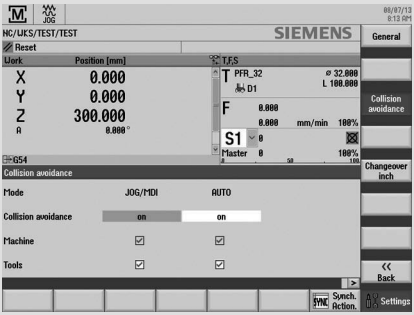
- PRT – no axis movement
  You can also activate the program processing without axis movements, also together with the “Dry run feedrate” function.
- DRY – dry run feedrate
  The traversing velocity programmed in conjunction with G1, G2, G3, CIP and CT is replaced with a defined dry run feedrate. Caution for activated dry run feedrate - the modified feedrate values may cause a violation of the cutting rate.
- RG0 – reduced rapid traverse
  You define the reduced rapid traverse in the settings for Automatic mode.
- M01 – programmed Stop 1
  The program processing stops for those blocks in which the M01 supplementary function is programmed. In this way you can check the previously-obtained result during the machining of a workpiece.
- DRF – handwheel offset
  This selection enables an additional incremental zero offset with the electronic handwheel while being machined in Automatic mode.
- SKP
  Skip blocks are skipped during machining.

2.10 Collision Avoidance



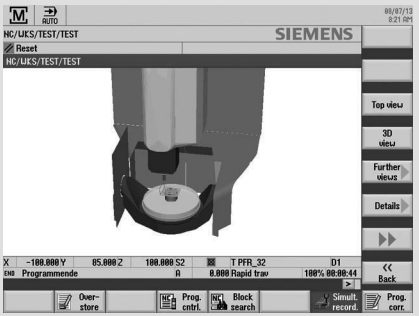
The SINUMERIK Collision Avoidance option provides optimum protection against unwanted collisions of moving machine components with static machine components. The collision monitoring is also possible for complex machining, such as 5-axis simultaneous milling and turning with the B-axis.

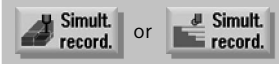
Note: Prerequisite for use of the collision monitoring is the availability of the relevant machine data and the appropriate visualization.



Settings → Collision avoidance

The collision monitoring can be activated in the Machine operating area for the JOG, MDA and Automatic operating modes.





A 3-D visualization of the collision monitoring is available for the simultaneous recording in the Automatic operating mode.

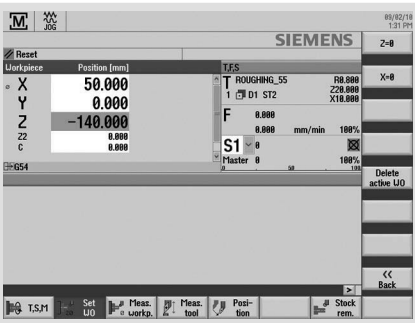
3 Turning technology

In addition to the guidelines for setup and programming, this chapter provides an overview of the technology cycles for drilling, turning, contour turning, milling and contour milling. You can also obtain information on the machining of workpieces on a counter-spindle and an overview of the cycles for the in-process measurement.

3.1 Setting up

The following sections contain information for setting the zero offset and measuring workpieces and tools in JOG.

3.1.1 Setting the zero point

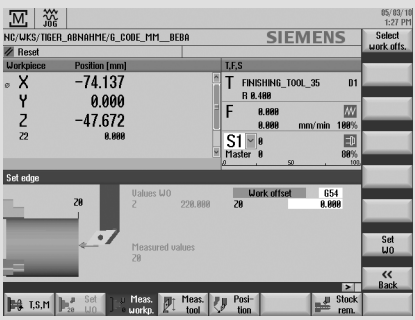


Requirement: A zero offset must be active.

In the “Set ZO” dialog box, you can write direct position values for individual axes in the actual value display.

The difference between the position value in the machine coordinate system (MCS) and the workpiece coordinate system (WCS) is saved in the active zero offset.

3.1.2 Workpiece zero



Meas. workp.

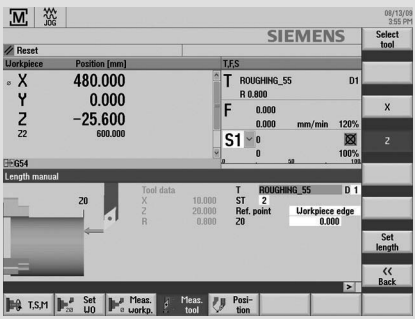
You are guided graphically for setting the workpiece zero.

To determine the zero point, determine the length of the workpiece and save the position of the cylinder face surface in the direction Z in a zero offset.

When the workpiece zero/zero offset is calculated, the tool length is automatically taken into account.

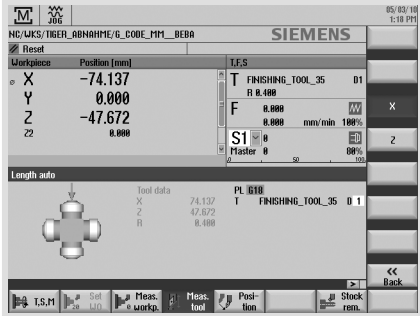
If you wish to measure the workpiece zero in "Measuring Only" mode, the measured values are merely displayed without any changes being made to the coordinate system.

3.1.3 Tool measurement



Meas. tool **Manual**

You are guided graphically for calibrating the tools (all tool types).

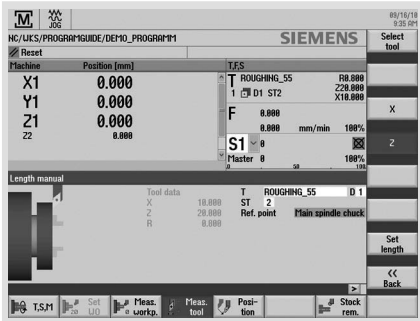


Meas. tool | Automatic

If you wish to measure your tools with a tool probe, the machine manufacturer must parameterize special measuring functions for that purpose.

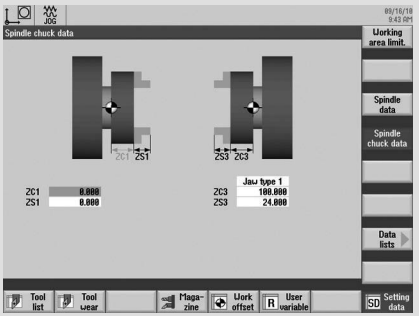
Enter the cutting edge position and the radius or diameter of the tool in the tool list before performing the actual measurement.

If turning tools with tool carrier that can be oriented are measured around Y using any positions of the swivel axis, then it should be taken into consideration that the turning tool is measured with the same tool position in both axes X/Z, assuming that this is possible.



Meas. tool

During the tool measurement, you can also scratch at the primary or counterclamping holder (fixed point in the machine).



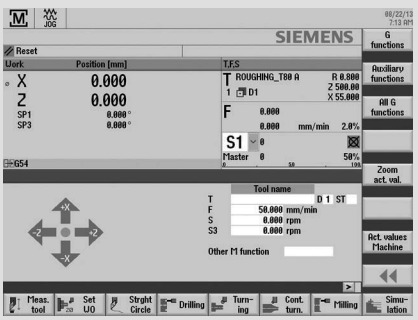
SD Setting data **Spindle chuck data**

Define the main and counterclamp holder in the spindle chuck data.

Parameter ZC1 is used as reference point for the tool measurement.

You must enter this parameter if you are machining with the counterspindle! This function is available only in the ShopTurn machining step programming.

3.2 Manual Machine



Note: Overviews of the drilling, turning, milling and contour-turning cycles are contained in Section 3.3, Programming.

The Manual Machine operating area provides you with machining capabilities as for a cycle turning machine. All important machining actions can be performed without an associated part program needing to be created.

The following functions are available:

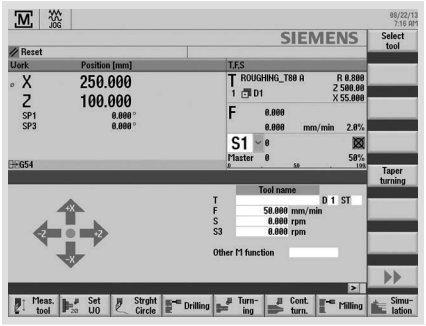
- Measure tool
- Traverse axes
- Set zero offset (ZO)
- Turn straight line / circle
- Drill, such as centered drilling, centering, deep-hole drilling
- Turn, such as stock removal, grooving, thread cutting
- Mill, such as face milling, pocket milling, multi-edge stud milling
- Turn contour

User-friendly input screens with help screens and animated elements are available for parameterizing the cycles.

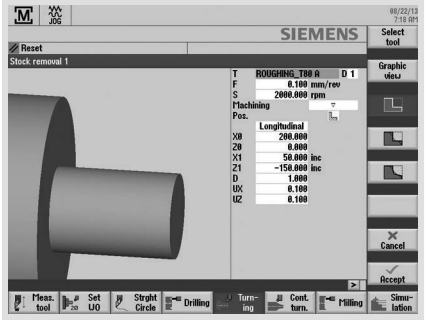
For the manual workpiece machining, the axes and spindles are controlled using the following options:

- Control of the cross slides using
  - Handwheels of the X- and Z-axis, or
  - Axis direction switches.
- Spindle control using
  - Spindle rotation direction switches.

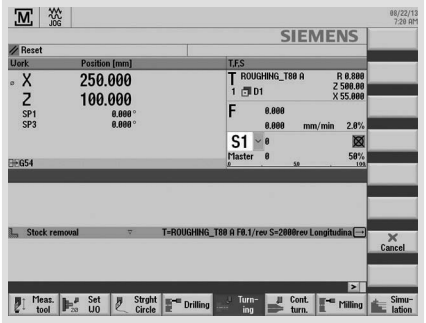
The following example with a stock-removal cycle illustrates the operation as a Manual Machine.



First position the axes, in the example, X=250, Z=100.



Parameterize the stock-removal cycle.



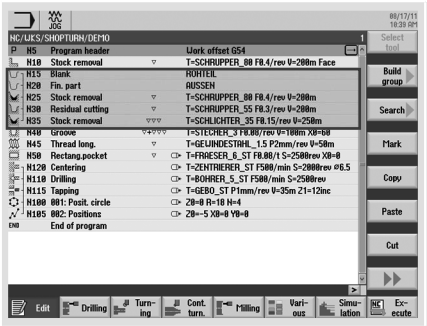
After the acceptance, start the machining with CYCLE START.

3.3 Programming

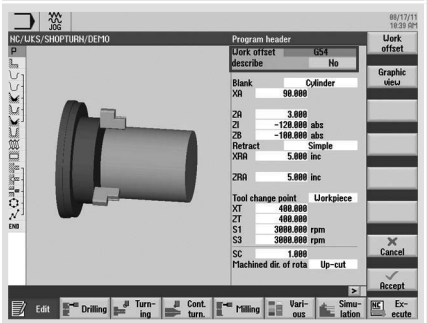
This section provides you with an overview of the design of the programs under ShopTurn machining step programming and programGUIDE.

3.3.1 ShopTurn

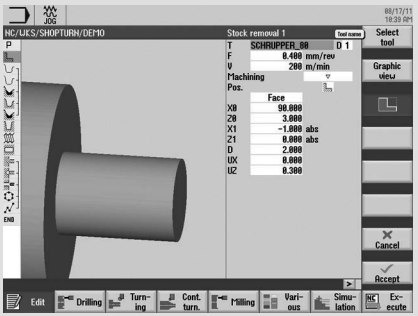
Each program line in the ShopTurn machining step editor represents a technological machining step (example: face turning, centering, ...) or the geometric data required for the machining steps (position patterns or contours). You do not need any knowledge of DIN/ISO for entering the individual machining steps. All required technical and geometric parameters are entered in cycle screen forms. If necessary you can also enter DIN/ISO blocks and control functions in the machining step editor.



The example shows the linking of the contours with the associated stock removal cycles.



You can decide in the program header whether a value from the program is to be written to the zero offset. This avoids the need for the separate back up of the zero points.



The technology data is directly integrated in the cycle.

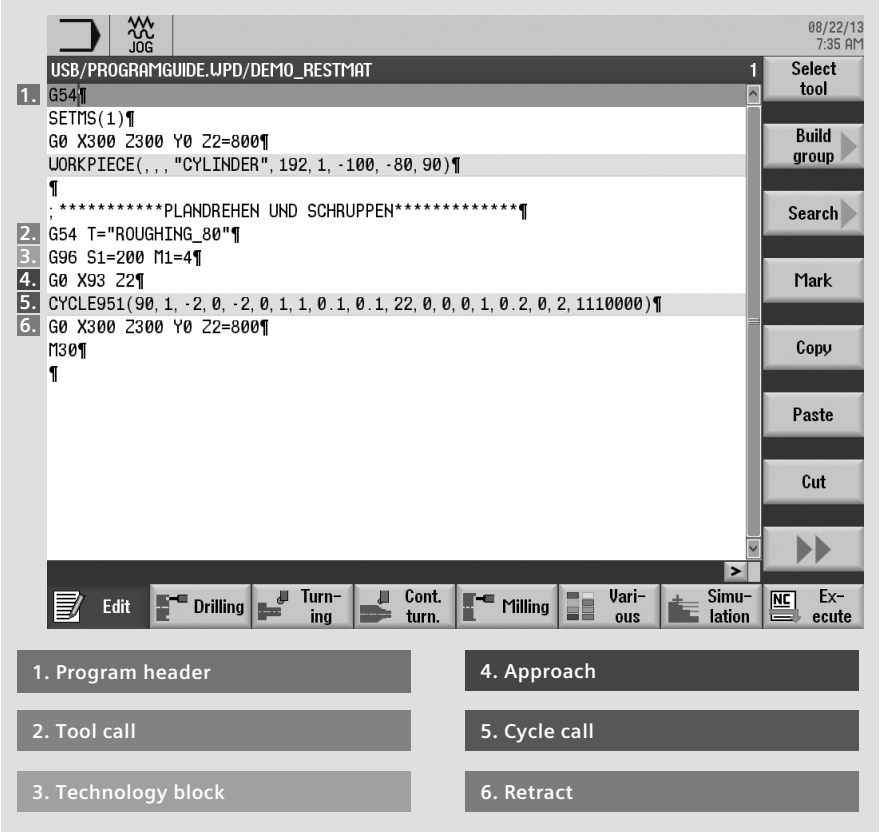
The workpiece is shown dynamically during programming.

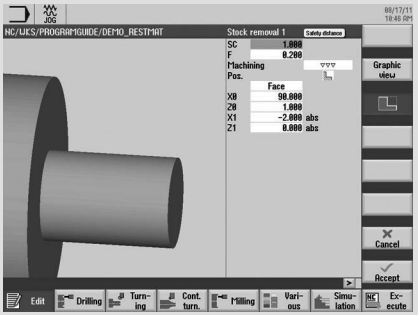
You can also enter parameters in the input fields of the cycles.

3.3.2 programGUIDE

In programGUIDE, in additional to the DIN/ISO editor, the technology cycles are also available similar to the ShopTurn machining step programming. This ensures an optimum integration. You also enter the calls for tool, feedrate, spindle speed, etc. in the DIN/ISO editor.

The following figure shows the structure of a program.





As example, you see from the above example the technology cycle for stock removal (see cycle call).

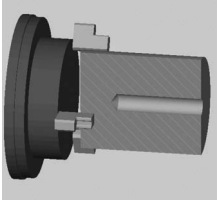
3.4 Drilling

This section provides an overview of the drilling cycles in the ShopTurn machining step programming and programGUIDE. As an example, the cycles for centered and off-centered drilling for ShopTurn machining step programming and programGUIDE are explained.

3.4.1 Overview of drilling cycles

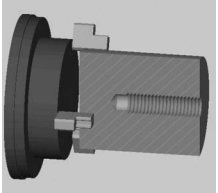
An overview of the drilling cycles in ShopTurn machining step programming and programGUIDE with application tips is shown below.

Drilling centric Only ShopTurn



Drilling centric

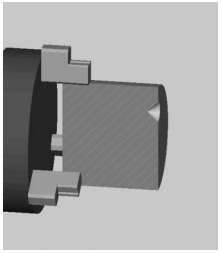
Cycle to generate centric drill holes with a static tool



Thread centric

Cycle for creating centric threads with a static tool

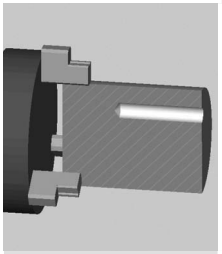
Centering



Cycle for centering at any positions

- For ShopTurn, only with a driven tool

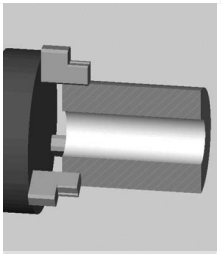
Drilling Reaming



Drilling

Cycle for drilling holes in an infeed at any positions

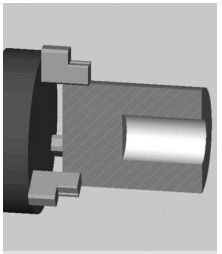
- For ShopTurn, only with a driven tool



Reaming

Cycle for drilling corrective work with a reamer at any positions

- For ShopTurn, only with a driven tool



ShopTurn:

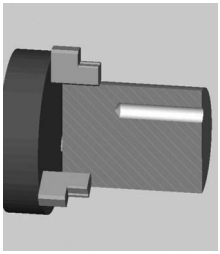
Boring

programGUIDE:

Boring

Cycle for boring drilled hole diameters with special boring tools at any positions

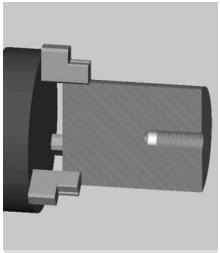
Deep hole drilling



Cycle for creating holes with more than one infeed at any positions

- For ShopTurn, only with a driven tool

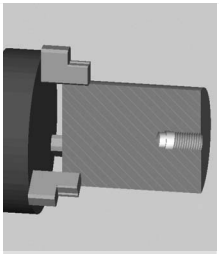
Thread



Tapping

Cycle for tapping at any positions

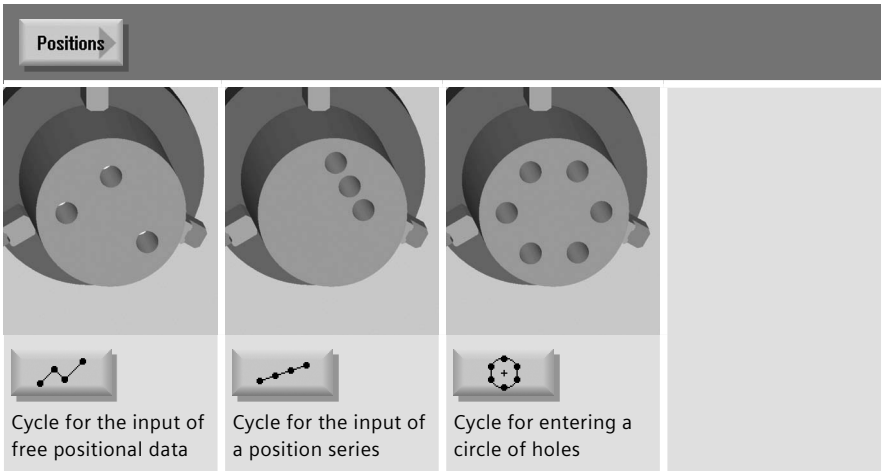
- For ShopTurn, only with a driven tool



Drl+thrd milling

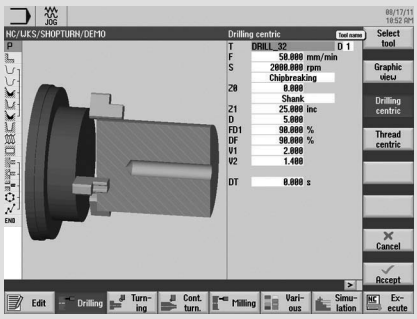
Cycle for thread cutting with special drill and thread milling cutters at any positions

- For ShopTurn, only with a driven tool



3.4.2 Drilling centered/off-centered with ShopTurn

The following example explains how you drill a centered/off-centered hole with ShopTurn machining step programming.

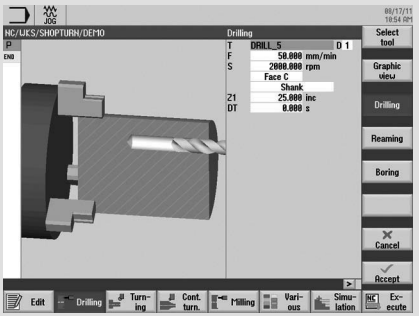


■ This cycle is available only under ShopTurn.

Drilling centric **Drilling centric**

You can use this cycle to drill a hole in the center of the end face. You can choose between chip breaking during drilling or retraction from the workpiece for stock removal.

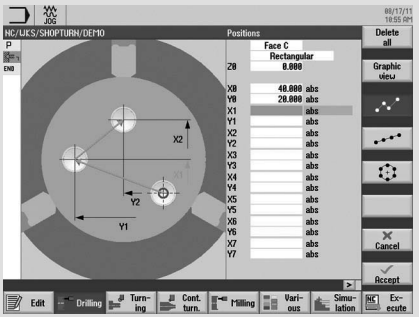
During machining, either the main spindle or the counterspindle rotates. You can use a drill, a rotary drill or even a milling cutter as the tool.



Drilling Reaming | Drilling

To make a drill hole at any position on the end face or the peripheral surface, use the “Drill” cycle.

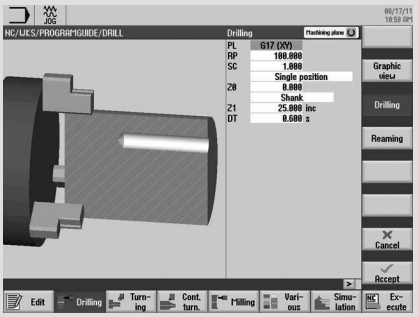
Enter the feedrate F, the final depth X1 and the dwell time DT. Then define the drilling position using the positions cycle (see below).



Positions e.g.

You can define a maximum of eight drill positions. To program other freely programmable positions, you must call the “Freely programmable positions” function again.

3.4.3 Drilling with programGUIDE

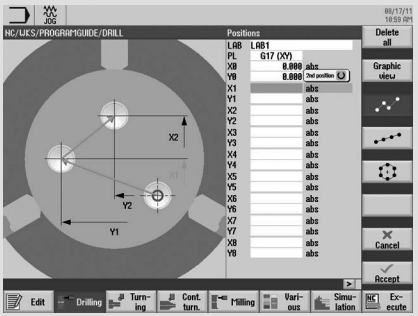


Drilling Reaming **Drilling**

To make a drill hole at any position (centered or off-centered) on the end face or the peripheral surface, use the “Drill” cycle.

Irrespective of whether you want to create a hole on the end face or peripheral surface, select the PL machining plane. Enter the drilling depth in the field Z1.

Then define the drilling position using the positions cycle (see below).



Positions e.g.

You can define a maximum of eight drill positions. To program other freely programmable positions, you must call the “Freely programmable positions” function again.

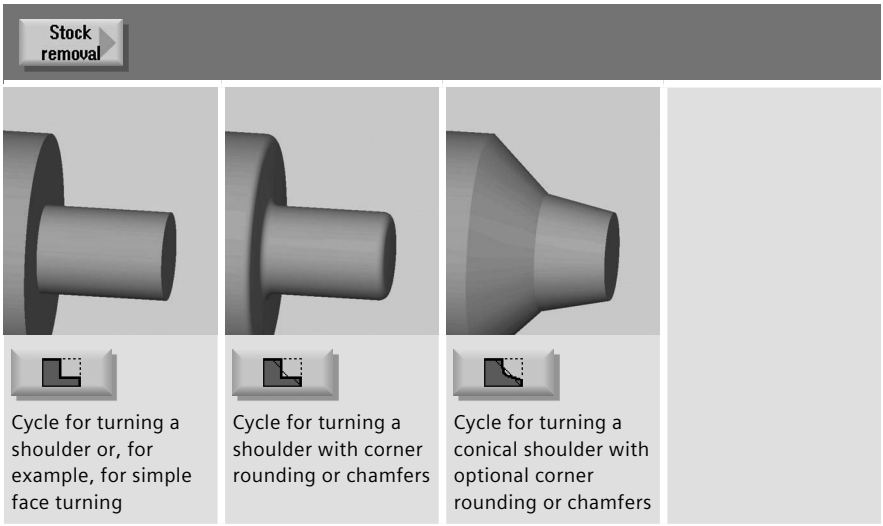
Note: You define the technology data, such as tool, direction of spindle rotation, feedrate, etc., before calling the cycle.

For the machining on the end face, activate the TRANSMIT function; for machining operations on the peripheral surface, activate the TRACYL function. The position pattern must be deselected again with MCALL.

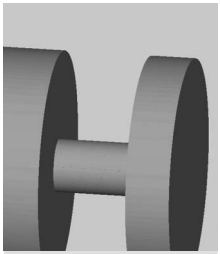
3.5 Turning

This section provides an overview of the turning cycles in the ShopTurn machining step programming and programGUIDE.

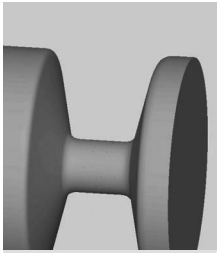
3.5.1 Overview of the turning cycles



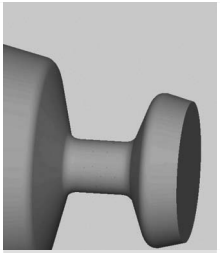
Groove



Grooving cycle without rounding

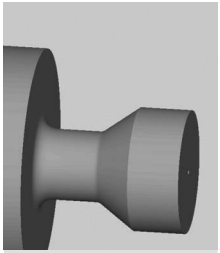


Extended grooving cycle with optional rounding or chamfers and additional conicity



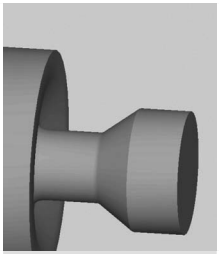
Extended grooving cycle on conical turned parts

Undercut



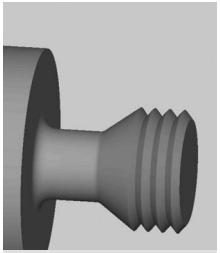
Undercut form E

Cycle for external or internal undercuts according to the standard E



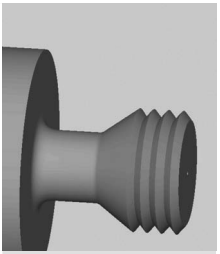
Undercut form F

Cycle for external or internal undercuts according to the standard F



Undercut thrd DIN

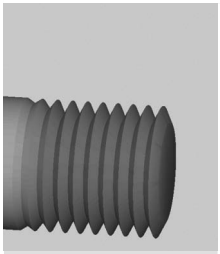
Cycle for external or internal thread undercuts according to the DIN standard



Undercut thread

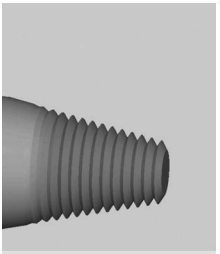
Cycle for external or internal thread undercuts without any standard (freely programmable)

Thread



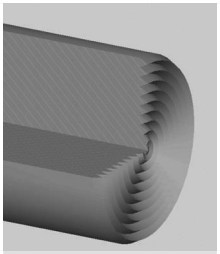
Thread long.

Thread cycle for longitudinal thread



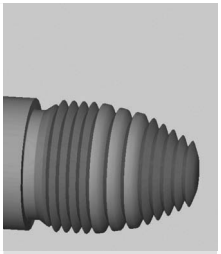
Thread taper

Thread cycle for tapered threads



Thread face

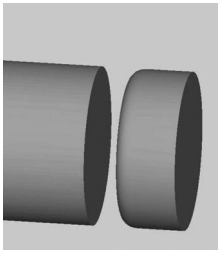
Thread cycle for face thread



Thread chain

Thread cycle for chaining different threads in a single operation

Cutoff

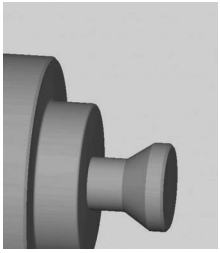


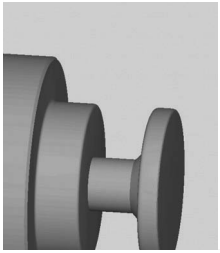
Cycle for parting rod parts

3.6 Contour turning

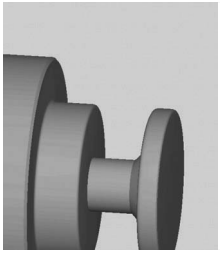
This section provides an overview of the contour machining cycles in the ShopTurn machining step programming and programGUIDE. The contour turning with ShopTurn machining step programming and programGUIDE is explained as an example.

3.6.1 Overview of the turning cycles for the contour machining

Stock removal or Cut resid stock		
	Stock removal cycle for machining any contours with the stock removal technology; it can be combined with automatic residual material detection	**Note:** Manual Machine does not support residual material detection.

Grooving or Groove resid.		
	Grooving cycle for machining any contours with the plunge-cutting technology; it can be combined with automatic residual material detection	**Note:** Manual Machine does not support residual material detection.

Plunge turning or Plunge turn.resid.

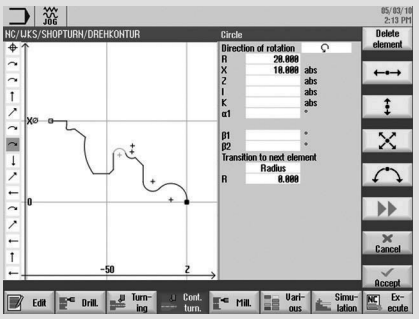


Plunge-turning cycle for machining any contours with the plunge-turning technology; it can be combined with automatic residual material detection

Note: Manual Machine does not support residual material detection.

Note: The technologies can be combined on a contour!

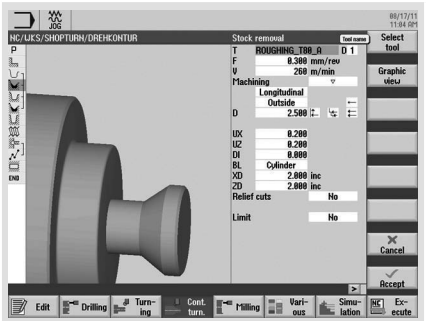
3.6.2 Contour turning with ShopTurn machining step programming



Cont. turn. New contour

Create a new contour in the contour calculator.

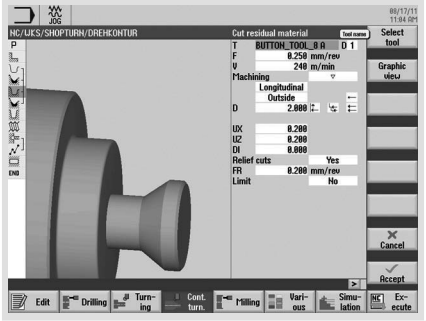
You can enter as many as 256 geometric elements.



Cont. turn. | Stock removal

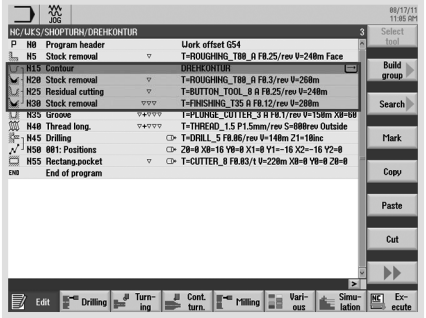
The settings that can be selected for stock removal and stock removal residual material include:

- Align cut segmentation to edges
- Alternating cutting depth



Cont. turn. | Cut resid stock

The “Stock removal residual material” cycle automatically performs a residual material detection.



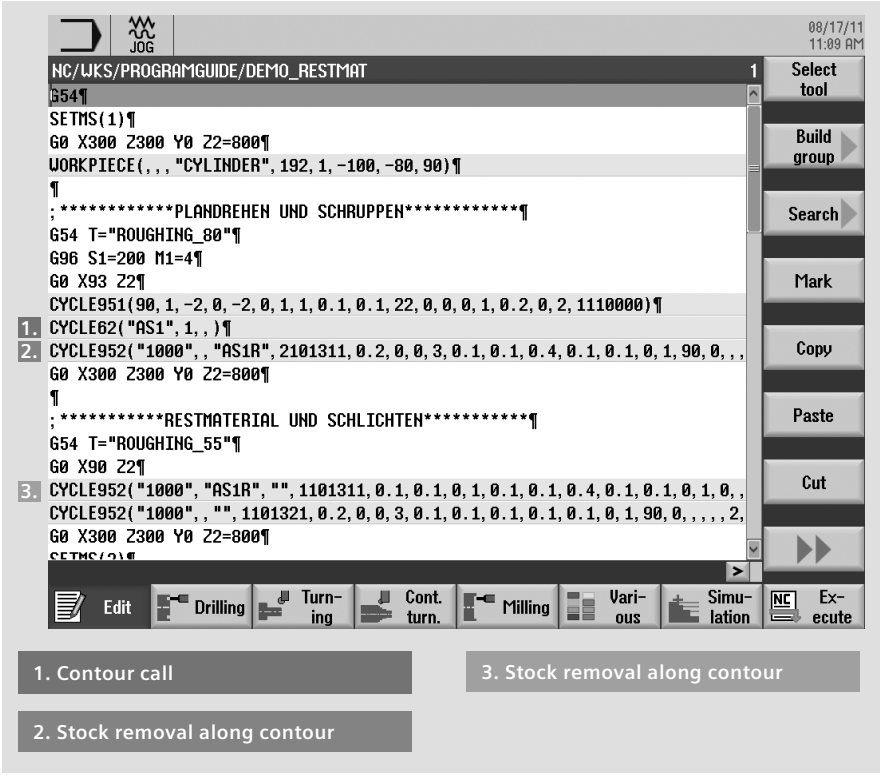
The contour and the stock removal cycles are linked in the machining step editor.

Before the residual stock removal cycle, you must always set a preceding roughing cycle on which the residual stock removal is based.

You can also program several successive residual material cycles, when, for example, different tools must be used.

3.6.3 Contour turning with programGUIDE

The contour call and calls of the stock removal and residual stock removal cycles are explained using the following example program.

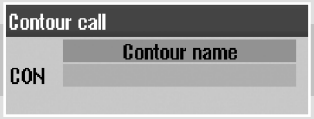
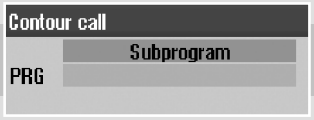
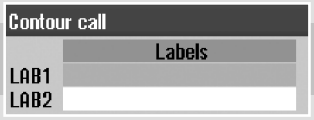


1. Contour call

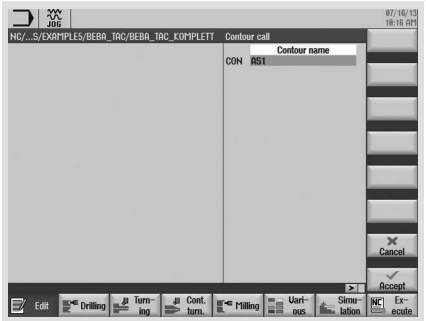
2. Stock removal along contour

3. Stock removal along contour

You have the following ways of including a contour call in a programGUIDE program:

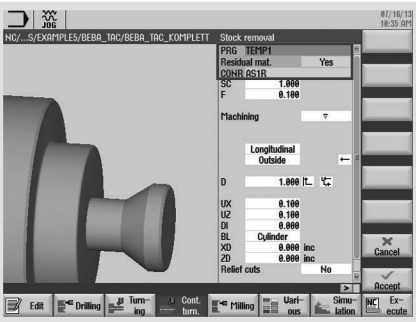
The contour is contained in the main program directly behind M30	
The contour is contained in a subroutine	
The contour is contained in the main program between two labels	
Call using labels in a subroutine	

We start by adding the contour call, in our example, the call is made using the contour name. The contour is then written and added at the program end (M30). In our example, the contour is given the name “AS1”.



First insert the contour call at the desired location in the program. Note the above figure for contour calls.

In the next step, we add the cycles for the stock removal.



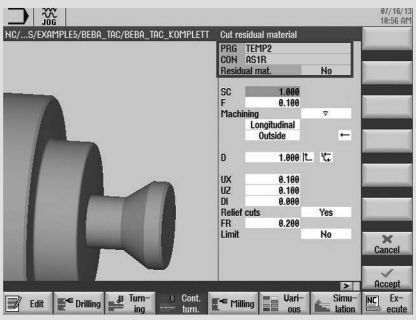
Note: The contour name for the residual stock removal must be unique because it will be used again in the next stock-removal cycle.

Cont. turn. → Stock removal

Enter a name in the “PRG” input field. This name is arbitrary, but must be unique. This means, for example, that the name must not be the same as the name of the contour to be machined. In the example, enter the name “TEMP1”.

Because the contour used in our example also has relief-cut elements that cannot be machined with the stock-removal tool, residual material results. This residual material is cut in the next machining operation. Consequently, the “Residual material” parameter must be set to “yes”. You must also specify a name for the residual-material contour, in our example, “AS1R”.

In the further program execution, the tool is now changed and the “residual material stock removal” cycle added.

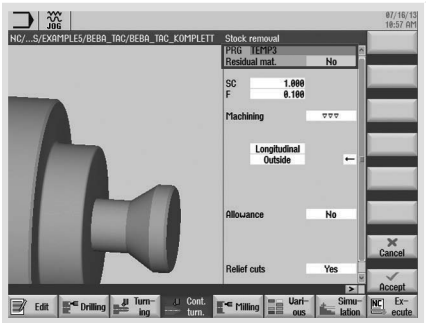


Cont. turn. → Cut resid stock

In the “residual material stock removal” cycle, enter a name in the “PRG” input field, in the example, “TEMP2”. You must now enter the name of the stock-removal contour from the previous stock-removal cycle in the “CON” input field. In our example, “AS1R”.

If residual material also results in this machining step, the “Residual material” parameter can also be set to “yes” here and a further residual-material contour created. In the example, we select, however, “no”.

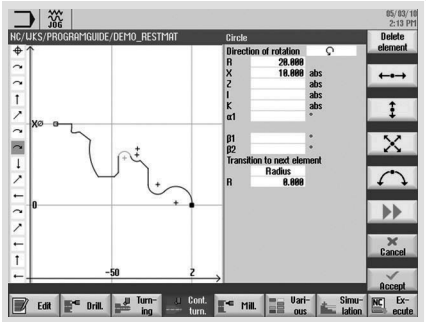
Finally, the contour is finished. To do this, select the “stock removal” cycle again.



Cont. turn. → Stock removal

Enter the name “TEMP3” in the “PRG” input field. Select the value “no” for the “Residual material” parameter and “finish” in the “Machining” field.

You must still define and add the target contour after the program end (M30).



Cont. turn. → Contour → New contour

Create a new contour in the contour calculator.

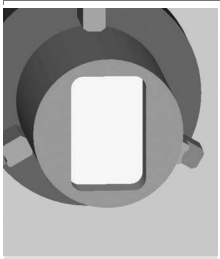
- You can enter as many as 256 geometric elements.

3.7 Milling

This section provides an overview of the milling cycles in the ShopTurn machining step programming and programGUIDE.

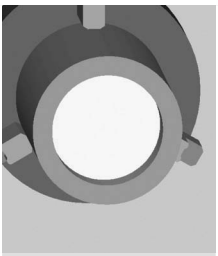
3.7.1 Overview of the milling cycles

Pocket



Rectang. pocket

Cycle to produce a rectangular pocket on the end face or the peripheral surface



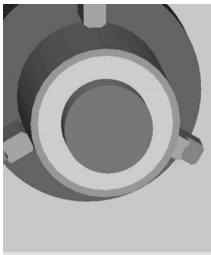
Circular pocket

Cycle to produce a circular pocket on the end face or the peripheral surface

Multi-edge spigot

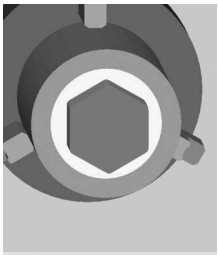
Rectang. spigot

Cycle to produce a rectangular spigot with a rectangular blank on the end face or the peripheral surface



Circular spigot

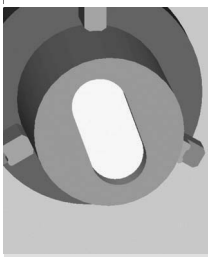
Cycle to produce a circular spigot with a round blank on the end face or the peripheral surface



Multi-edge

Cycle to produce a polyhedron on the end face

Slot

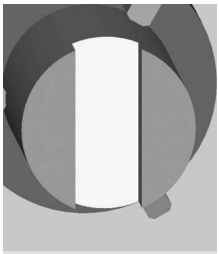


Longit. slot

Cycle to produce longitudinal grooves on the end face or the peripheral surface

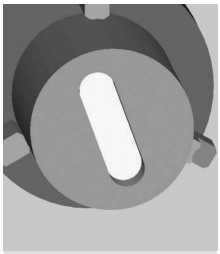
Circumfer. slot

Cycle to produce any number of circumferential grooves on the end face or the peripheral surface



Open slot

Cycle to produce open grooves on the end face or the peripheral surface using the vortex milling or plunge cutting technologies

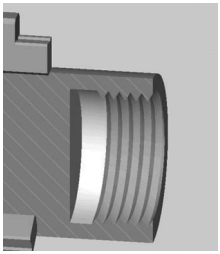


Elongated hole

Cycle to produce elongated holes on the end face or the peripheral surface

■ Only in programGUIDE

Thread milling



To produce internal or external threads on the end face or the peripheral surface using the milling technology

Engraving

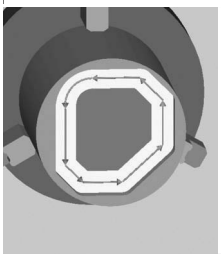
Engraving cycle for characters and numbers on the end face or the peripheral surface with any alignment

3.8 Contour milling

This section provides an overview of the contour milling cycles in the ShopTurn machining step programming and programGUIDE. The contour milling with ShopTurn machining step programming and programGUIDE is explained as an example.

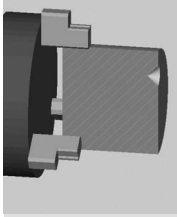
3.8.1 Overview of the milling cycles for the contour machining

Path milling



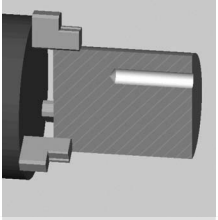
Cycle for machining arbitrary contours by milling along the contour path on the end face or the peripheral surface

Rough drill



Centering

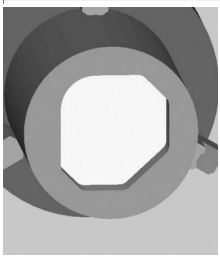
Cycle for centered drilling for the specific predrilling of a start hole for pocket milling



Rough drill

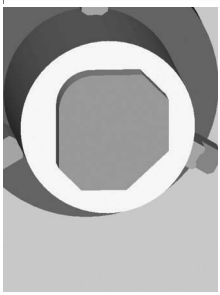
Cycle for drilling for the specific predrilling of a start hole for pocket milling

Pocket or Pocket res.mat.



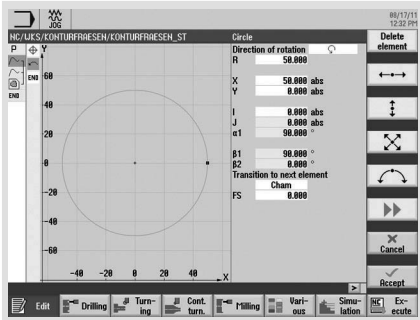
Cycle for machining any pocket contours on the end face and the peripheral surface; it can be combined with automatic residual material detection

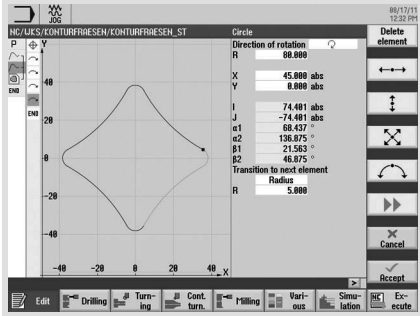
Spigot or Spigot res. mat.



Cycle to produce any spigot contours with a defined initial blank on the end face and the peripheral surface; it can be combined with automatic residual material detection

3.8.2 Contour milling with ShopTurn machining step programming





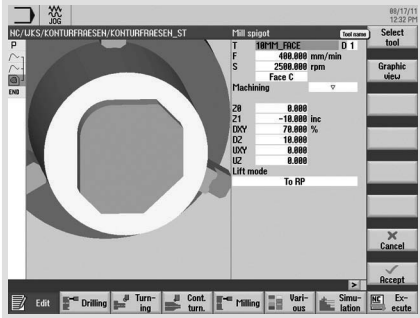
Mill. Cont. mill. Contour

New contour

You can define open contours or closed contours (pockets, islands, spigots) and machine them with path milling or milling cycles.

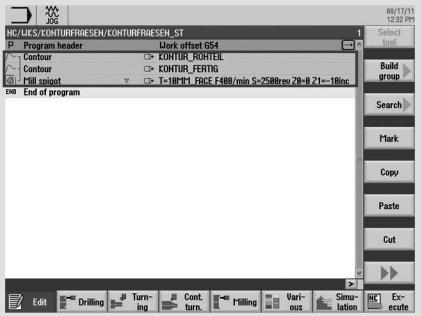
In the example, a raw and finished part contour is created in the contour calculator. Contours for spigots must be closed. ShopTurn interprets the first specified contour as a blank contour and all others as spigots.

- You can enter as many as 256 geometric elements.



Mill. Cont. mill. Spigot

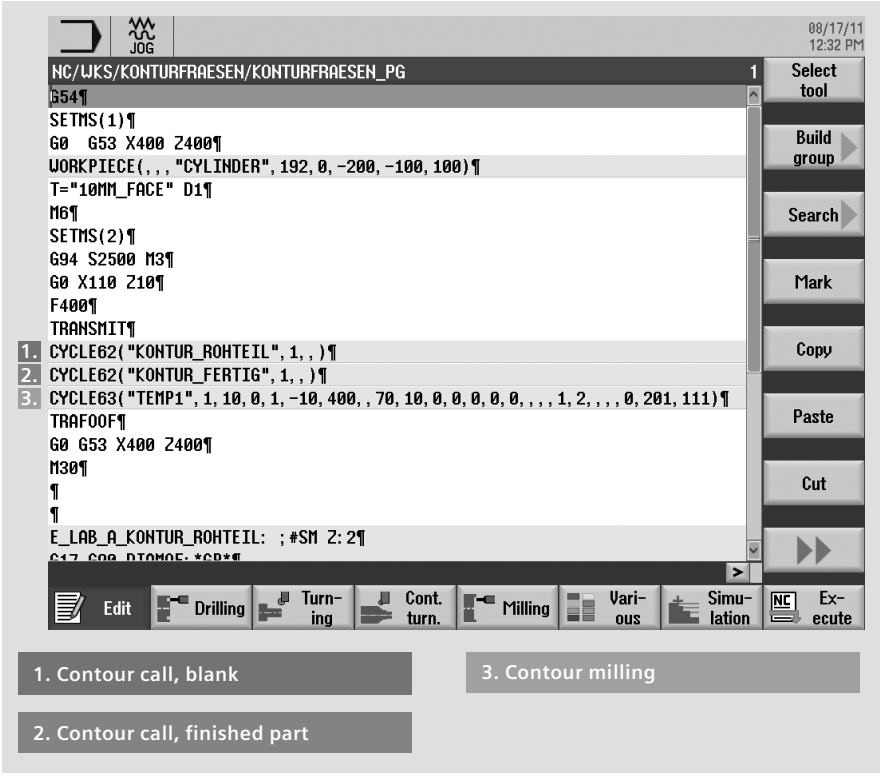
Define whether the contour is to be created on the end face or the peripheral surface.



The contours and the milling cycle are linked in the machining step editor.

3.8.3 Contour milling with programGUIDE

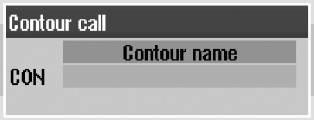
The contour call for the unmachined and finished part and the call for the "Mill spigot" milling cycle are explained using the following example program.



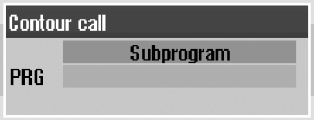
- For the machining on the end face, activate the TRANSMIT function; for machining operations on the peripheral surface, activate the TRACYL function. TRAFOOF deselects the transformations again.

You have the following ways of including a contour call in a programGUIDE program:

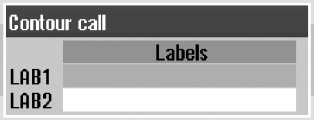
The contour is contained in the main program directly behind M30



The contour is contained in a subroutine



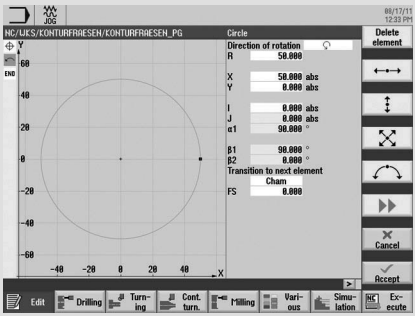
The contour is contained in the main program between two labels

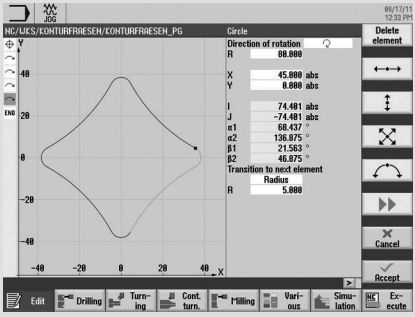


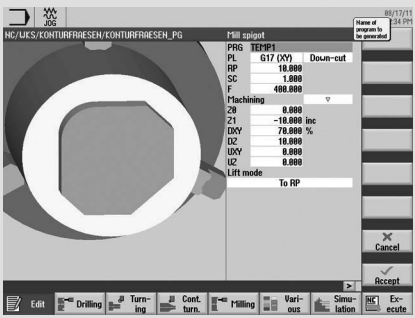
Call using labels in a subroutine

First insert the contour calls at the desired location in the program.

Note the above figure for contour calls.







Mill. → Cont. mill. → Contour

New contour

Create the unmachined and finished part contour in the contour calculator.

- You can enter as many as 256 geometric elements.

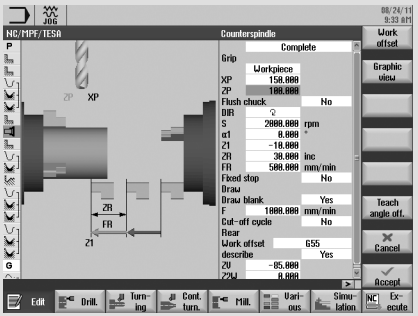
Mill. → Cont. mill. → Spigot

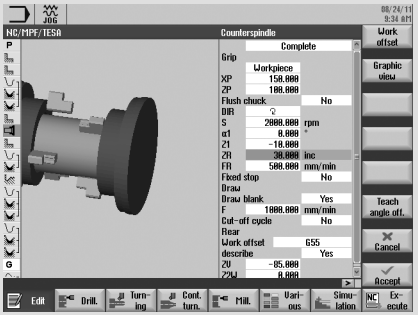
Enter the name for a program to be generated and select the machining plane. The program names can be chosen freely.

During the program execution, the G codes for the stock removal or the milling of contour elements using CYCLE952 (contour turning) or CYCLE63 (contour milling), including the residual material removal, will be generated in the program described above. Since these are normal NC programs, you can continue to use them, for example, to use the calculated code for a cycle-free and thus faster program.

3.9 Counterspindle

If your lathe has a counterspindle, you can machine workpieces using turning, drilling and milling functions on the front and rear faces without reclamping the workpiece manually.





Z2W is the position of the counterspindle on which machining is performed after the transfer. For the first workpiece, move as far right as possible!

ZV is the workpiece length dimension specified on the drawing. Caution! negative value.

The cycle and contour definition support is identical to that for machining with the main spindle; the Z axis values are simply mirrored.

A dialog provides support for the transfer of the main spindle to the counterspindle. You can choose between the following options for the transfer:

- Gripping: Grip the workpiece with the counterspindle
- Withdrawing: Withdraw the workpiece from the main spindle with the counterspindle
- Rear face: Move the workpiece with the counterspindle to the new machining position
- Complete: Grip, withdraw (optionally with cutting-off) and rear face steps
- Front face: Zero offset for machining the next front face (for bars)

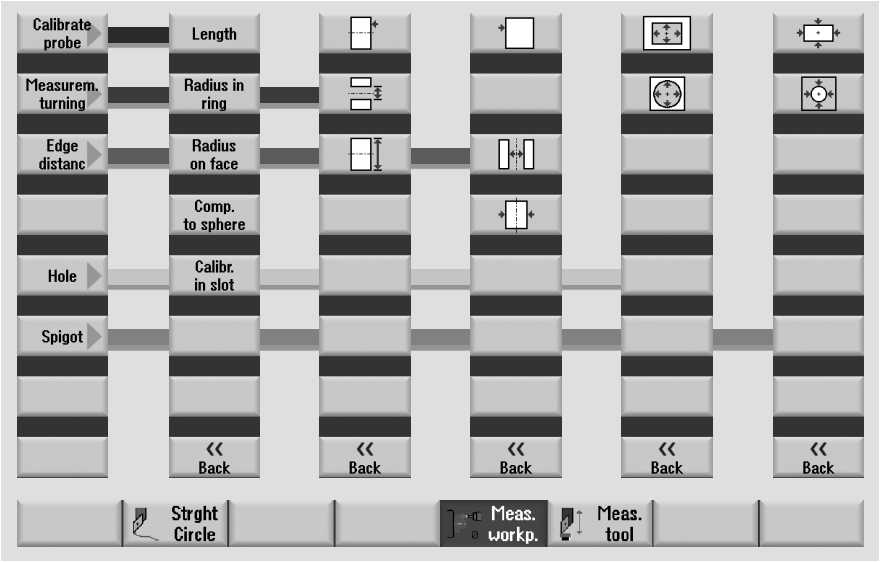
Refer also to the information about the setting data in the Tool measurement section.

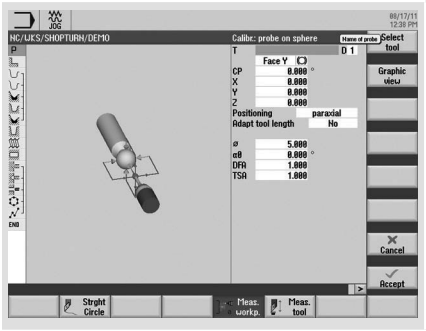
3.10 Measuring in the Automatic mode - in-process measurement

For measurement tasks in Automatic mode, powerful measuring cycles are available for both ShopTurn and programGUIDE. Input screens with dynamic help displays are used for convenient entry of the measuring parameters.

3.10.1 Measure workpiece

The figure below provides an overview of the measuring variants.



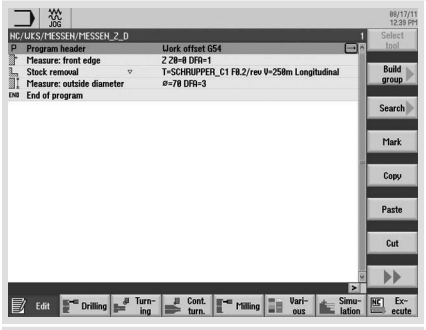


Meas. workp. | Calibrate probe | Comp. to sphere

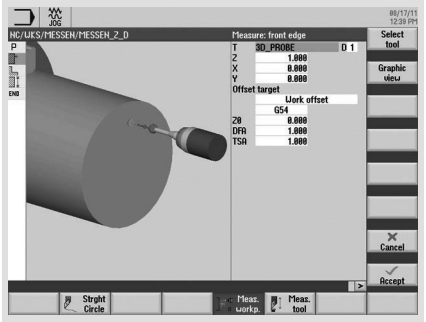
Using this measuring variant, a workpiece probe can be calibrated at any position in space.

3.10.2 Example

The following example for ShopTurn illustrates the use of the measuring cycles.

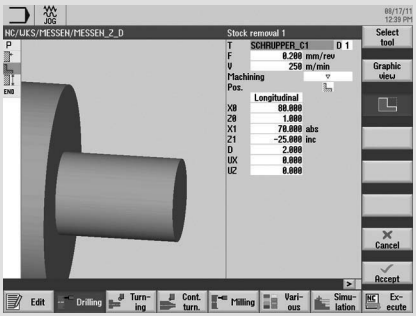


Sample program with measuring cycles in the machining step editor.



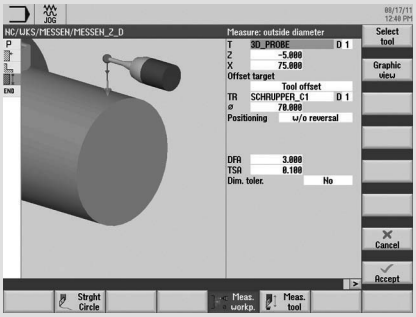
Meas. workp. | Measurem. turning

To automatically determine the workpiece zero at the program start and to write the zero offset, the “Measure front edge” cycle is used in this example.



Turn-ing | Stock removal

Turn a 70-mm diameter with the stock removal cycle.



Meas. workp. | Measurem. turning

Automatic measuring of the D70 diameter with the “Measure external diameter” cycle. Any deviations of the specified dimension are automatically written in the wear parameter of the tool and so compensated.

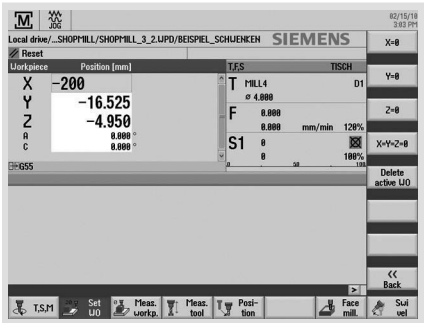
4 Milling technology

In addition to the guidelines for setup and programming, this chapter provides an overview of the technology cycles for drilling, milling and contour milling. You also obtain information on functions such as cylinder surface transformation, unmachined part input for simulation, swivel plane, high-speed settings and an overview of the cycles for the in-process measurement.

4.1 Setting up

In addition to the guidelines for setup and programming, this chapter provides an overview of the technology cycles for drilling, milling and contour milling. You also obtain information on functions such as cylinder surface transformation, unmachined part input for simulation, swivel plane, high-speed settings and an overview of the cycles for the in-process measurement.

4.1.1 Setting the zero point



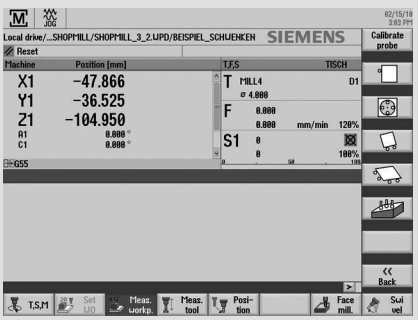
Set WO

In the “Set ZO” dialog box, you can write direct position values for individual axes in the actual value display.

The difference between the position value in the machine coordinate system (MCS) and the workpiece coordinate system (WCS) is saved in the active zero offset.

Requirement: A zero offset must be active.

4.1.2 Workpiece zero



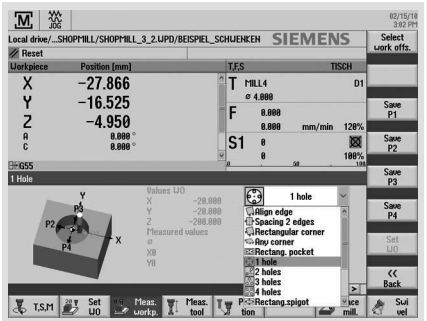
Meas. workp.

The reference point for programming a workpiece is always the workpiece zero. You are guided graphically for setting the workpiece zero.

You can perform the measurement using edge finders, dial gauges, reference tool or switching 3D button.

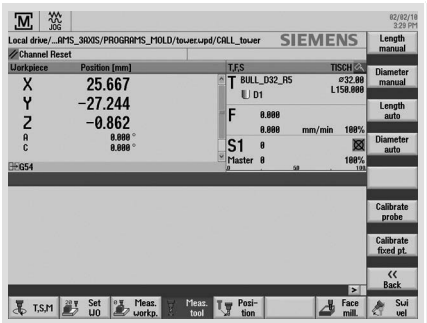
The following measuring variants are available for selection:

- Calibrate probe calibration length/ diameter
- Measuring point/edge/rib, align edge at 2 points
- Measure perpendicular/any corner
- Measure rectangular/circular pocket
- Measure/align at 2, 3 or 4 holes
- Measure rectangular/circular spigot
- Measure/align at 2, 3 or 4 spigots
- Align measuring plane in conjunction with multi-axis machines



You can select favorites for the measuring variant starting at the third softkey of the vertical softkey bar. To do so, press the softkey to which you want to assign the desired measuring variant. Open the selection list with any key and accept the desired measuring variant (see figure) with the Input key.

4.1.3 Tool measurement



Meas. tool

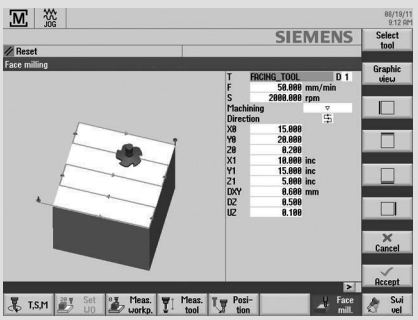
You are guided graphically for calibrating the tools (all tool types).

The tool compensation value can be directly determined in the machine set-up.

The following versions are available for determining the compensation values of the tools:

- Manually measure the length/diameter
- Automatically measure the length/diameter (cycle for switching measuring boxes)
- Fixed-point calibration of probe/adjustment t

4.1.4 Face milling

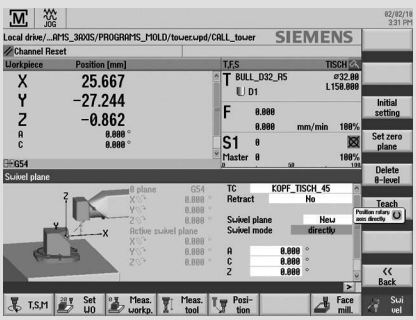


To prepare the blank for machining, a face milling cycle is provided in setup mode. You can select the tool directly from the tool list.

In addition to the input of the feedrate and the spindle speed or cutting speed, you can specify the machining strategy and direction as well as the machining limitation.

Because the input values are retained even after the power has been switched off and on, the user can restart the face milling operation with minimum effort.

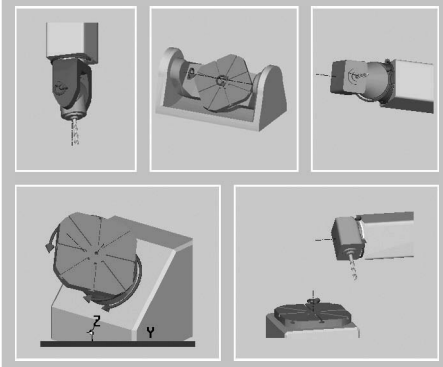
4.1.5 Swiveling in JOG



The swivel function (JOG) gives you the capability of swiveling a machining plane as required:

- Machining inclined surfaces
- Measure with approached axes

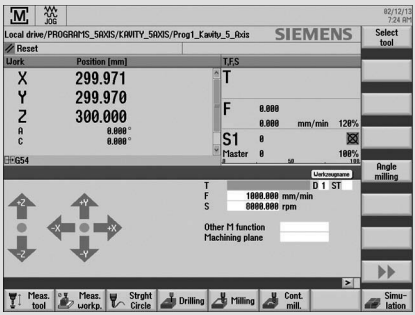
Animated elements provide support for the direction selection:



You can swivel directly and axis-by-axis, including an optional coordinate rotation.

Note: Support is provided for all common machine kinematics!

4.2 Manual Machine



Note: Overviews of the drilling, milling and contour-milling cycles are contained in Section 4.3, Programming.

All important machining actions can be performed in the Manual Machine operating area without an associated part program needing to be created.

The following functions are available:

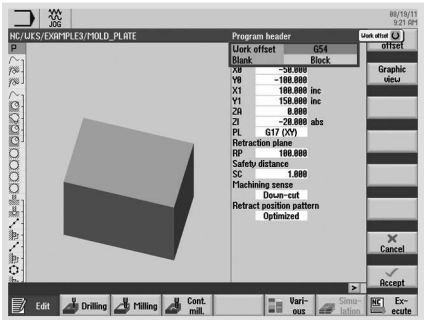
- Measure tool
- Traverse axes
- Set zero offset (ZO)
- Turn straight line / circle
- Drill, such as centering, deep-hole drilling, thread cutting
- Mill, such as face milling, pocket milling, multi-edge stud milling
- Mill contours

User-friendly input screens with help screens and animated elements are available for parameterizing the cycles.

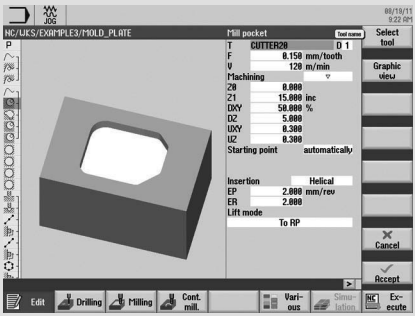
4.3 Programming

4.3.1 ShopMill machining step programming

Each program line in the ShopMill machining step editor represents a technological machining step (example: face milling, thread milling, ...) or geometric data required for the machining steps (position patterns or contours). You do not need any knowledge of DIN/ISO for entering the individual machining steps. All required technical and geometric parameters are entered in cycle screen forms. If necessary you can also enter DIN/ISO blocks and control functions in the machining step editor.



In the program header, you define program parameters such as the blank and the zero offset.

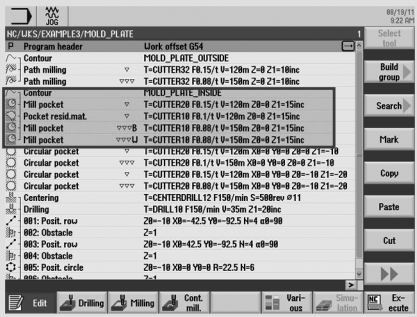


The technology data such as the tool call, the feedrate via mm/min or feed per tooth (including automatic conversion) and the speed or the cutting rate (including automatic conversion) is integrated directly in the cycle.

You can also enter parameters in the input fields of the cycles.

During the programming, you can use the "Graphic view" function to dynamically display the workpiece.

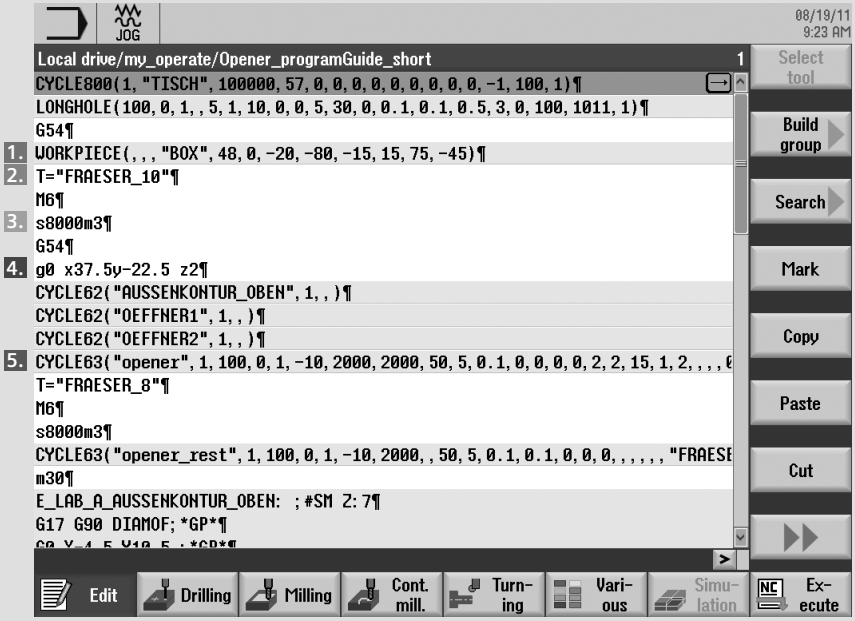
Note: The conversion when changing between feedrate using mm/min and feed per tooth and between speed and cutting rate is performed automatically.



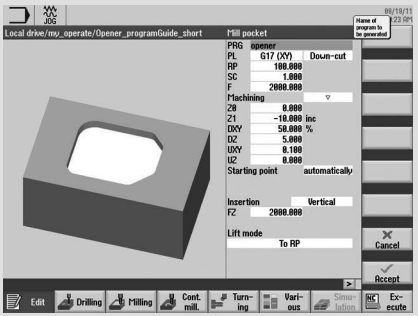
The example shows the linking of the contour with the contour milling cycles.

4.3.2 programGUIDE

In programGUIDE, in additional to the DIN/ISO editor, the technology cycles are also available similar to the ShopMill machining step programming. This ensures an optimum integration. The calls for tool, feedrate and cutting rate, etc. are also entered in the DIN/ISO editor. The following figure shows the structure of a program.



1. Program header/blank input
2. Tool call
3. Technology block
4. Approach
5. Cycle call

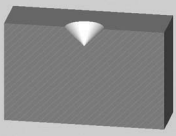


The example shown here uses the “Mill pocket” technology cycle from above (see CYCLE63 cycle call).

4.4 Drilling cycles

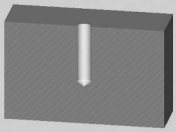
An overview of the drilling cycles in ShopMill and programGUIDE with application tips is shown below.

Centering



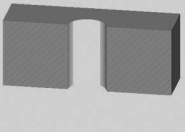
Cycle for centering at any positions/ position patterns

Drilling Reaming



Drilling

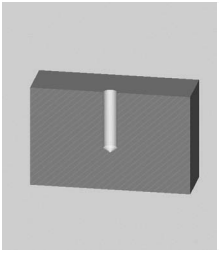
Cycle for drill holes in an infeed with drilling depth based on the shank or tip at any positions/position patterns



Reaming

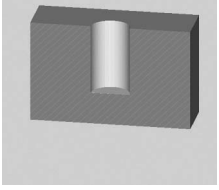
Cycle for reaming at any positions/ position patterns

Deep hole drilling



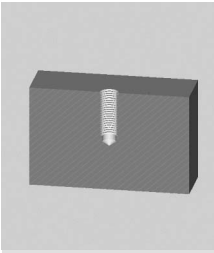
Cycle for drill holes with drilling depth based on the shank or tip at any positions/position patterns

Boring



Cycle for boring drill hole diameters with special boring tools at any positions/ position patterns

Thread



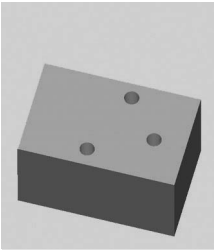
Tapping

Cycle for tapping with and without compensating chuck at any positions/position patterns

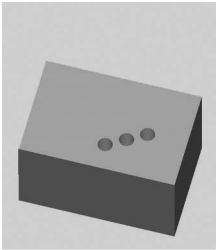
Drl+thrd milling

Cycle for producing a thread with special tapping drill and thread milling cutters at any positions/ position patterns

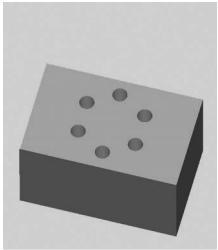
Positions



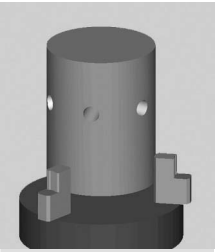
Cycle for the input of free positional data



Cycle for entering a position pattern - hide line, grid and frame, incl. position



Input of a position pattern - hide graduated circle, full circle and position on rotary axes, incl. position

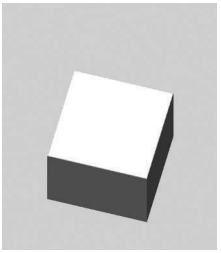


Display rotary axis in the position patterns

4.5 Milling cycles

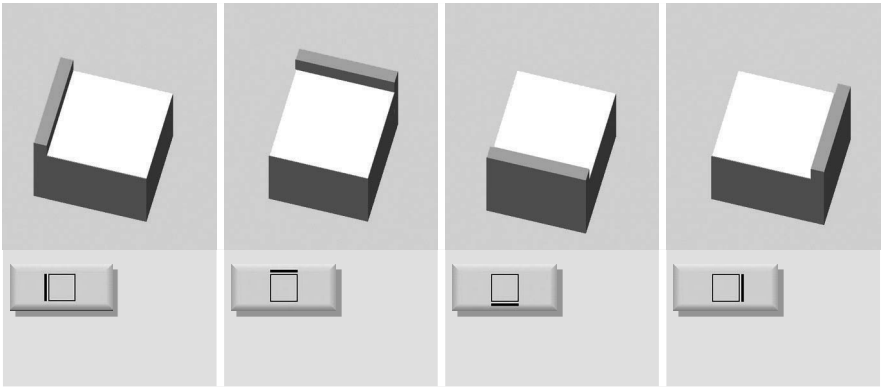
An overview of the milling cycles in the ShopMill machining step programming and programGUIDE with application tips is shown below.

Face milling

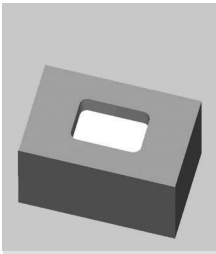


You can use this cycle to face mill any workpiece. A rectangular surface is always machined.

You can optionally select the limits using the appropriate softkeys. You can select a maximum of three limits in combination, otherwise use the pocket milling cycle.



Pocket



Rectang. pocket

Cycle to produce any circular pocket at any positions or position patterns.

Machining methods:

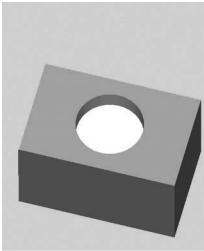
Roughing, finishing, chamfering

Infeed:

Plane-by-plane, helical

Insertion strategies:

Vertical, helical, predrilled



Circular pocket

Cycle to produce a circular pocket at any positions or position patterns.

Machining methods:

Roughing, finishing, chamfering

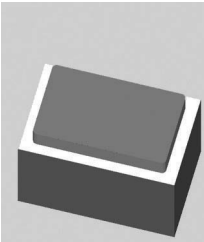
Infeed:

Plane-by-plane, helical

Insertion strategy:

Vertical, helical.

Multi-edge spigot



Rectang. spigot

Cycle for producing a rectangular spigot at any positions/position patterns

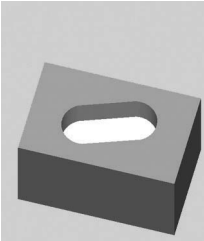
Circular spigot

Cycle for producing a circular spigot at any positions/position patterns

Multi-edge

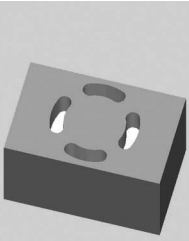
Cycle for producing a polyhedron at any positions/position patterns

Slot



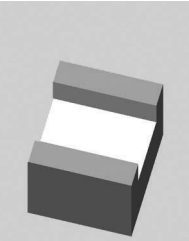
Longit. slot

Cycle for producing longitudinal grooves with vertical, helical and oscillating insertion strategies at any positions/position patterns



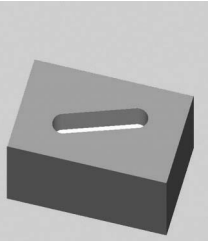
Circumfer. slot

Cycle to produce any number of circumferential grooves on a graduated circle or full circle



Open slot

Cycle for producing open grooves with the vortex milling or plunge cutting technologies at any positions/position patterns



Elongated hole

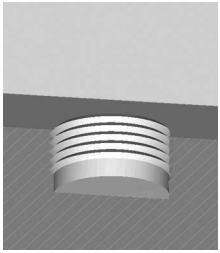
Cycle for producing elongated holes

Machining types:

Plane-by-plane, oscillating

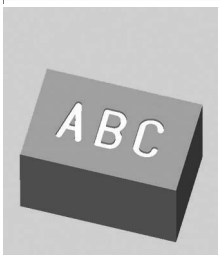
■ Only in programGUIDE

Thread milling



Cycle for producing internal or external threads with the milling technology at any positions/position patterns

Engraving



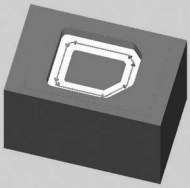
Engraving cycle for characters and numbers with any alignment

4.6 Contour milling

This section provides an overview of the contour milling cycles in the ShopMill machining step programming and programGUIDE. The contour milling with ShopMill machining step programming and programGUIDE is explained using an example.

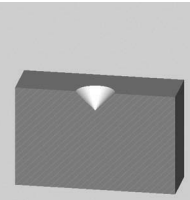
4.6.1 Overview of the milling cycles for the contour machining

Path milling



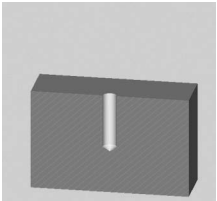
Cycle for the machining of arbitrary contours, incl. radius compensation and approach/departure movements

Rough drill



Centering

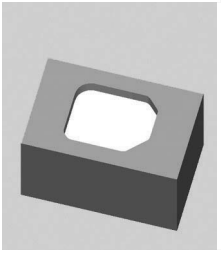
Cycle for centered drilling for the specific predrilling of a start hole for pocket milling



Rough drill

Cycle for predrilling for the specific predrilling of a start hole for pocket milling

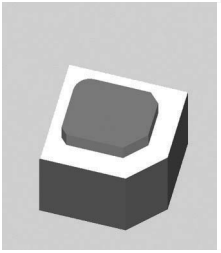
Pocket or Pocket res.mat.



Cycle for machining any pocket contours, with helical and oscillation insertion strategies; it can be combined with automatic residual material detection

Note: Manual Machine does not support residual material detection.

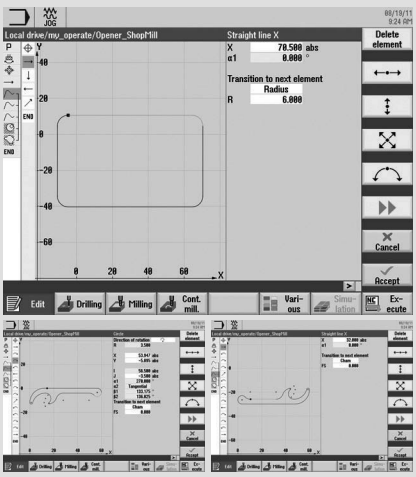
Spigot or Spigot res. mat.

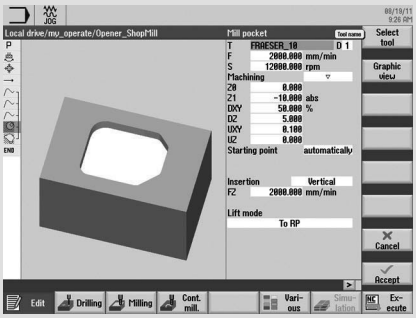


Cycle to produce any spigot contours with a defined initial blank; it can be combined with automatic residual material detection

Note: Manual Machine does not support residual material detection.

4.6.2 Contour milling with ShopMill machining step programming





Cont. mill. | New contour

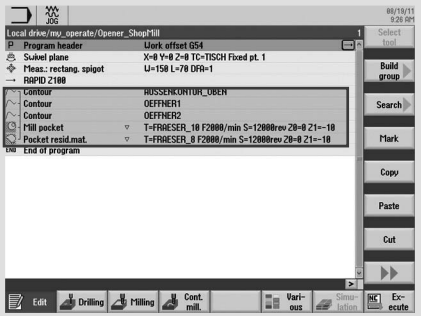
You can define open contours or closed contours (pockets, islands, spigots) and machine them with contour milling cycles.

The example shows the creation of a contour pocket and two contours for the islands. This allows you to easily cut the surface/pocket outside the islands.

- You can enter as many as 256 geometric elements.

Cont. mill. | Pocket

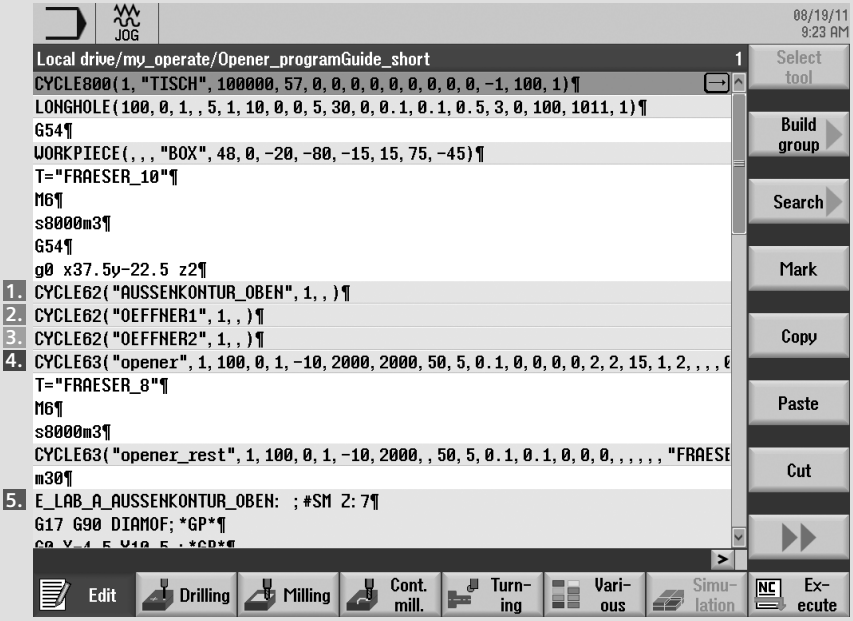
In the “Mill pocket” technology cycle, you create the tool, the feedrate, the spindle speed, as well as the complete machining and insertion strategy, etc.

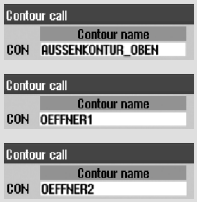


The contours and the “mill pocket” and “pocket residual material” milling cycles are linked in the machining step editor.

4.6.3 Contour milling with programGUIDE

The following example program explains the contour call for the pocket and the islands and the call of the “Mill pocket” milling cycle.

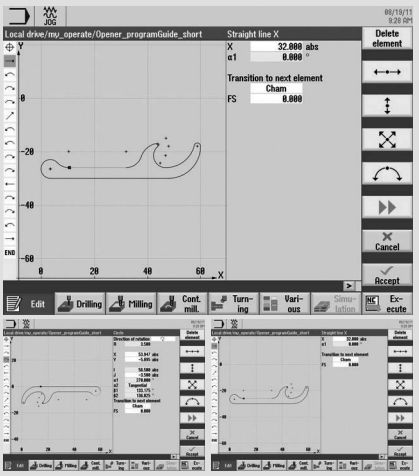




Cont. mill. → Contour → Contour call

First insert the contour calls at the desired location in the program.

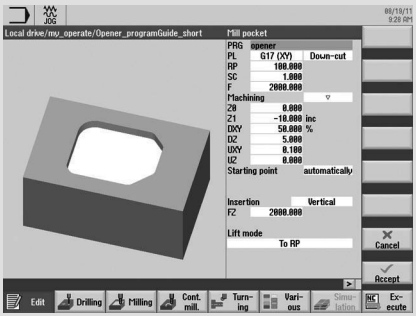
Note the above figure for contour calls.



Cont. mill. → Contour → New contour

Create the pocket and the island contours in the contour calculator.

- You can enter as many as 256 geometric elements.

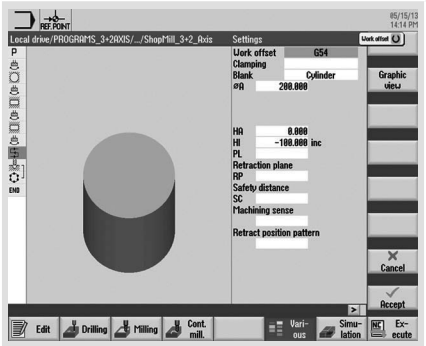


Enter the name for a program to be generated and select the machining plane. The program names can be chosen freely.

During the program execution, the G codes for the milling of contour elements using CYCLE952 (contour turning) or CYCLE63 (contour milling), including the residual material removal, will be generated in the program described above. Since these are normal NC programs, you can continue to use them, for example, to use the calculated code for a cycle-free and thus faster program.

4.7 Miscellaneous

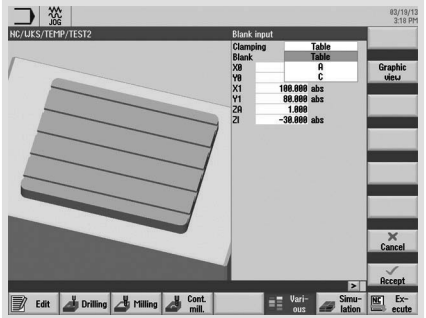
The “Miscellaneous” softkey provides the following functions



Settings ShopMill only

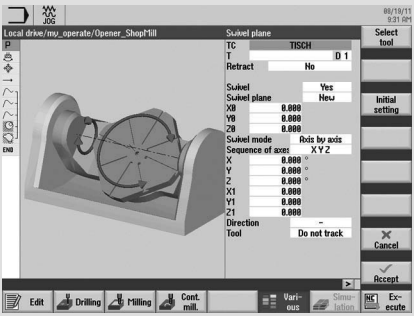
All parameters defined in the program header, with the exception of the dimension unit, can be changed at any location in the program.

The settings in the program header are modal, i.e. they remain active until they are changed.



Blank programGUIDE only

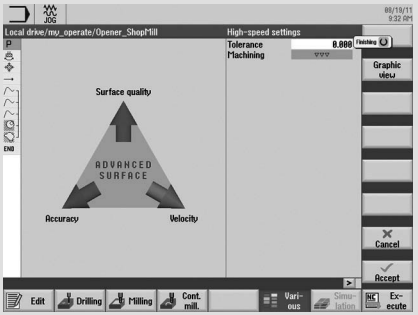
The “Blank input” cycle can be used in programGUIDE to enter the blank data. You select here the form of the raw part and the clamping, etc.

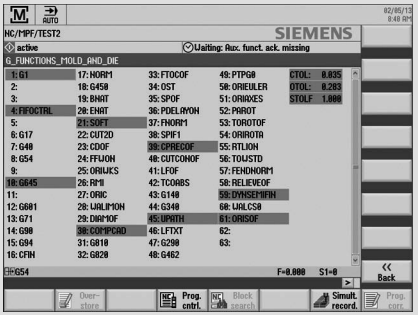


Swivel plane

The CYCLE800 swivel cycle is used to swivel to any surface in order to either machine or measure it. In this cycle, the active workpiece zeros and the zero offsets are converted to the inclined surface taking into account the kinematic chain of the machine by calling the appropriate NC functions, and rotary axes (optionally) positioned.

To set and align the tool, call the “Swivel tool” function.





HighSpeed Settings

Machining of free-form surfaces involves high requirements regarding velocity, precision and surface quality.

You can achieve optimum velocity control depending on the type of processing (roughing, rough-finishing, finishing) very simply with the "HighSpeed Settings" cycle.

The "HighSpeed Settings" function is used to specify the tolerance of the machining axes and the machining type. You can choose between the following technological machining types:

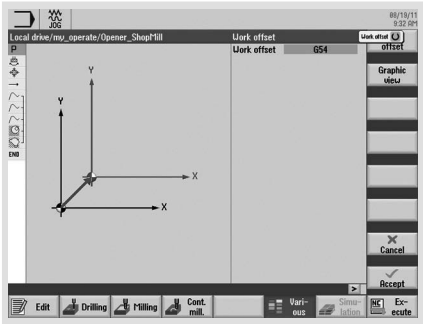
- Finishing
- Rough-finishing
- Roughing
- Deselection (default setting)

All G functions

Note: Depending on the Advanced Surface option, you can display in the Automatic operating mode an overview of all G-Code functions for the mold making.

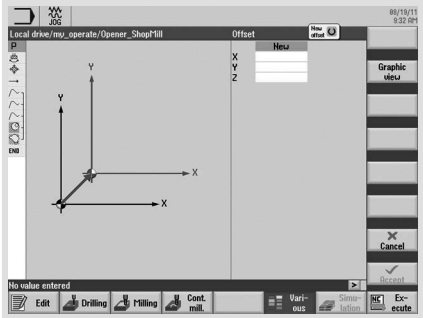
Transformations

You can use the following functions under ShopMill:



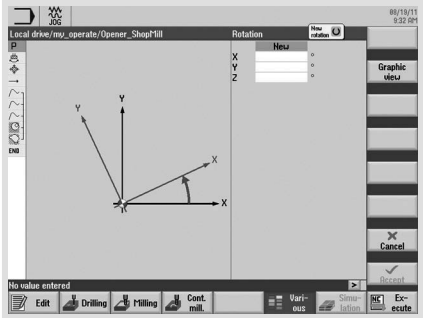
Work offset

You can call zero offsets (G54, etc.) from any program.



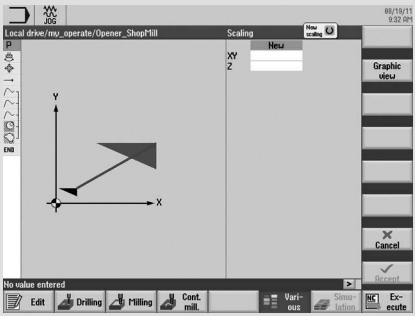
Offset

For each axis, you can program an offset of the zero point.



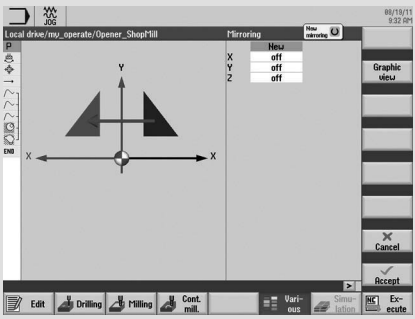
Rotation

You can rotate every axis through a specific angle. A positive angle corresponds to counterclockwise rotation.



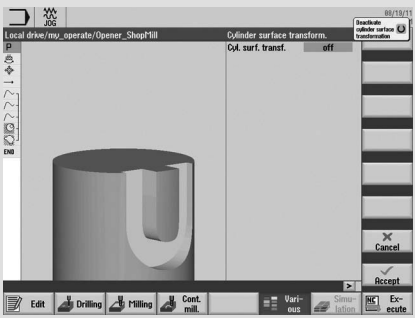
Scaling

You can specify a scale factor for the scaling in X/Y/Z. The programmed coordinates are then multiplied by this factor.



Mirroring

You have the option of mirroring all axes. Enter the axis to be mirrored in each case.



Cylinder surface

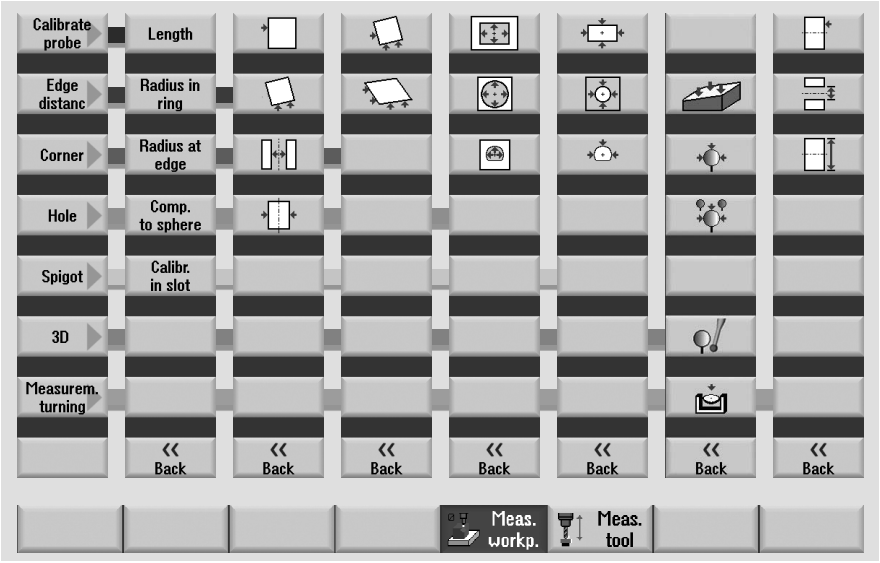
You can use this function to mill parallel-sided grooves with and without groove side offset.

4.8 Measuring in the Automatic mode – in-process measurement

For measurement tasks in automatic mode, powerful measuring cycles are available for both ShopMill and programGUIDE. Input screens with dynamic help displays are used for convenient entry of the measuring parameters.

4.8.1 Measure workpiece

The figure below provides an overview of the measuring variants.

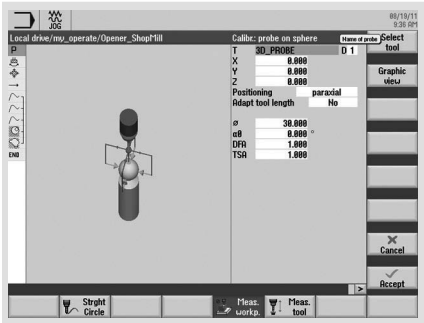


Measuring cycles for workpiece probe for the milling technology:

- Calibration – length, radius in ring, radius at edge, trim to sphere, alignment in groove
- Measure edge – point/surface, align edge, groove/rib distance,
- Measure corner – right-angled corner with 3 points or any corner with 4 internal/external points
- Measure drill holes – circle segment with 4 or 3 points – rectangular pocket
- Measure spigots – circle segment with 4 or 3 points – rectangular
- Measure 3D – align plane - sphere

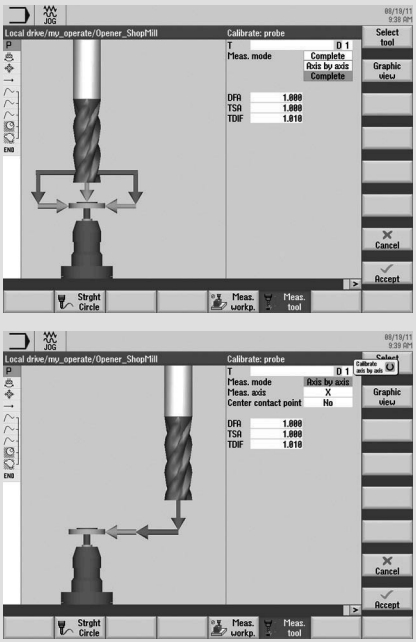
Applications: Measuring only, zero offset, tool offset

Note: The CYCLE 995 and CYCLE 996 measuring cycles are available in conjunction with the “Measure kinematics” option.



Using this measuring variant, a workpiece probe can be calibrated at any position in space.

4.8.2 Measure tool



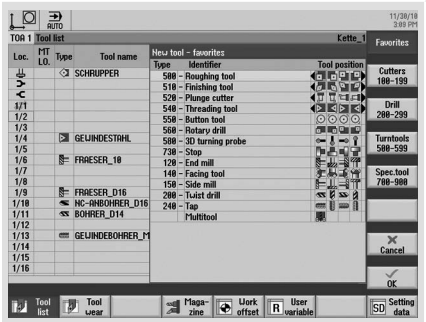
The following measuring cycles for measuring with measuring box are available for selection:

- Calibration: Length, radius, complete
- Measure: Length and radius
- Measure with rotating or stationary spindle
- Tool offset in geometry or wear
- Measure in machine MCS or workpiece WCS coordinate system

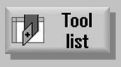
5 Multitasking (with SINUMERIK 840D sl)

This chapter contains useful information regarding the machining of workpieces on multitasking machines. This includes the extended tool management, multi-axis kinematics, turning functionality for milling (milling-turning technology) as well as milling functionality for turning (turning-milling technology).

5.1 Tool management



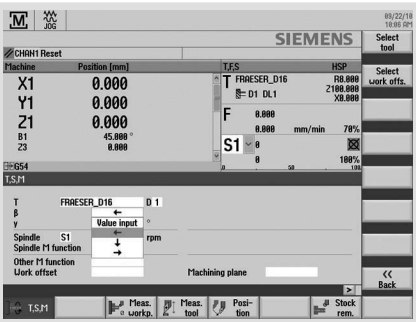
or



For multitasking machines - for turning-milling or milling-turning - extended tool management for turning and milling tools is available. You can also deploy complex tools such as multitools. Additional parameters, such as distance description using location number or angle and different tool types for each location, are available for multitools. All tools are represented as icons.

5.2 Turning-milling technology with ShopTurn and programGUIDE

5.2.1 Swivel axis in TSM mode

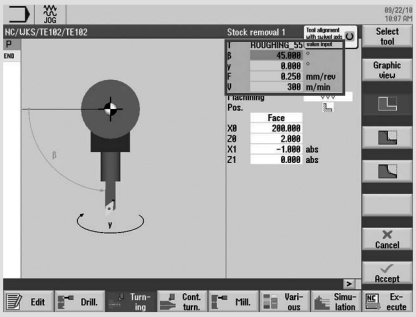


T,S,M

In TSM mode, you can control the swivel axis by entering a value or with the direction arrows.

You can approach the β angle (B axis position) and the γ angle (tool spindle position) using input parameters, e.g. to manually set the zero point or calibrate the tool.

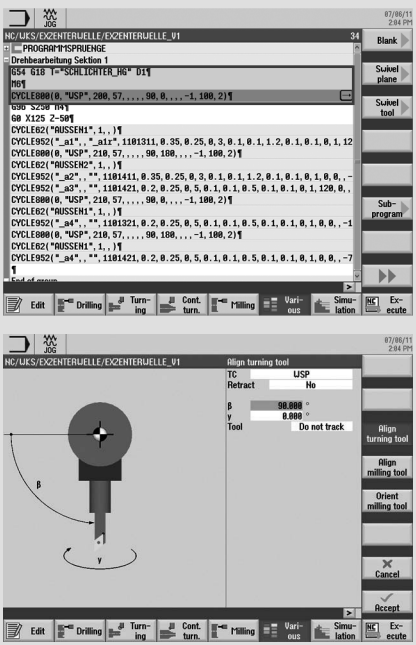
5.2.2 Turning with ShopTurn and B axis



Turn-ing e.g. Stock removal

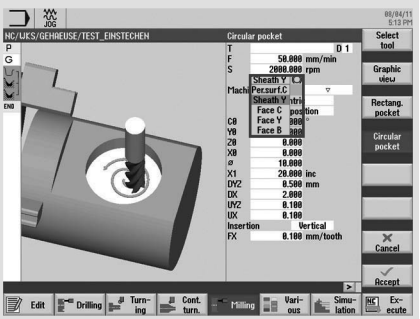
In all turning cycles under ShopTurn, enter the setting angle of the B axis (β) and the positioning angle of the tool spindle (γ) directly in the technology area of the cycle. Use the γ angle to specify whether a tool operates as standard or overhead.

5.2.3 Turning with programGUIDE and B axis



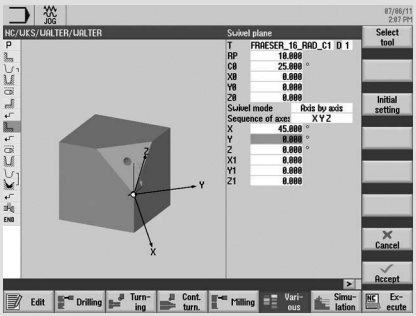
You can use the “Align turning tool” function to align the turning tool in the CYCLE800. You can then program the turning.

5.2.4 Milling with ShopTurn and B axis



All milling cycles on the lathe are identical with those in the ShopMill machining step programming.

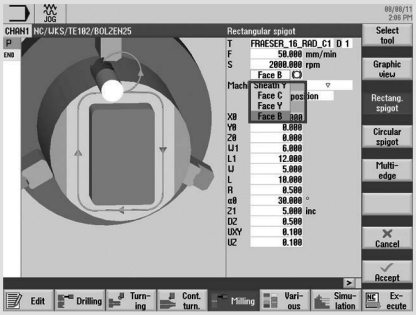
You define in the cycle the selection of the plane when milling on the end face or the peripheral surface.



Vari-ous → Swivel plane

For milling on swiveled planes, use the CYCLE800 swivel cycle on the turning machine.

Note: The first choice is normally YXZ, according to the hardware, for the axis sequence. Turning around the Y axis!

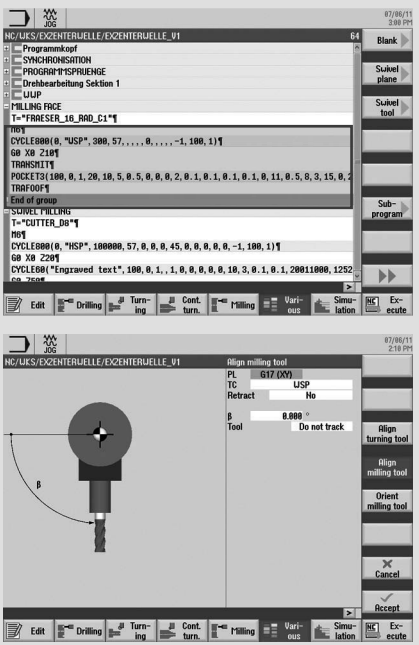


Mill. e.g. Spigot → Rectang. spigot

In the milling cycle, then select the “Face B” plane for the combination with swiveling.

Swivel plane
Rectang.spigot

5.2.5 Milling with programGUIDE and B axis

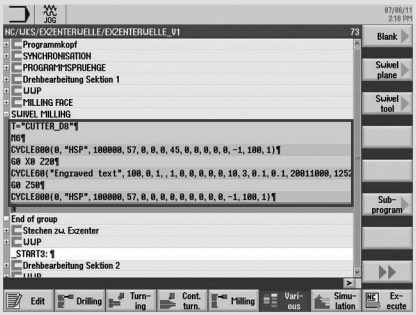


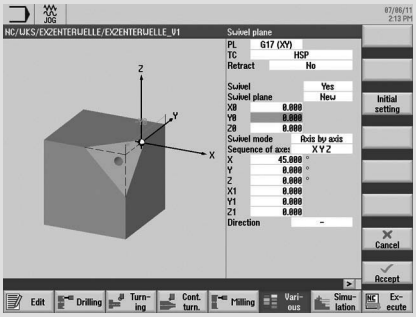
Vari-ous → Swivel tool → Align milling tool

You use the “Align milling tool” function to position the milling tools for machining on the end face or peripheral surface.

Note: The alignment of the milling tool causes, however, only a swivel of the B axis and the calculation of the tool tip, but no swivel of the plane!

You can then program on the end face, for example with TRANSMIT.



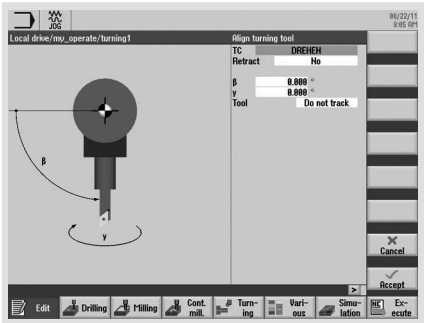


To program on the swiveled plane, use the “Swivel plane” function to select the CYCLE800. In this case, a new machining plane will be created on which you can now create all milling functionality, also with the help of cycles.

5.3 Milling-turning technology with programGUIDE

This section provides an overview of the technology cycles in programGUIDE for turning and contour turning with the milling technology. The contour turning is explained using an example. To check the programming, you can also use the “Simulation” function for the milling-turning.

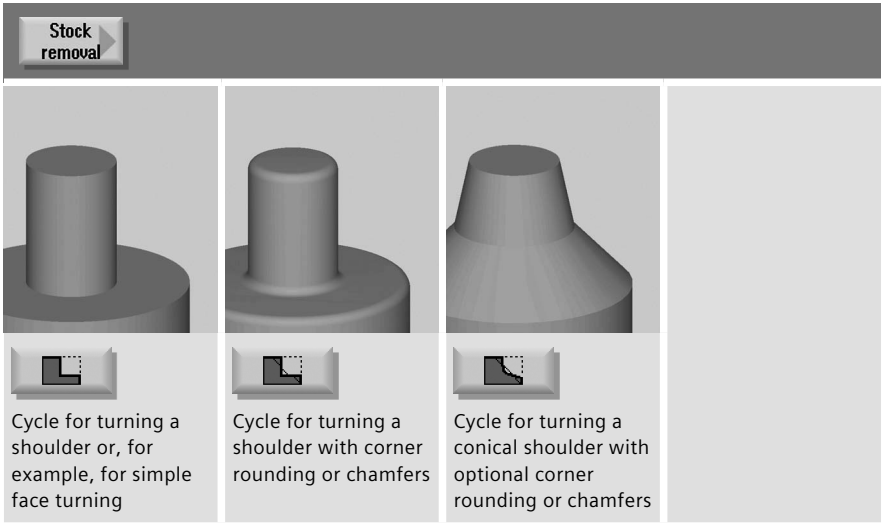
5.3.1 Align turning tool



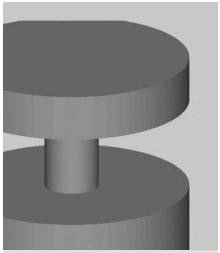
The “Align turning tool” function provides support for the milling/turning centers with multi-axis kinematics. The position and orientation of the turning tool can be changed by rotating the swivel axis (β angle) and rotating at the tool axis (γ angle).

5.3.2 Overview of the turning cycles in the milling technology

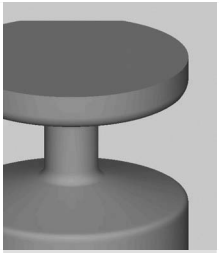
An overview of the turning cycles in programGUIDE with application tips follows.



Groove



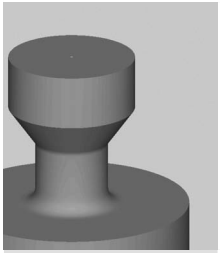
Grooving cycle without rounding



Extended grooving cycle with optional rounding or chamfers and additional conicity

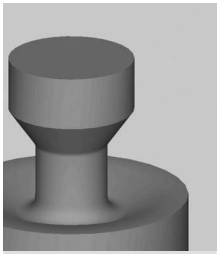
Extended grooving cycle on conical turned parts

Undercut



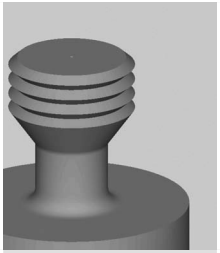
Undercut form E

Cycle for external or internal undercuts according to the standard E



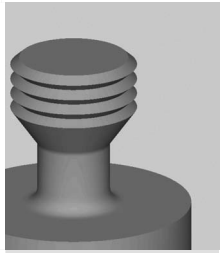
Undercut form F

Cycle for external or internal undercuts according to the standard F



Undercut thrd DIN

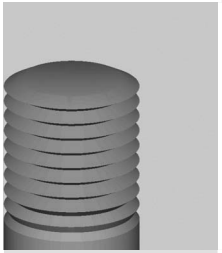
Cycle for external or internal thread undercuts according to the DIN standard



Undercut thread

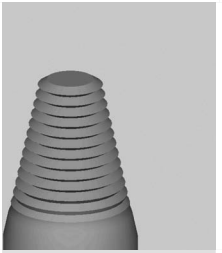
Cycle for external or internal thread undercuts without any standard (freely programmable)

Thread



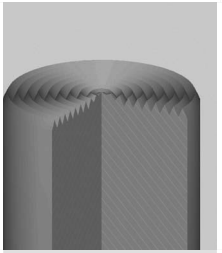
Thread long.

Thread cycle for longitudinal thread



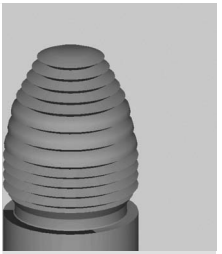
Thread taper

Thread cycle for tapered threads



Thread face

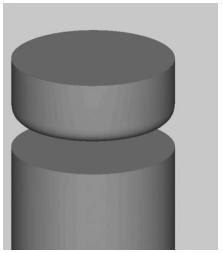
Thread cycle for face thread



Thread chain

Thread cycle for chaining different threads in a single operation

Cutoff

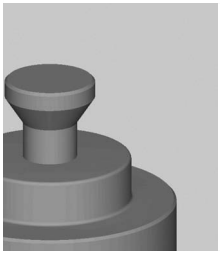


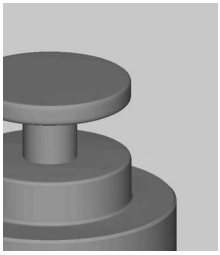
Cycle for parting rod parts

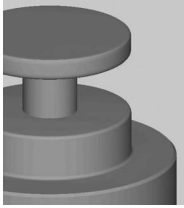
5.3.3 Contour turning in the milling technology

This section provides an overview of the cycles for contour turning with programGUIDE in the milling technology. The contour turning with programGUIDE is explained as an example.

5.3.3.1 Overview of the contour turning cycles in programGUIDE

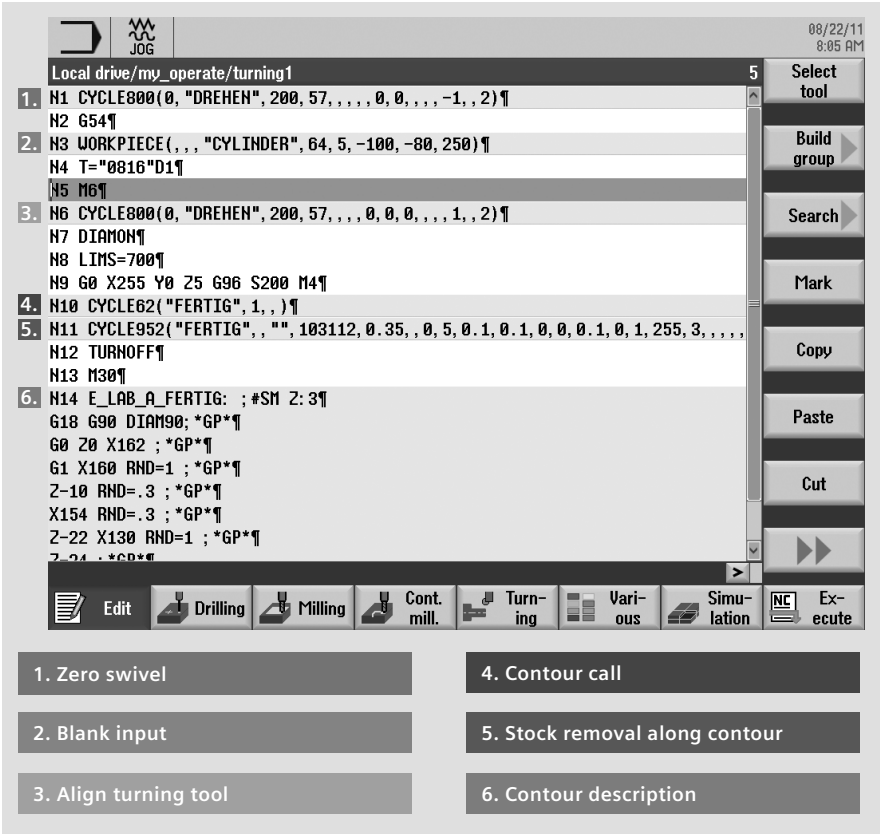
Stock removal or Cut resid stock	
	Stock removal cycle for machining any contours with the stock removal technology; it can be combined with automatic residual material detection

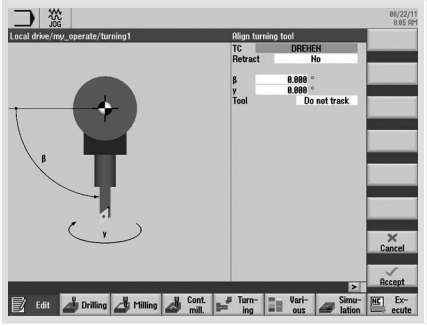
Grooving or Groove resid.	
	Plunge-turning cycle for machining any contours with the plunge-turning technology; it can be combined with automatic residual material detection

Plunge turning or Plunge turn.resid.	
	Plunge-turning cycle for machining any contours with the plunge-turning technology; it can be combined with automatic residual material detection

5.3.3.2 Contour turning with programGUIDE in the milling technology

The contour call and calls of the stock removal and residual stock removal cycles are explained using the following example program.





Vari-ous | Swivel tool | Align turning tool

The “Align turning tool” function provides support for the milling/turning centers with multi-axis kinematics. The position and orientation of the turning tool can be changed by rotating the swivel axis (β angle) and rotating at the tool axis (γ angle).

You have the following ways of including a contour call in a programGUIDE program:

The contour is contained in the main program directly behind M30

Contour call	
	Contour name
CON	

The contour is contained in a subroutine

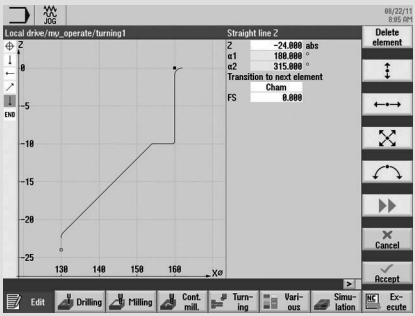
Contour call	
	Subprogram
PRG	

The contour is contained in the main program between two labels

Contour call	
	Labels
LAB1	
LAB2	

Call using labels in a subroutine

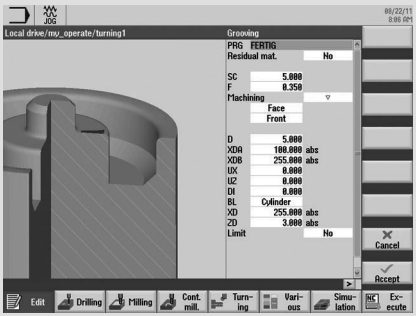
Contour call	
	Labels in subprogram
PRG	
LAB1	



First insert the contour call at the desired location in the program. Note the above figure for contour calls.

Create a new contour in the contour calculator.

- Create a new contour in the contour calculator.



Enter for grooving the name of a program to be generated (PRG). The program names can be chosen freely.

During the program execution, the G codes for the stock removal or the milling of contour elements using CYCLE952 (contour turning) or CYCLE63 (contour milling), including the residual material removal, will be generated in the programs described above. Since these are normal NC programs, you can continue to use them, for example, to use the calculated code for a cycle-free and thus faster program.

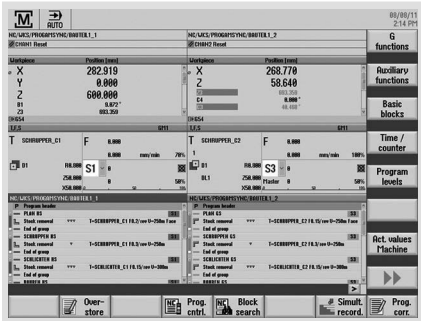
5.4 Multi-channel machining

You can use programSYNC to easily synchronize and visualize multi-channel machining. To this end, proceed as follows:

- Create the structure of the part programs with the help of blocks
- Program the individual machining steps
- Simulate the part programs
- Load the part programs (for each channel or spindle)

The following sections describe the programming under programSYNC.

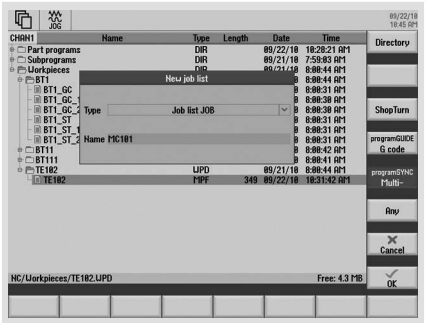
5.4.1 Machine basic screen



You can choose between a one- or two-channel view as machine basic screen.

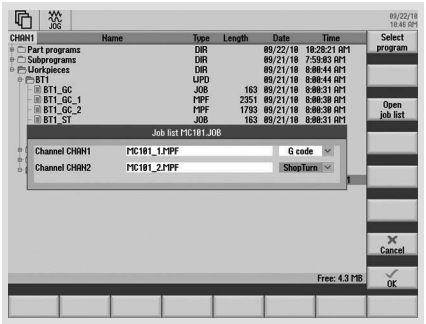
The active channel is color-highlighted for the two-channel view.

5.4.2 programSync multi-channel

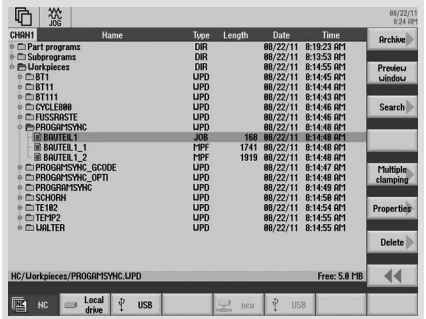


New programSYNC Multi-

To structure multi-channel programs, use the "programSYNC multi-channel" softkey to create a job list.



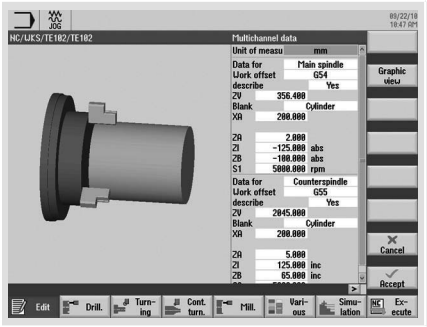
You can then assign any ShopTurn or G code programs to the appropriate channels.



Open or ▶

To edit the assigned programs, open the job list (*.JOB) in the program manager. The programs open together in the dual editor.

5.4.3 Multi-channel program data

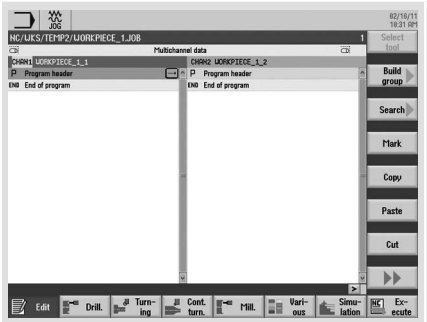


Edit

Open the program header in the editor.

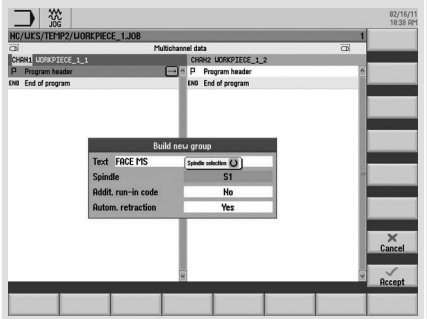
The program header shows a standard overview of the multi-channel data for both ShopTurn and programGUIDE.

5.4.4 Dual editor



Edit

The dual editor supports you for the concurrent creation of the programs for both channels.

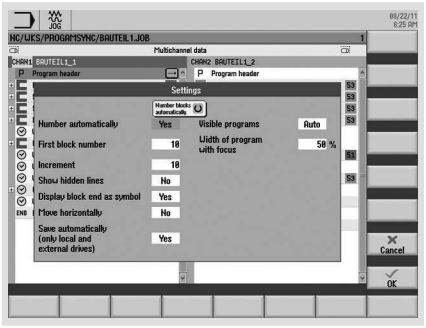


Edit

Build group

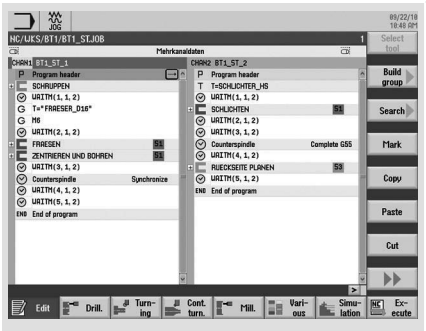
The block generation is the basis of the multi-channel programming.

Generate the program structure with the help of blocks. Then enter the appropriate program code within the respective blocks.

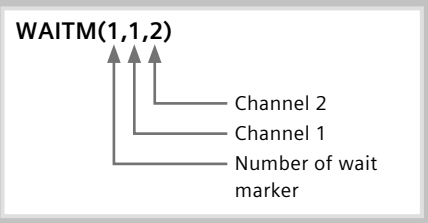


You can configure the editor. You can specify how many programs should be visible, etc.

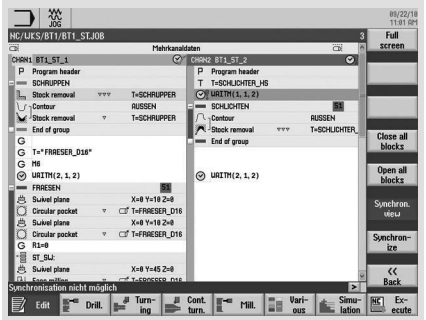
5.4.5 Time synchronization



You can insert wait markers for the chronological matching of the machining blocks in the various channels, e.g. WAITM(1,1,2).



5.4.6 Synchronous view



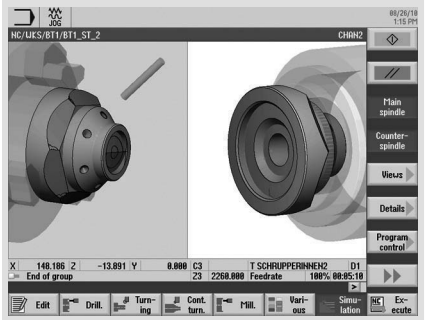
You can use the “Synchronous view” function to check the wait markers.

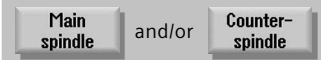
The wait markers are compared with each other to produce a clear program presentation.

The result is visualized:

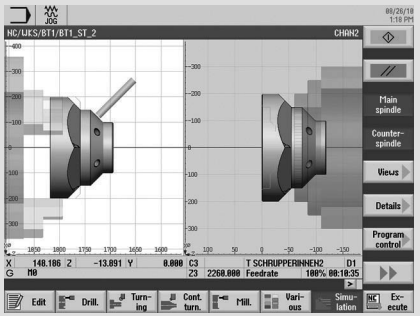
- Without conflict/fault
- Conflict/fault

5.4.7 Simulation





3D simulation with 3-level view and volume model of the finished part.



Simu-lation

Main spindle and/or Counter-spindle

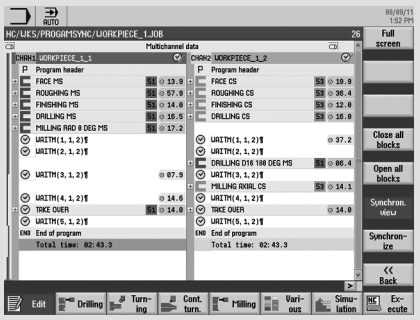
Views Side view

Simulation in the side view.

Ex-ecute Simult. record. CYCLE START

You can also use the simultaneous recording function in Automatic mode.

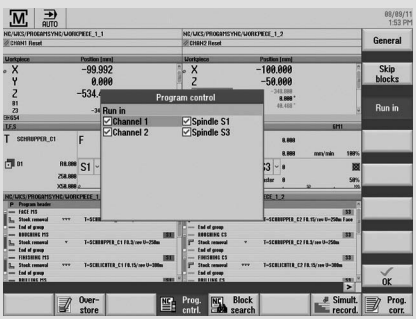
5.4.8 Determining the machining time, optimization



Edit

After a successful simulation, the system determines the execution time of the machining steps. This allows you to optimize the channels with each other.

5.4.9 Program control



In Automatic mode, you can choose between different approach variants - by channel or by spindle.

6 Appendix

6.1 G-Code

- **G code**
  G-code according to DIN66025 and in ISO dialect mode
- **G Functions**
  G0, G1, G2, G71 ...
- **Language commands (extended G functions)**
  CIP, SOFT, FFWON ...
- **Frame operations (programmable work offsets)**
  The workpiece coordinate system can be shifted, scaled, mirrored or rotated with the commands TRANS, SCALE, MIRROR, ROT.
- **User variables**
  The user can define his own variables by name and type.
- **System variables**
  System variables can be read/written in all programs. They enable access to work offsets, tool offsets, axis positions, measured values, controller states, etc.
- **Arithmetic operations**
  Arithmetic operations are available to combine the variables:
  - Computational operations + - * / sin cos exp etc.
  - Logical operations == <> >= etc.
- **Program control structures**
  BASIC-style language commands are available for flexible programming of the user cycles: IF-ELSE-ENDIF, FOR, CASE ...Shortcuts

6.2 Shortcuts

Functions – Softkeys – Shortcut keys

Control key:	
CTRL + P	For screenshots – Storage location: Commissioning (password) – System data – HMI data – Logs – Screenshots
CTRL + L	Language switchover
CTRL + C	Copy
CTRL + X	Cut
CTRL + V	Paste
CTRL + Y	Redo (editor functionality)
CTRL + Z	Undo – max. five lines in the editor (editor functionality)
CTRL + A	Select all (editor functionality)
CTRL + NEXT WINDOW	Go to start of program
CTRL + END	Go to end of program
CTRL + ALT + S	Save complete archive – NCK/PLC/drives/HMI
CTRL + ALT + D	Backup log files on USB or CompactFlash card
CTRL + E	Control energy

CTRL + M	Maximum simulation speed
CTRL + F	Search in all screen forms Wildcards "?" and '"*" can be used in search screen forms. “?” stands for any character, “*” for any number of any characters.
Miscellaneous:	
Shift + INSERT	Commenting out of cycles and direct editing of programGUIDE cycles
Shift + END	Select up to end of block
Shift + NEXT WINDOW	Select up to start of line
ALT + NEXT WINDOW	Jump to start of line
ALT + S	Enter Asian characters
=	Calculator function
HELP	Help function
END	Jump to end of line

Simulation and simultaneous recording:	
▲ ▶ ▼ ◀	Move
Shift + ▲ / ▼	Rotate in 3D display
▲ ▼	Move section
CTRL + ▲ / ▼	Override +/- (simulation)
CTRL + S	Single block on/off (simulation)
Insert key:	
	It brings you into the Edit mode for text boxes or into the Selection mode of combo boxes and toggle fields. You can exit this without making any changes by pressing Insert again.
	Undo function,as long as no Input key is pressed or no data has already been transferred to the fields.
Toggle key:	
	You can directly switch between toggle fields using the Toggle key (Select) without having to open them. With Shift-Toggle you can switch through these in the reverse direction.
Cursor key:	
◀ ▶	Open/close directory Open/close program Open/close cycle

6.3 Further information

CNC equipment of SINUMERIK

www.siemens.com/sinumerik
<*http://www.siemens.com/sinumerik*>

Doconweb

www.automation.siemens.com/doconweb
<*http://www.automation.siemens.com/doconweb*>

CNC4you - User portal

www.siemens.com/cnc4you
<*http://www.siemens.com/cnc4you*>

SINUMERIK - User forum

www.siemens.cnc-arena.com
<*http://www.siemens.cnc-arena.com*>

CAD/CAM from Siemens

www.siemens.com/plm
<*http://www.siemens.com/plm*>

7 Index

Aligning milling tool
Multitasking machines 120
Aligning turning tool
Multitasking machines 118
Animated elements 14
Blank 106
Block search 31
Boring 93
Centering 49, 92
Centric drilling 48
Collision Avoidance 34
Contour call programGUIDE
Contour milling 74, 104
Contour turning 62, 129
Contour milling
programGUIDE 103
ShopMill 101
Contour turning
programGUIDE 128
Counterspindle 76
Cutoff 57, 125
Cutting 55, 123
Cycles
Parameters 20
Variables 20
Cylinder jacket 110
Deep hole drilling 50, 93
Drilling Reaming 49, 92
Drilling with programGUIDE 53
Drilling with ShopTurn
centered/off center 51
Dual editor
programSYNC 134
Editor
Open second program 18
Search 19
Settings 18
Settings multi-channel 135
Engraving 68, 98
Face milling 85, 95
Graphic view 14
Groove 56, 124
Help
context-sensitive 15
Help Screen 14
HighSpeed settings 108
in-process measurement 77, 111
Workpiece measurement – align-
ment workpiece probe 112, 113
Machine basic screen
Multi-channel machining 132
Machining time
programSYNC 137
Manual Machine
Milling 87
Turning 42
Measuring cycles
Example 78
Overview 77, 111
Mirroring 110
Mold making view 30
Multi-channel program data 134
Multiple clamping 28
Offset 109

Open job list....133
Path milling
Contour....69, 99
Plunge-cutting
Contour....58, 126
Plunge-turning
Contour....59, 127
Pocket....66, 96
Contour....70, 100
Positions....51, 94
Program control
programSYNC....138
Program management....25
Create archive....27
Create directory....25
Create ShopMill/ShopTurn program.
25
Create tool....25
*.jpg, *.png, *.bmp....26
PDF....26
Preview....26
programGUIDE create G code....25
Properties....26
Save setup data....27
Programming
Form blocks....19
ISO code....17
programGUIDE....17, 91
programSYNC....17
ShopMill....89
ShopMill/ShopTurn....17
programSYNC....132
Dual editor....134
Job list JOB....133
Machining time....137
program control....138
Simulation....136
Synchronous view....136
Time synchronization....135
Wait marks....135
Rotation....109
Rough drill....69, 99
Scaling....110
Settings
Editor....18
Editor multi-channel....135
Set ZO....16, 38, 82
ShopMill
Program header....88
ShopTurn
Program header....44
Simulation
programSYNC....136
Simultaneous recording....32
Slot....67, 97
Spigot
Contour....70, 100
Spigot polyhedron....67, 97
Stock removal
Contour....58, 126
Swivel
Plane....86
Swivel axis
TSM mode....117
Swivel plane
Multitasking machines....119, 121
Synchronous view
programSYNC....136
Thread....50, 57, 94, 125
Thread milling....68, 98
Time synchronization
programSYNC....135
Tool
Favorites....23
Multitool....23
Search....23
Tool list....21
Details....21
Sister tool....21
Tool management

Multitasking machines 116
Multitools 116
Tool measurement
Counterclamping holder 40
JOG 39, 84
Tool probe 40
Tool simulation
3D 29
Cross-section 30
Details 29
Tooltip 15
Tool wear 22
Transformation 109
TSM cycle 16
TSM mode
Swivel axis 117
Undercut 56, 124
Wait marks
programSYNC 135
Workpiece zero 39, 83
Zero offset 24, 109

CNC equipment of SINUMERIK
www.siemens.com/sinumerik

Doconweb
www.automation.siemens.com/doconweb

CNC4you - User portal
www.siemens.com/cnc4you

SINUMERIK - User forum
www.siemens.cnc-arena.com

CAD/CAM from Siemens
www.siemens.com/plm

The information provided in this brochure contains descriptions or characteristics of performance which in case of actual use do not always apply as described or which may change as a result of further development of the products. An obligation to provide the respective characteristics shall only exist if expressly agreed in the terms of contract.

All product designations may be trademarks or product names of Siemens AG or supplier companies whose use by third parties for their own purposes could violate the rights of the owners.

Siemens AG
Industry Sector
Motion Control Systems
Postfach 31 80
91050 ERLANGEN
GERMANY

Subject to change without prior notice
Order no.: 6FC5095-0AA84-0BA1
Printed in Germany

© Siemens AG 2013