SIEMENS

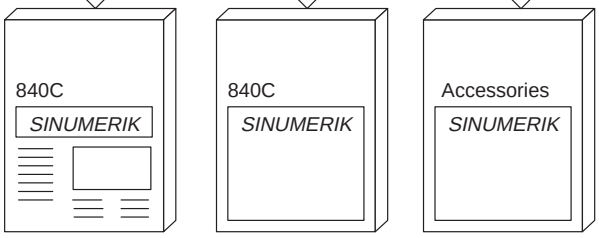
SINUMERIK 840C
Software Version 6
Simulation – Milling and Turning

User's Guide 07.97 Edition

User Documentation

SINUMERIK 840C/OEM Version for Windows

General Documentation

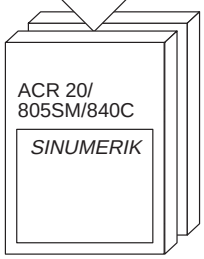


Brochure

Catalog NC 36

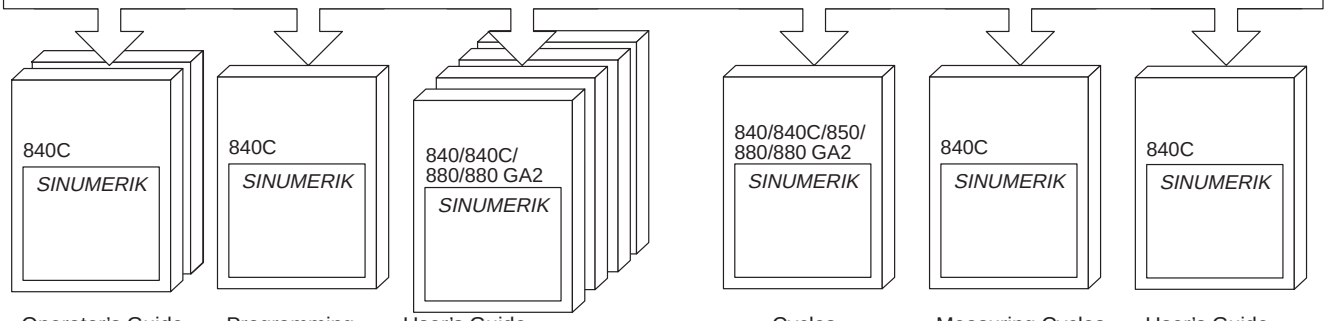
Catalog NC Z

User/Manufacturer/ Service Docum.



Link to SINEC L2–DP with Module
- IM 328-N, Slave
- IM 329-N, Master and Slave

User Documentation



Operator's Guide
- OEM Version for Windows
- Standard Diagnostics Guide

Programming Guide

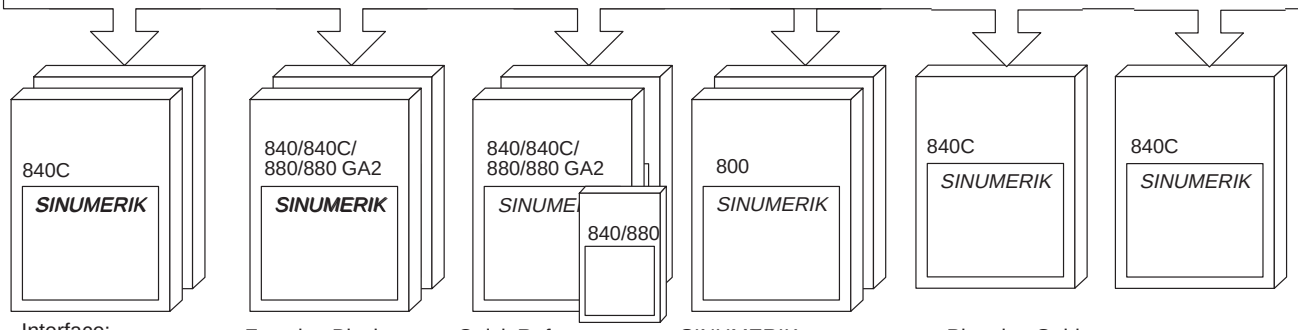
User's Guide Graphic Progr. System
- Drilling/Boring and Milling Parts 1+2
- Turning Parts 1+2
- On PC
- Environment Description 840C

Cycles, Programming Guide

Measuring Cycles Version 20 User's Guide

User's Guide Simulation – Milling and Turning

Manufacturer Documentation



Interface:
- Signals
- Connection Conditions

Function Block Packages, Function Macros

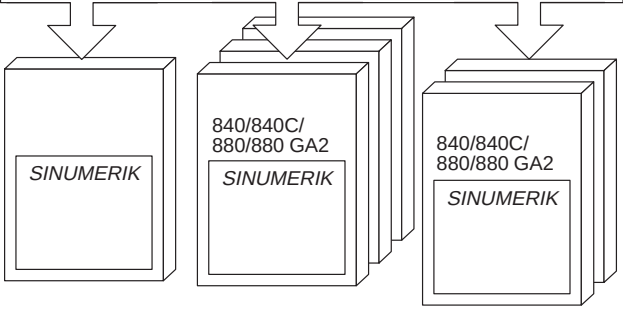
Quick Reference, Planning PLC 135 WB/WB2/WD S5–HLL

SINUMERIK WS 800A
- CL800 Cycle Language
- User's Guide

Planning Guide Graphic Programming System

OEM Version for Windows
- User's Guide
- Alarm Dialog for PC

Manufacturer Documentation

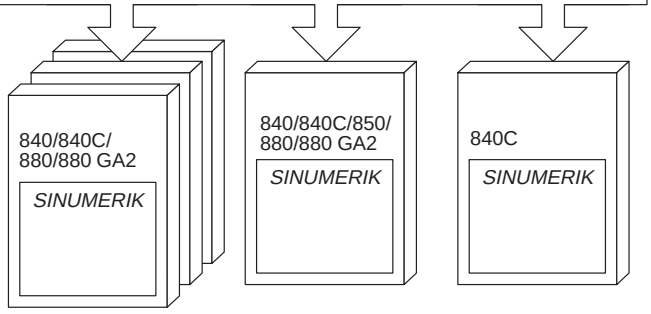


Description of Functions Safety Integrated

Computer Link
- SINT
- SIN PS 231
- SIN PS 315

Computer Link
- Message Frame Description
- General Description

Service Documentation



Installation Guide
- Instructions
- Lists
- Difference Description Windows

Measuring Cycles Version 20 Start-up Guide

Spare Parts List

SIEMENS

SINUMERIK 840C Software Version 6

Simulation – Milling and Turning

User's Guide

User Documentation

Valid for

Control	*Software Version*
SINUMERIK 840C/CE (Standard/Export Version)	as from SW 4.8

07.97 Edition

SINUMERIK® documentation

Printing history

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

The status of each edition is shown by the code in the “Remarks” column.

Status code in the “Remarks” column:

A . . . . New documentation.
B . . . . Unrevised reprint with new Order No.
C . . . . Revised edition with new status.
If factual changes have been made on the page since the last edition, this is indicated by a new edition coding in the header on that page.

Edition	Order No.	Remarks
12.93	6FC5198–3AA30–0BP0	**A**
10.94	6FC5198–4AA30–0BP0	**C**
09.95	6FC5198–5AA30–0BP0	**C**
10.96	6FC5198–5AA30–0BP1	**C**
07.97	6FC5198–6AA30–0BP0	**C**

Siemens quality for software and training
to DIN ISO 9001, Reg. No. 2160–01.

This publication was produced with Interleaf V 5.4

Order No. 6FC 5198–5AA30–0BP2
Printed in the Federal Republic of Germany

Siemens–Aktiengesellschaft

Preliminary Remarks

Notes for the reader

The SINUMERIK documentation is organized in 4 parts:

- General documentation
- User documentation
- Manufacturer documentation and
- Service documentation

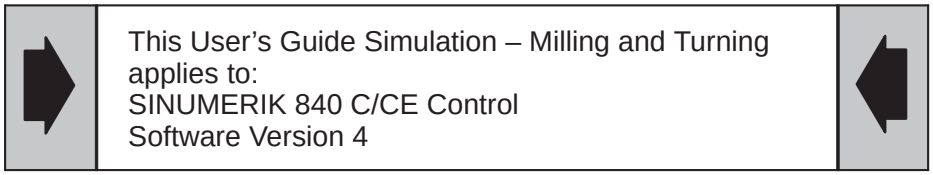
This manual is intended for users of machine tools. It provides all the detailed information that the user requires for operating and parameterizing the graphical simulator on the SINUMERIK 840 C.

The manual describes the functionality provided with the standard products in the range. A description of add-on features or modifications made by the machine vendor cannot be included in this manual.

Additional publications are available for the SINUMERIK 840 C as well as for all the SINUMERIK controls (e.g. publications on the universal interface, measuring cycles, etc.) from your local Siemens office.

Other functions not described in this documentation might be executable in the control.

This does not, however, represent an obligation to supply such functions with a new control or when servicing.



Contents

1 Introduction

The simulator makes it possible to represent a machining process on screen.

1.1 General remarks

The simulator makes it possible to represent a machining process on screen. The machining process is visualized dynamically on the basis of the control data (part program, work schedule, TOA file, machine data, etc.). You can therefore test programs produced graphically and DIN programs before starting them up on the machine. Several side views, the ability to rotate the workpiece about an axis and a zoom function help you to check the result.

The correction and refinement of NC part programs is cost and time-intensive. Performing test runs directly on the CNC machine has disadvantages that can be avoided with the aid of simulation:

- The downtime caused by setup, machine and programming time during the test run is eliminated.
- No workpieces are required (when using the original unmachined part there is a risk that the testpiece will be rendered unusable due to programming errors).
- Collisions caused by incorrect positional data can be observed on the screen. Traversing movements not required for the machining process increase the machining time and can be detected on the screen.
- The use of incorrect tools and technology data can be detected on the screen during simulation.
- Incorrect cycles and incorrect cycle parameters can be detected before machining.

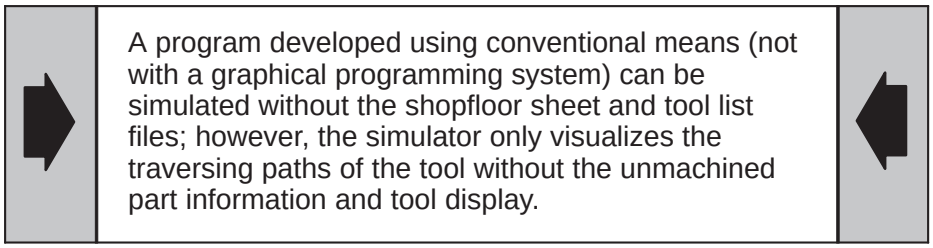
The simulator operates like a stand-alone NC control, i.e. a set of machine data is used, axes interpolate, almost all of the NC programming commands can be used, etc. There is no immediate relationship between the NCK on the SINUMERIK 840 C and the simulation software. The required data, workpieces, programs, etc. can be loaded into the simulation area from the hard disk. All other functions are independent of the system software. It is therefore possible to run simulation parallel to but not simultaneously with NCK machining.

The simulator is configured for the mechanical requirements by defining the machine axes (X, Y, Z...).
The NC functionality is configured by loading the machine data and parameter blocks.

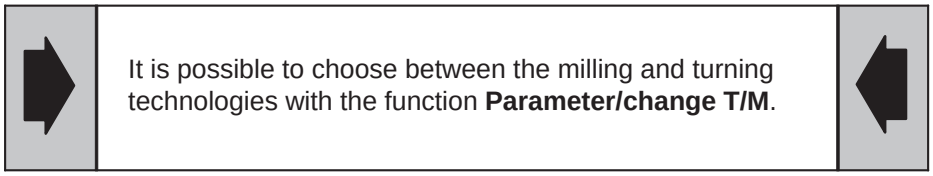
A workpiece program produced on a graphical programming system automatically contains a

- Job list
- Shopfloor sheet and
- Tool list.

The simulation software processes this information. A program developed by conventional means (on an editor) does not contain these files. You will need to create these files if the information from the job list, shopfloor sheet and tool list is required during simulation.



A program developed using conventional means (not with a graphical programming system) can be simulated without the shopfloor sheet and tool list files; however, the simulator only visualizes the traversing paths of the tool without the unmachined part information and tool display.

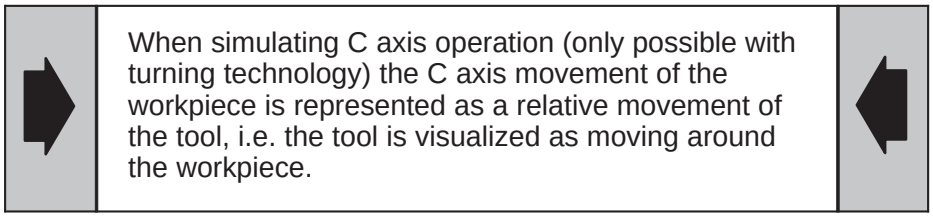


It is possible to choose between the milling and turning technologies with the function **Parameter/change T/M**.

Please note that the functions configured via the PLC are not included in the simulation (e.g. software cams, limit switches, spindle positioning, etc.).

The tool change is performed by the simulator. The tool management implemented in the PLC program is **not** supported by the simulator.

Collision monitoring cannot be performed by parameterization, but visual inspection can be carried out by simultaneously displaying the plan view and side or front views. The C-axis mode can only be simulated in the turning technology.



When simulating C axis operation (only possible with turning technology) the C axis movement of the workpiece is represented as a relative movement of the tool, i.e. the tool is visualized as moving around the workpiece.

The simulator only provides for a single channel. You can select the channel from an input display.

The simulator is configured for 2 1/2D machining (3 linear axes are allowed for). In addition, 2 rotary axes can be simulated and peripheral surfaces and face ends can be machined with the turning technology. You can simulate DIN programs with @ commands, zero offsets, etc.

It is also possible to simulate programs with relief cuts and machining operations on multiple sides.
The simulation screen display highlights the various machining depths through different colors.
You can adjust the color levels of the depth ranges from parameter displays.

The color levels are visualized in the following views:

- Milling: “3D view and plan view”
- Turning: “Face end view” (face machining), contour machining (XZ plane) is not represented in color levels.

Inclined tool movements are broken down and represented as a staircase (Fig. 1.1).

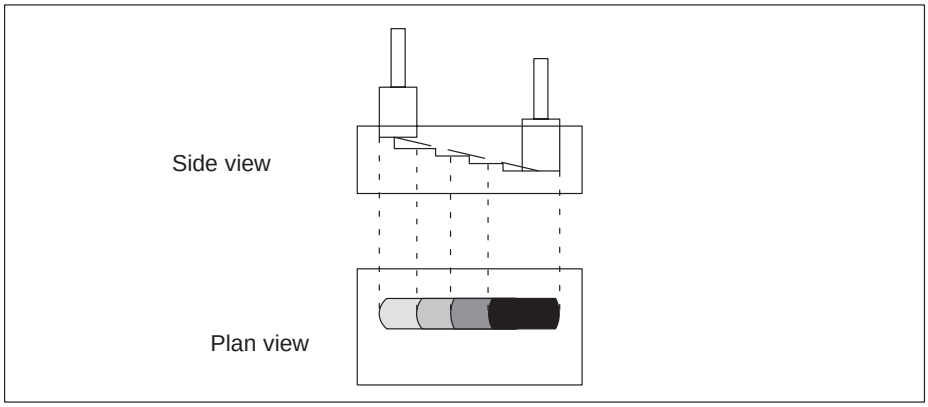


Fig. 1.1 Inclined tool movement

Tools are displayed in boxes.

Milling

The machining plane is defined in the part program. The plane selected in plan view **only** displays the machining operations that take place on this plane.
There is no relationship between the machining operations on different planes in the plan view; the complete machining process is only displayed in the 3D view.
Only the face can be visualized in the machining of cylindrical parts.

Turning

The machining plane is defined in the part program. The plane selected in the plan view represents the machining operations that take place on this plane.
The views

- “plan view” and
- “peripheral surface”

are updated on **STOP** or when the machining plane is changed.

1.2 SIMULATION as part of the SINUMERIK 840C standard user interface

The **Simulation** software package described in this User's Guide does not belong to the scope of supply of the standard SINUMERIK 840C software but can be provided as an optional extra.

You will find a description of the **Machine, Parameter, Programming, Utilities** and **Diagnosis** user areas in the "SINUMERIK 840C Operator's Guide, Software Version 6".

Key symbols

It is assumed that you are using the keypad on the operator panel. You will find a description of the keypad along with a list of the differences between the operator panel keypad and the standard MF2 keyboard in the Operator's Guide.

Prerequisites

This User's Guide refers exclusively to the operator actions of the **Simulation** user area. The **Simulation** User's Guide follows directly on from the Operator's Guide. It is assumed that the reader is acquainted with the functions described in the Operator's Guide and these functions are therefore not described here.

1.3 General operator actions

Operating sequence

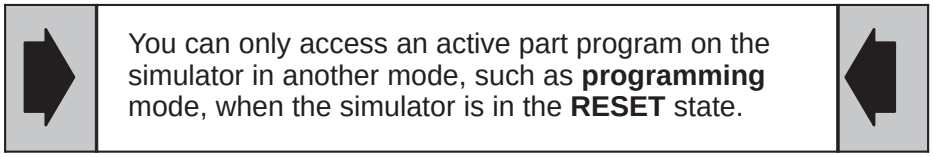
Use the **arrow keys** to select the parameter field that you want to change.

Press the **search key** to switch to the next option in SELECT parameters (toggle fields).

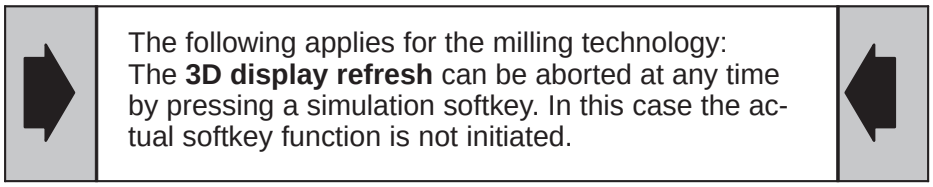
Activate the **shift key** in conjunction with the **search key** to switch to the next option in the opposite sequence in SELECT parameters.

Press the **OK** softkey to save the current parameter settings.

Press the **Abort** key to cancel the modified parameter settings.



You can only access an active part program on the simulator in another mode, such as **programming** mode, when the simulator is in the **RESET** state.



The following applies for the milling technology:
The **3D display refresh** can be aborted at any time by pressing a simulation softkey. In this case the actual softkey function is not initiated.

Note for milling technology

The 3D display is not updated continuously during the machining simulation. The result of the machining process is visualized in the 3D display after the program has terminated.

Certain user selections cause the 3D display to be updated.

Example

A simulation program has been executed. The “Automatic refresh 3D” setting in the **Parameter** display was not active, i.e. the unmachined part was visualized in the 3D display.
If you now switch to another display such as **Window large** and then return to the basic menu, the 3D display is updated and shows the current state of machining.

■

2 Simulation Functions

This section uses the standard workpiece EXMP_M to describe all of the functions in the **Simulation** area.

EXMP_M is the demonstration program for the milling technology.

EXMP_T is the demonstration program for the turning technology.

2.1 Requirements

Perform the following operations to activate **simulation** of the workpiece EXMP_M:

After you have switched on the control and pressed the **area switchover key**

Programm.

activate the **Programm**.ing area.

Once you have extended the horizontal softkey bar with the **ETC key**

Simulation

you can call the simulation function with the **Simulation** softkey.

Technology “Turning” is active with standard workpiece EXMP_T.

Parameter

Press the function **Parameter** and then

Change T/M

the function **Change T/M** to change to workpiece EXMP_M for technology “Milling”.

Fig. 2.1 below shows the basic display for **simulation** with the technology “Milling” and the standard workpiece EXMP_M:

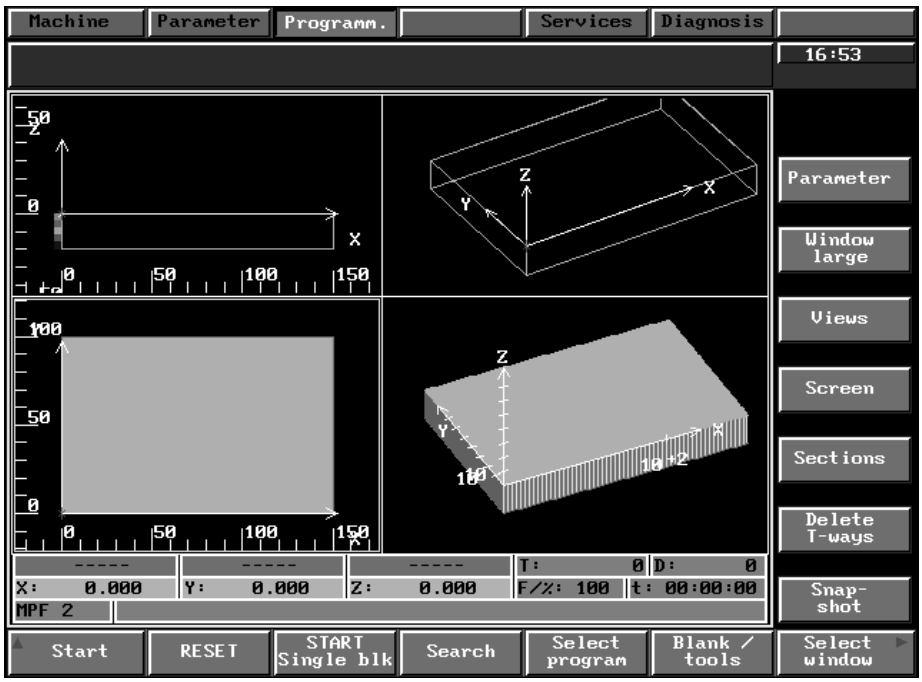


Fig. 2.1 Simulation basic display, milling technology

In the **Simulation** area you can trace the result of DIN 66025 machining programs on the screen.

To do this, select any program.

This program can be simulated while another program is being executed on the machine.

2.2 Changing the technology

Turning is the default setting when the simulation program is installed.

Changing the technology

If you want to switch over from “Milling” to “Turning” and vice versa,

Parameter

press the vertical softkey function **Parameter** and

Change T/M

activate the other technology for simulation with the softkey **Change T/M**.

When simulation for “Turning” is started up for the **first** time:

- the file SIMCONF (config file)
- the machine data record STANDD_T from the Siemens branch and
- the workpiece EXMP_T

are loaded as a default.

When simulation for “Milling” is started up for the **first** time:

- the file S_CONF_M (config file)
- the machine data record STANDD_M from the Siemens branch and
- the workpiece EXMP_M

are loaded as a default.

If another workpiece is currently selected in the programming function, this workpiece is loaded.

2.3 Selecting the program/workpiece

Workpieces

- EXMP_M (**“**milling”) and
- EXMP_T (**“**turning**”**)

are installed as standard under the GLOBAL directory in the operating software.

If you are in the **Programm**.ing area and have selected a workpiece, simulation is started up with this workpiece.

If you have not selected a workpiece, the workpiece simulated last is active.

Prerequisites

- The **Simulation** area must be selected.
- No simulation program should be active. A new program can only be selected when the end of the program has been reached or by pressing the **RESET** softkey.
- You can select a new workpiece at any time.

2.3.1 Selecting the program

Key sequence

Select program

Press the softkey. A list will be displayed of the main programs and subroutines defined in the JOBLIST belonging to the workpiece.
If no JOBLIST exists, all of the programs stored under the workpiece will be listed.

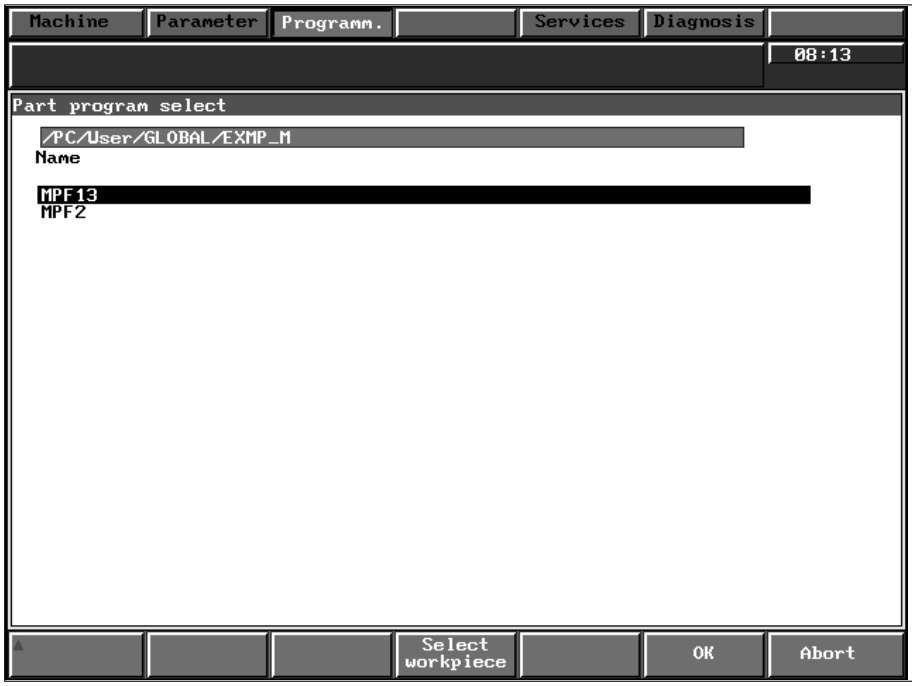


Fig. 2.2 Directory of the main programs and subroutines defined in the joblist for the program

Use the **arrow keys** to position the cursor on the part program of the current workpiece to be simulated.

OK

Press the **OK** softkey to confirm your selection.

You can use the operating functions of the data selector to choose a workpiece or program.

2.3.2 Selecting the workpiece

Prerequisites

First press the **Select program** softkey. You are now at the workpiece program level.

Key sequence

Select workpiece

Press the **Select workpiece** softkey.

Use the **arrow keys** to position the cursor on the selected workpiece.

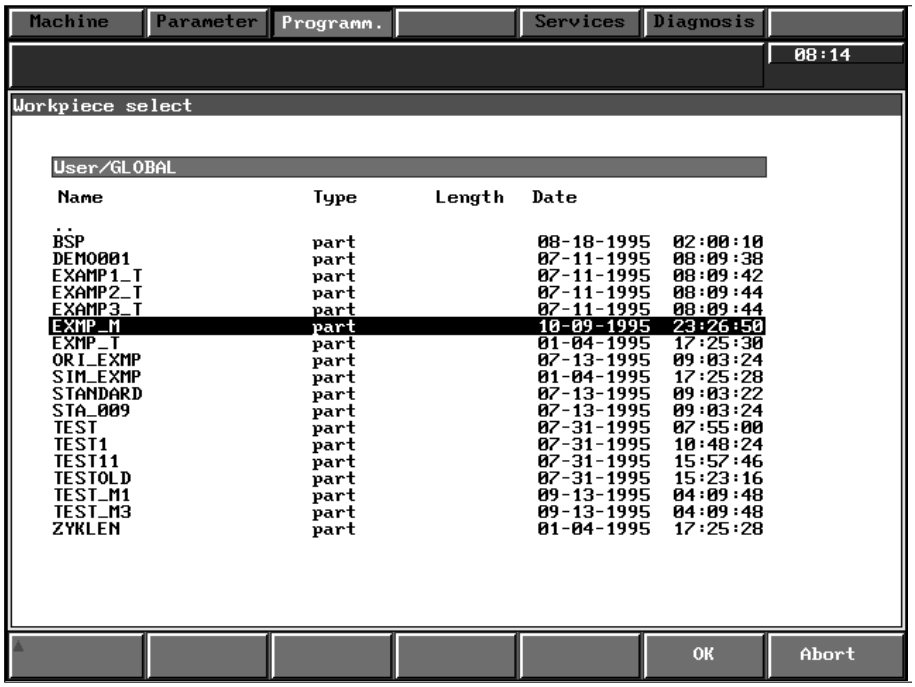


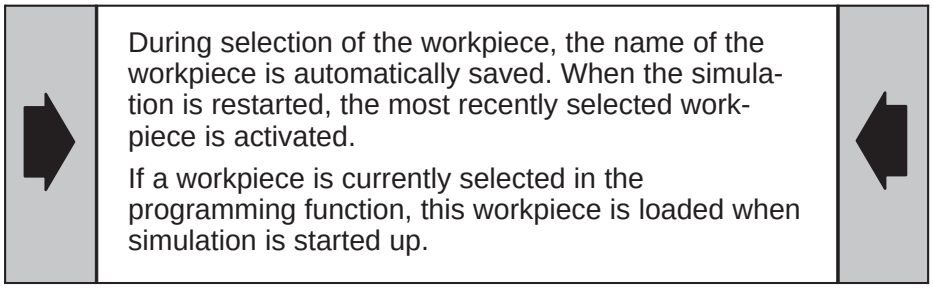
Fig. 2.3 Directory of the GLOBAL workpieces in the simulation area

OK

Press the **OK** softkey to confirm your selection.

Confirming the selected workpiece starts the setup procedure for the simulation.

When the workpiece is selected, the STANDARD workpiece, all of the simulation parameters and the files belonging to the selected workpiece are **always** loaded. If the files TOA, SEA, SEA4, RPA and ZOA exist in the workpiece, the system data within the simulation are overwritten. Otherwise the previous data are retained.
Only the related programs (MPF, SPF) are valid for workpiece selection.



During selection of the workpiece, the name of the workpiece is automatically saved. When the simulation is restarted, the most recently selected workpiece is activated.

If a workpiece is currently selected in the programming function, this workpiece is loaded when simulation is started up.

Note

If the channel-specific data:

- R parameter and
- Coordinate rotation

are assigned to a workpiece, these are also loaded by the simulator in channel-specific format. If a workpiece does not have any channel-specific data, the data are filled with ZERO.

The workpiece is loaded in the following sequence:

- Loading of the STANDARD workpiece
- Loading of the simulation workpiece

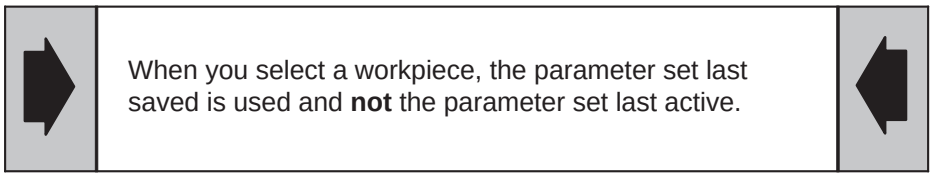
The simulation workpiece overwrites the same data of the STANDARD workpiece. The following are loaded when the workpiece is selected:

- Parameter settings
- Program modification settings
- Search settings
- Window, view settings, etc.

The load list is only booted when the simulation is reselected (see Section 3.2.6 “Loadlist”). The definition of the parameter set is **not** workpiece-specific, i.e. only one set of parameters exists.

You can use the **save parameters** function to back up all parameters and their settings. When selecting the workpiece, the most recently saved parameter set is used. If no user parameter set exists, the standard Siemens data set is automatically copied into the user branch and loaded.

The user parameter set can be deleted completely from the **services/ data management** module. You will find this file SIMCONF and S_CONF_M or S_CONF_T in the **SIMULATION/CONFIGURATION** directory. When the user file is deleted, the standard Siemens data set applies again.



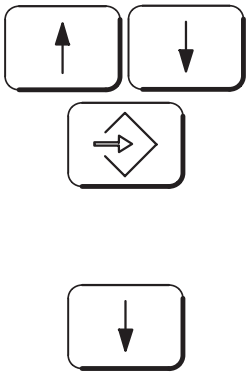
When you select a workpiece, the parameter set last saved is used and **not** the parameter set last active.

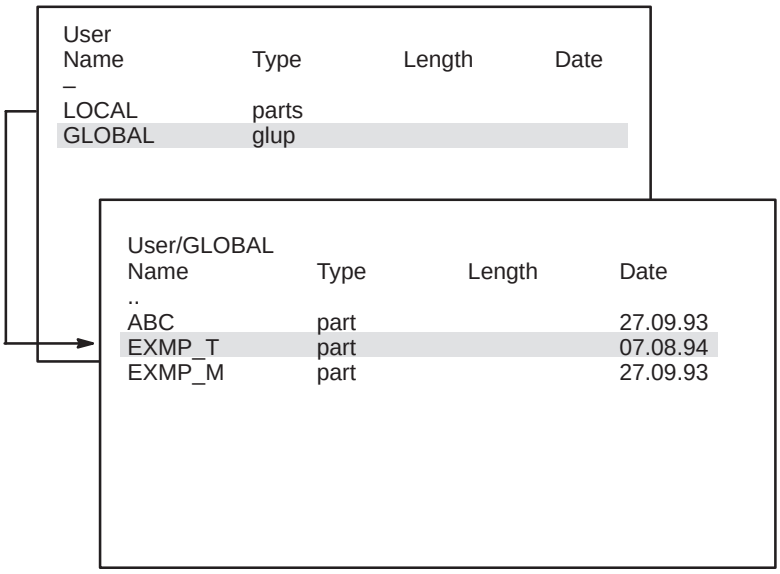
2.3.3 Selecting the EXMP_T workpiece

Select program

Select workpiece

From the basic display in the **Simulation** area press the **Select program** and **Select workpiece** softkeys consecutively.
Browse through the workpiece data as follows:





OK

Press the **OK** softkey to start the setup procedure for the simulation with the EXMP_T workpiece.

Press **RECALL key** or

Abort

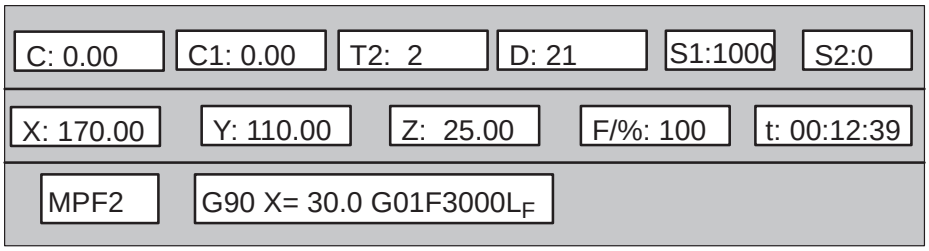
the **Abort** softkey to quit the menu **without** saving the changes.

2.4 Displays

The current traversing block and the current program are displayed in the simulation. Also displayed are the current destination position in programming coordinates, override, calculated cutting time.

The T number with accompanying tool offset is displayed in the milling technology.

Display for turning technology:



An * displayed before the current block indicates an insert block (e.g. from the TRC, contour definition).
Cycles are **not** displayed here.

2.5 START / STOP

Using the EXMP_M (milling) and EXMP_T (turning) example workpieces, provided as standard with the simulation software, this section describes the functions of a running simulation.

Prerequisites

- Workpiece EXMP_M or any other workpiece has been selected
- The standard simulation parameter block has **not** been modified, i.e. no zero offsets, TOA data blocks, etc. have been changed.

START

Press the **START** softkey to start the simulation. The following horizontal softkey bar (see Figs. 2.4 and 2.5) is displayed:

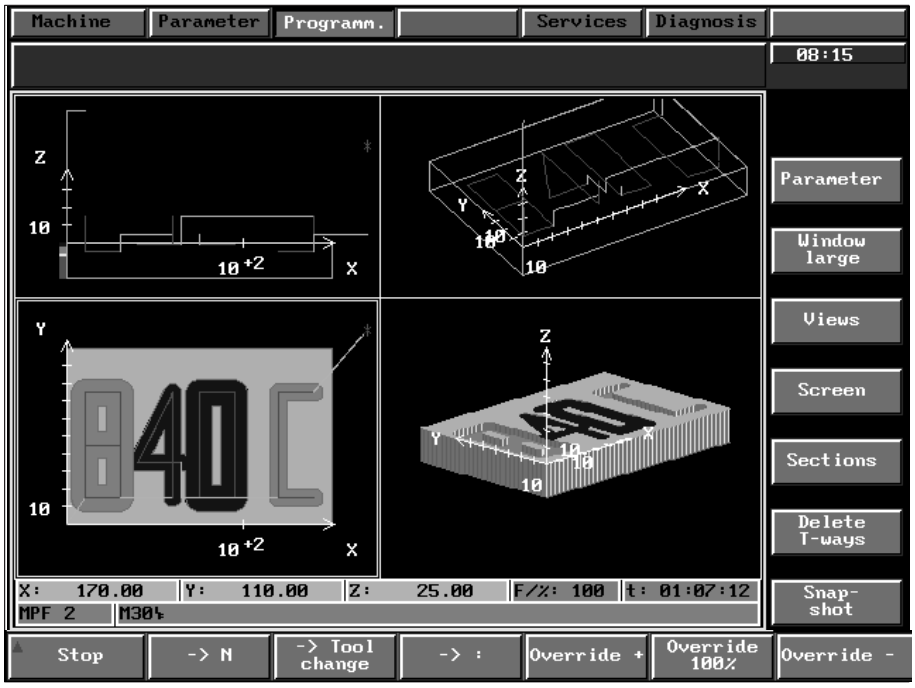


Fig. 2.4 Simulation run for the EXMP_M workpiece, milling technology

Depending on the feedrate, the maching time of the workpiece is calculated from the traversing paths.

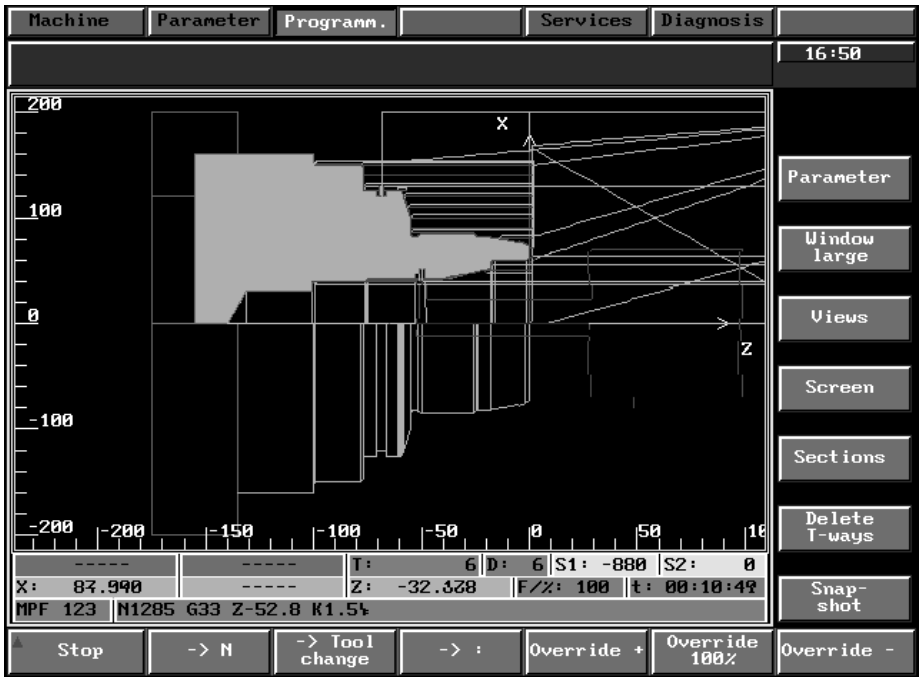


Fig. 2.5 Simulation run for the EXMP_T workpiece, turning technology

STOP

Press the **STOP** softkey to stop the simulation.
The horizontal and vertical basic menu for the **STOP/RESET** simulation state is displayed on the screen.

→ N

Press the → **N** softkey to suppress the screen output until the next block is reached.

→ Tool change

Press the → **Tool change** softkey to suppress the screen output until the next tool change identifier is reached.

→ :

Press the → **:** softkey to suppress the screen output until the next main block is reached.

Note

The screen display can be updated at any time using the

Snap-shot

Snapshot function.

If no block number, tool change or main block exists, the screen output is suppressed until the end of the program is reached.

Override +

Press the **Override +** softkey to increase the speed of the simulation.

Override –

Press the **Override –** softkey to reduce the speed of the simulation.

Override 100%

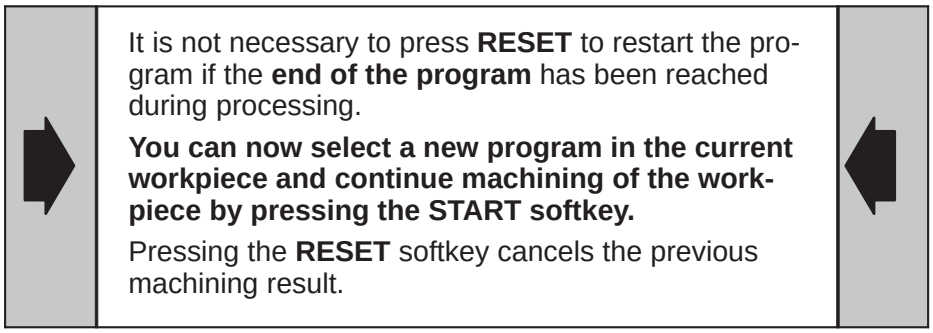
Press the **Override 100%** softkey to adjust the simulation speed to the value set:

- The parameter “Full load” in the **Parameter** screen determines the preset value for the display precision.
- In timed mode the 100% value is equal to the programmed feedrate.

2.6 START / RESET

START

Press the **START** softkey to resume machining. The simulation restarts from the point reached when **STOP** was pressed or from the start of the program if **RESET** has been pressed (or if the end of the program M30, M17, M02 has been reached).
The horizontal basic menu for the **START** simulation state is displayed on the screen.



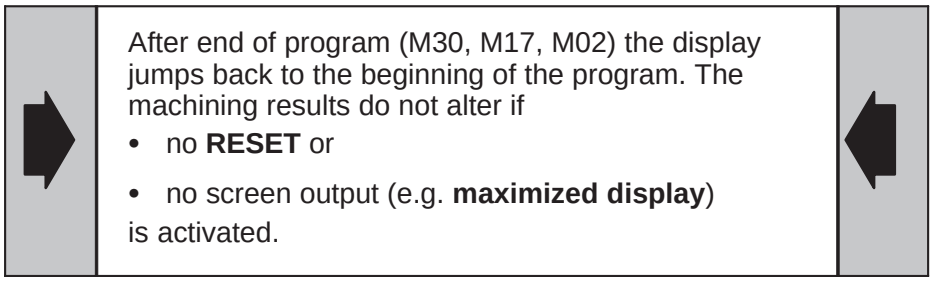
It is not necessary to press **RESET** to restart the program if the **end of the program** has been reached during processing.

You can now select a new program in the current workpiece and continue machining of the workpiece by pressing the START softkey.

Pressing the **RESET** softkey cancels the previous machining result.

RESET

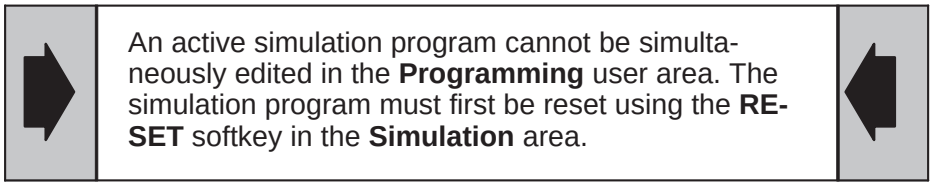
If you press the **RESET** softkey the unmachined part is displayed on the screen. An asterisk (*) is displayed at the starting point. The starting point is identical to the reference point from the machine data block.



After end of program (M30, M17, M02) the display jumps back to the beginning of the program. The machining results do not alter if

- no **RESET** or
- no screen output (e.g. **maximized display**)

is activated.



An active simulation program cannot be simultaneously edited in the **Programming** user area. The simulation program must first be reset using the **RESET** softkey in the **Simulation** area.

2.7 Start Single block

Start
Single Block

If you press the **Start Single Block** softkey, the simulation traverses a single block and then goes to the STOP status.

2.8 Search

Search

Press the **Search** softkey to display a parameter screen for starting and stopping the search function.

The search can be performed on a

- Tool change (specify the T-number, search up to the following change identifier)
- Main block
- Block number in a specific main program or subroutine.

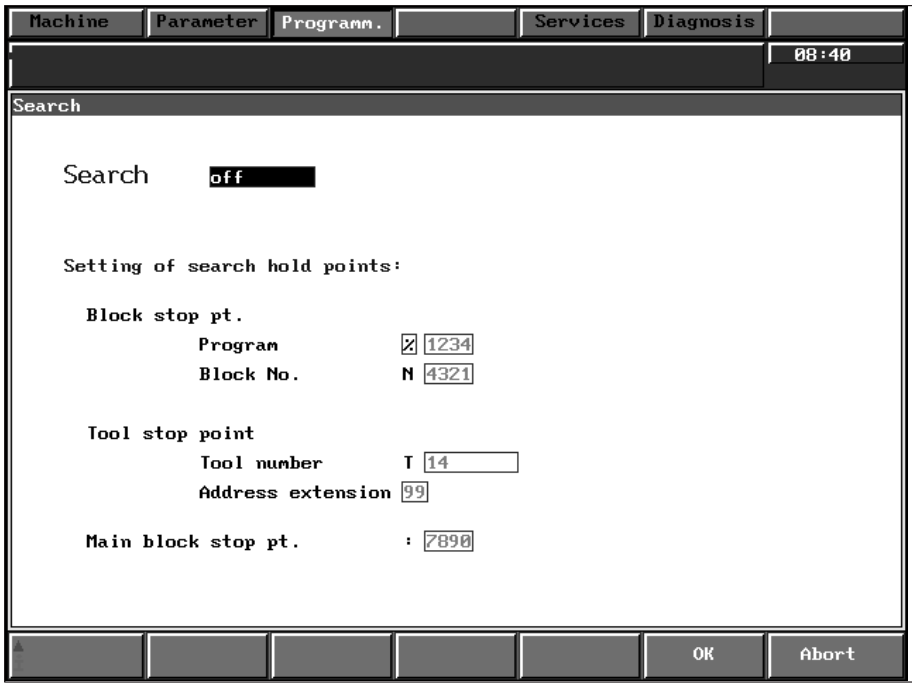


Fig. 2.6 Search settings in the simulation mode

The type of search is specified in a selection box.

- “Search off” (default)

No search takes place.

- “Tool change”

You can specify a T number. The search continues until the next tool change identifier is located.

- “Main block”

The search is performed until the specified main block number is located.

- “Block number”

You can specify a “program” and a “block number” as stop point criteria.

When a search is enabled, the simulation display is refreshed at the destination position. The simulation changes to **STOP** status. The graphical output is suppressed until the search destination is reached. If the search destination is not found, the simulation display is generally refreshed when the program terminates.

The search is always performed with calculation. Only the screen output is suppressed.

The new settings which you confirm by pressing **OK** take effect immediately. If the settings are to be retained after simulation is terminated, the parameters must be saved on the hard disk.

START

Press the **START** softkey to start the block search.

Note

The search is only performed if it is enabled in the settings display.

2.9 Blank/tools

Blank/ tools

Via the **Blank/tools** softkey, you can alter and re-enter the geometries for

- blank
- tools
- Tool location assignment (activation/deactivation in parameter display)
- chucks and chucking scenarios

for the current workpiece.

You can enter the geometry via a parameter display with explanatory auxiliary graphics.

The following Fig. 2.7 shows an example for a cutting tool:

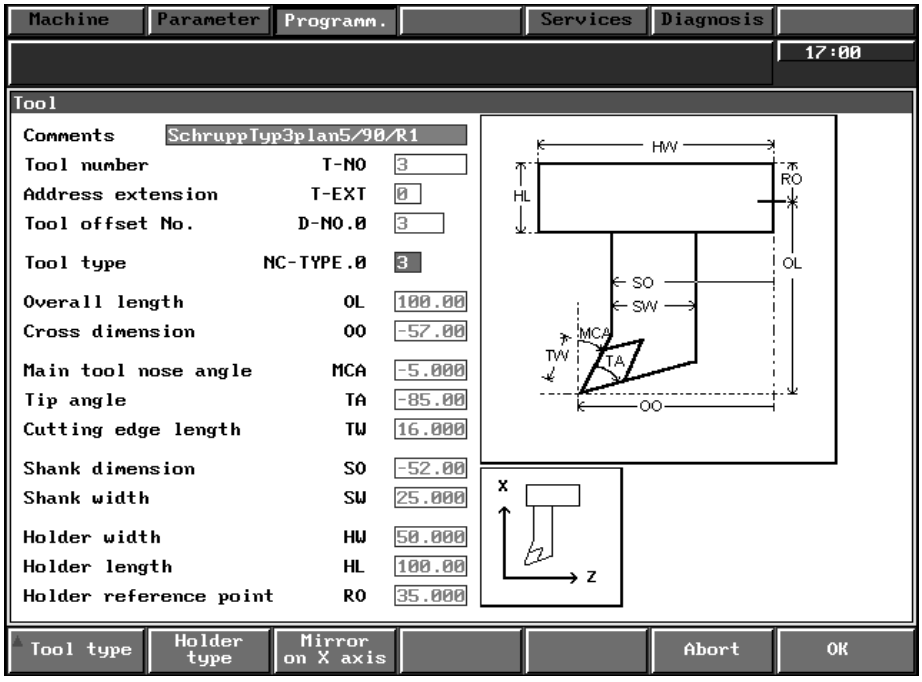


Fig. 2.7 Cutting tool

An existing shopfloor sheet or a tool list is updated with the new parameters. If those are not available, the simulation generates a new tool list or shopfloor sheet under the current workpiece.

Any changes become effective for the current workpiece simulation by pressing **OK**.

Blank

The blank can be the geometry of a

- cylinder
- hollow cylinder or
- cuboid.

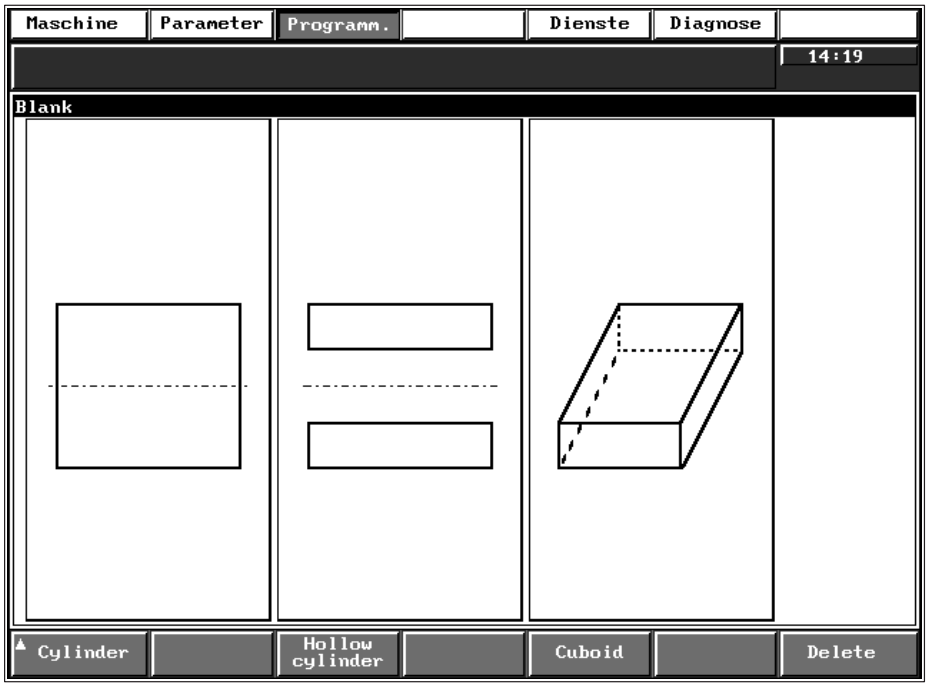


Fig. 2.8 ”Blank selection” menu

Cylinder

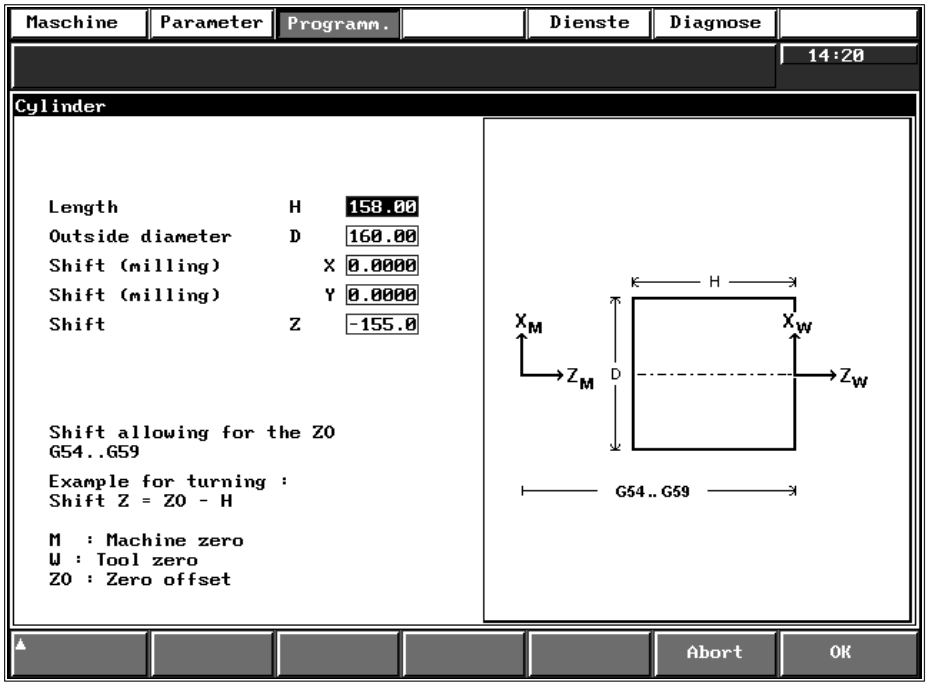


Fig. 2.9 “Cylinder” blank selection

You should not enter the geometry of a blank as a cuboid for the **Turning** technology.

In the **Turning** technology, the **Offset Z** parameter is evaluated.
In the **Milling** technology, the offset in **X, Y, Z** is evaluated.

Tool

The tool can be the geometry of type

- cutting tool
- recessing tool
- thread cutter
- drill or
- milling cutter.

The following figure shows the menu level for the selection of **Tools**:

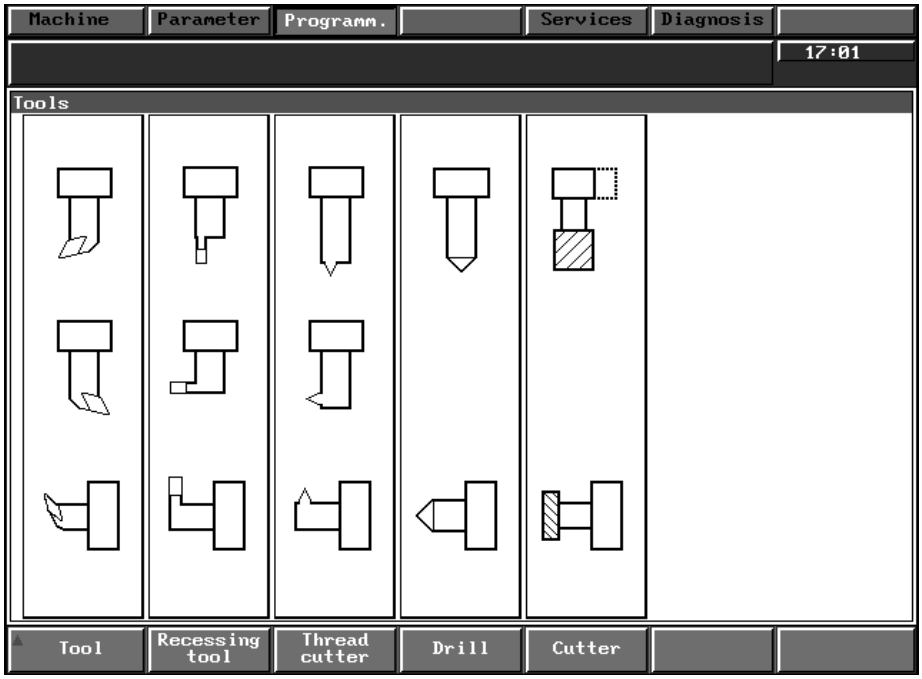


Fig. 2.10 “Tools” menu

The simulation does **not** check whether a cutting tool or drill, for example, is amongst the tools provided.

Tool location reassignment

In Turning technology, you can assign the tool to a certain turret location. If the turret locations have been assigned differently for various workpieces, the assignment of tools to turret locations can be altered at the operator interface.

A maximum of 12 tool locations can be reassigned.

Tools without location coding can be loaded directly, i.e. T 17 is the tool with the identification number 17.

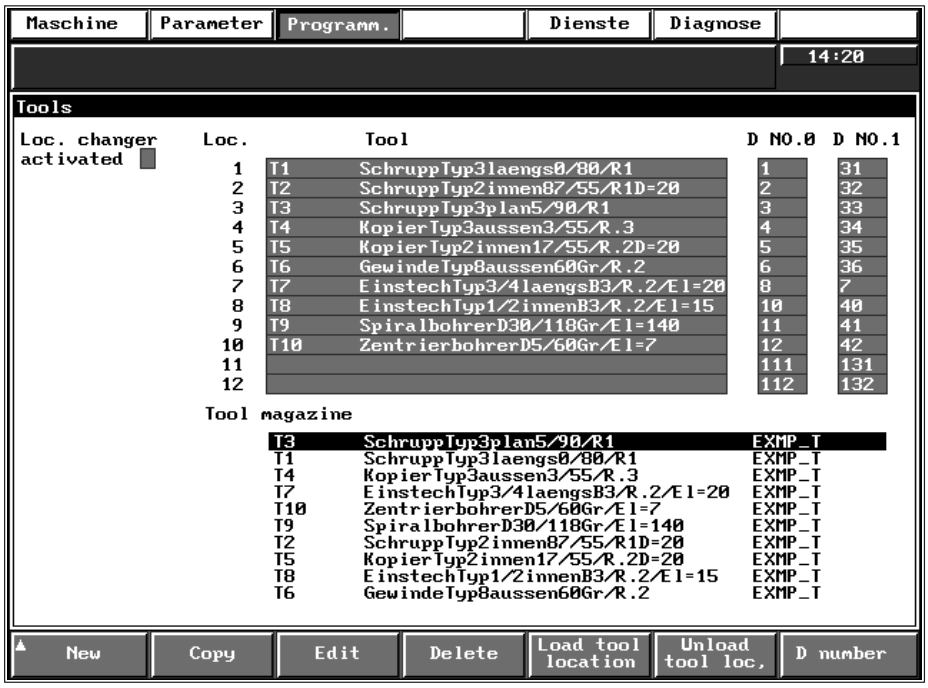


Fig. 2.11 Tool relocation for turning

Load tool location

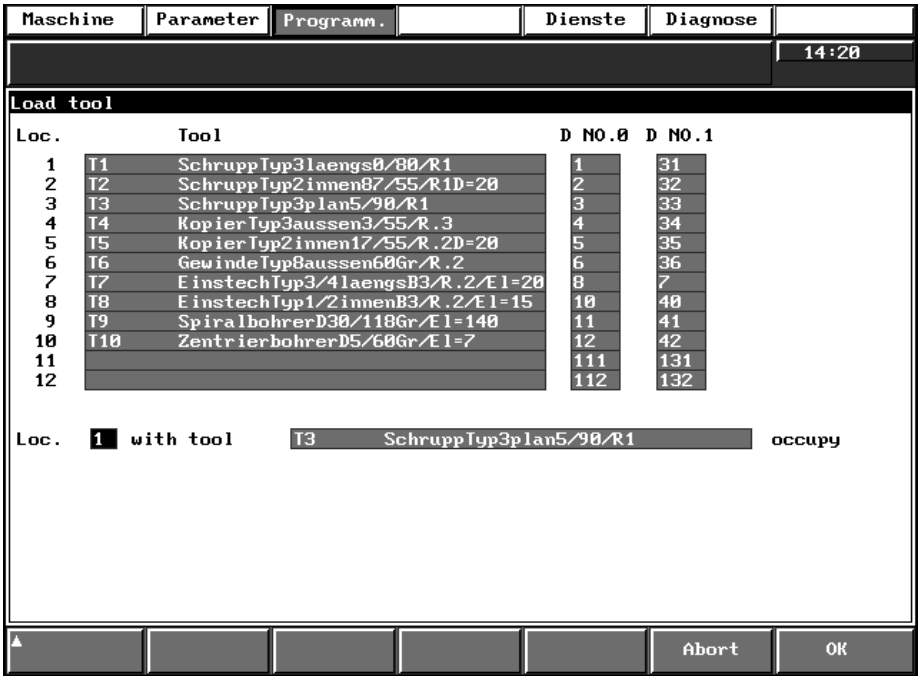


Fig. 2.12 Load tool location

2.10 R parameters

Extend the horizontal softkey bar with the **ETC key**.

R parameters

You can look at the current **R parameter** values with the R parameters function.

2.11 Modify program

Extend the horizontal softkey bar with the **ETC key**.

Modify program

Press the **Modify program** softkey to display a screen with parameters for skipping blocks and setting stop points.

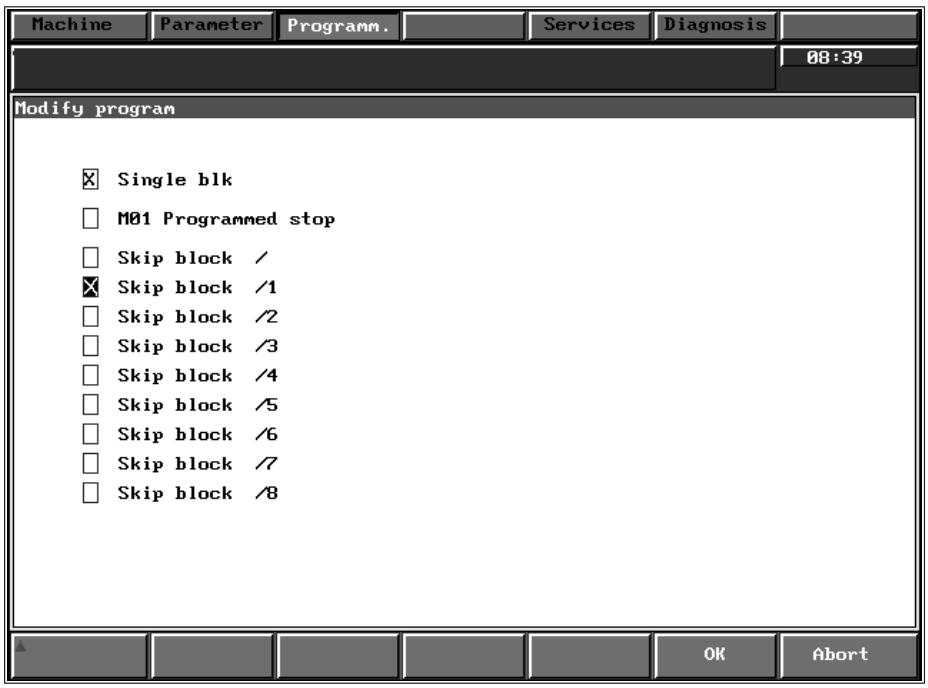
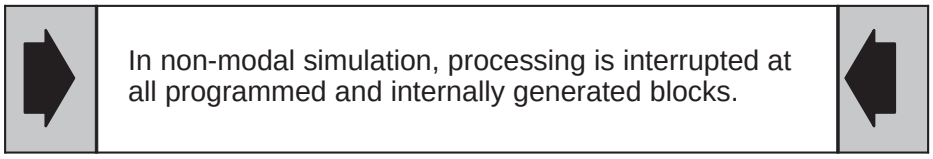


Fig. 2.13 Program modification in the simulation mode

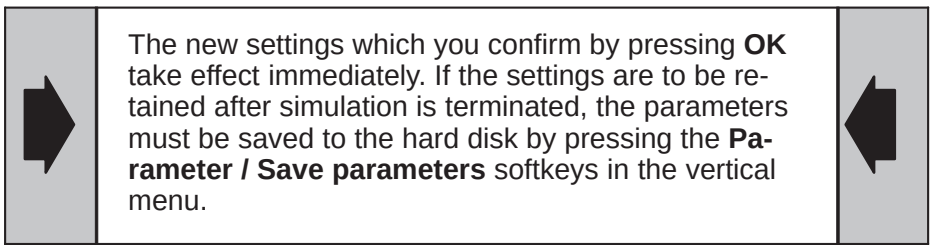
Skip blocks You can set up to 9 different skip levels (default: inactive).

Single block You can set stop points for individual blocks. All programmed blocks as well as blocks generated by the block preparation modules (e.g. tool radius compensation) are interrupted (default: inactive).



In non-modal simulation, processing is interrupted at all programmed and internally generated blocks.

M01 You can activate ”programmed stop at M01 (default: inactive).



The new settings which you confirm by pressing **OK** take effect immediately. If the settings are to be retained after simulation is terminated, the parameters must be saved to the hard disk by pressing the **Parameter / Save parameters** softkeys in the vertical menu.

2.12 Message log

Extend the horizontal softkey bar with the **ETC key**.

Message log

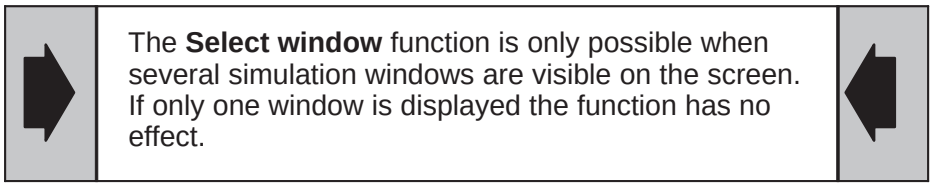
You can then look up all the displayed and suppressed messages (see Section 3.4 “Ignored message texts”) in the simulation with the function **Message log**.

The log is deleted when you terminate simulation.

2.13 Select window

Select window

Press the **Select window** softkey to select the next simulation window in clockwise direction. The selected window is surrounded by a frame.



The **Select window** function is only possible when several simulation windows are visible on the screen. If only one window is displayed the function has no effect.

2.14 Window display large

Key sequence

Window large

Press the **Window large** softkey in the vertical softkey bar to display a full screen view of the selected window. It is not possible to manipulate the picture in the large display format.

The **Window large** function only provides an enlarged view of a selected window. Existing zoomed views, view sections, etc. are not displayed; the display is simply maximized to produce a window filling the entire screen.

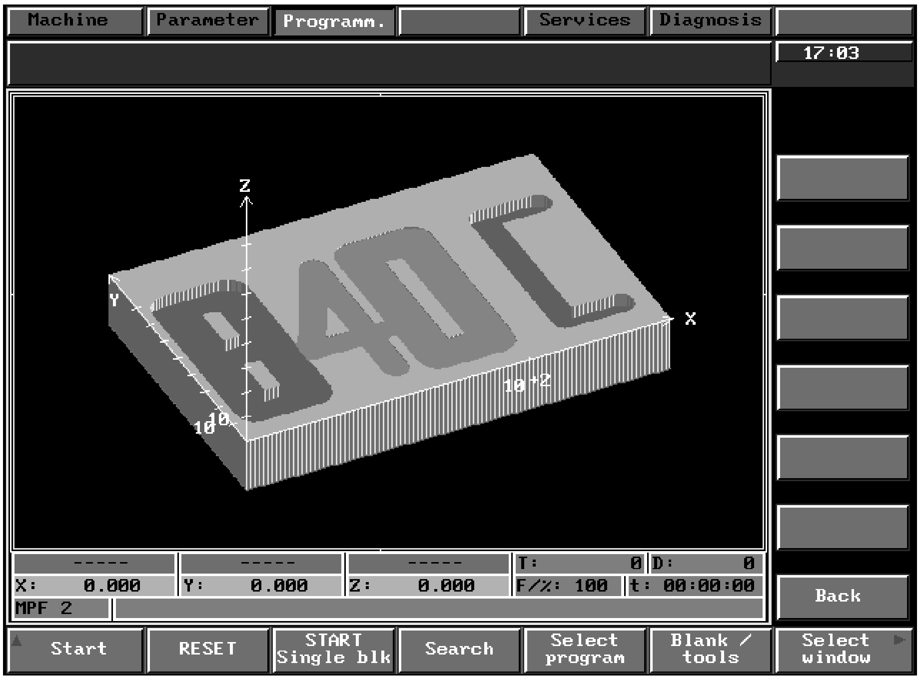


Fig. 2.14 Window display large, e.g. milling technology

2.15 Views

Views

Pressing the **Views** softkey displays a submenu for setting the number of windows, the window splitting and the rotation of the view perspectives.

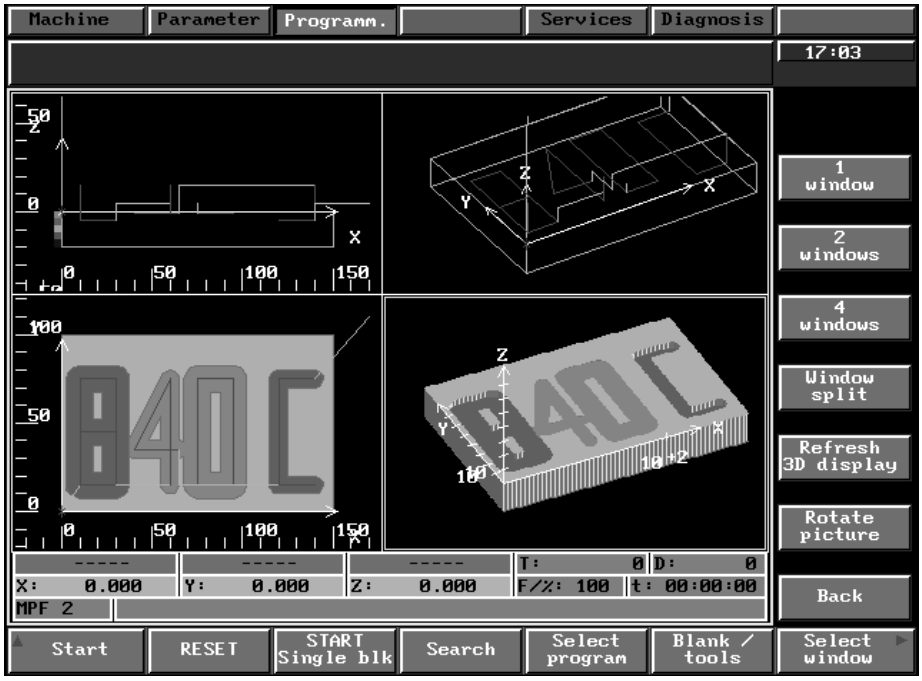


Fig. 2.15 Views, e.g. milling technology

The simulation views such as plan view, side view, etc. are entered in the **Parameter/Basic settings/ Views Select** parameter display. These settings define which views appear on the screen when you press the softkeys **1 window, 2 windows, 3 windows** or **4 windows** softkey.

2.15.1 Window views

Key sequence

1 window	Press the **1 window** softkey to display the window you have selected via the **Select window** function individually on the screen.
2 windows	Press the **2 windows** softkey to display two simulation windows which you have determined under **Parameter/Basic settings/Views select**.
4 windows	Press the **4 windows** softkey to display four simulation windows which you have determined under **Parameter/Basic settings/Views select**.
Refresh 3D display	Press the **Refresh 3D display** softkey to update all of the 3D views displayed on the screen.

Note

The **Refresh 3D display** softkey is only effective with the milling technology. Because there are no 3D views in the turning technology, this softkey is suppressed here.

2.15.2 Window split

Prerequisite

At least 2 windows must be visible on the screen.

Window split

Press the **Window split** softkey. With two simulation windows a partition line is displayed, with four windows a crosshair appears.

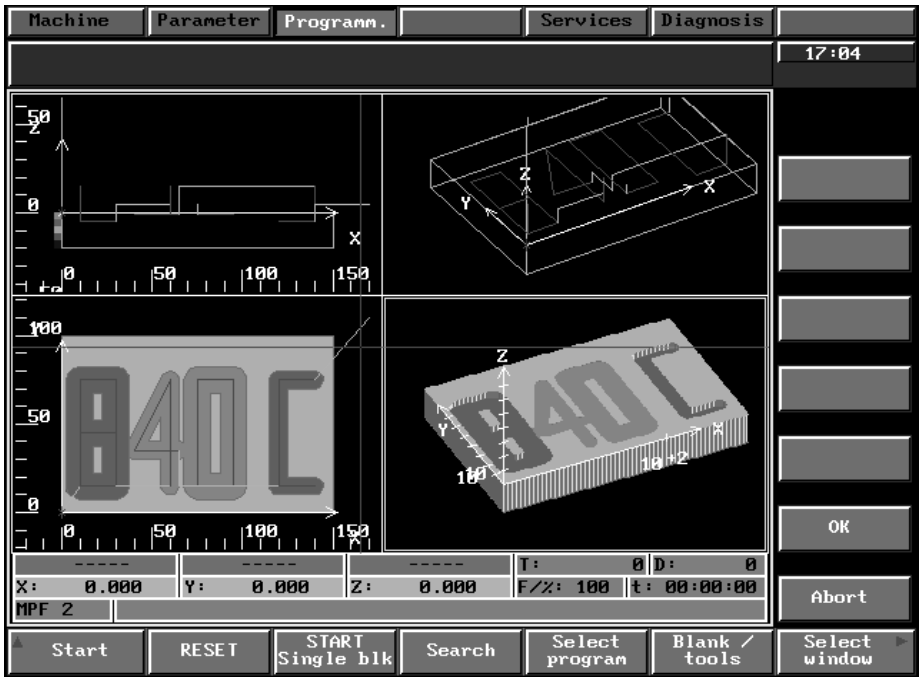
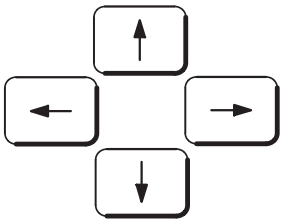
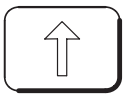


Fig. 2.16 Window split, e.g. for milling

Key sequence



Use the **arrow keys** to position the partition lines and split the windows as desired.
The partition lines move by one pixel every time an arrow key is pressed.



If you press the **shift key** in conjunction with the **arrow keys**, the displacement is larger. The displacement can be adjusted by changing the “Scroll value” setting in the **basic settings/representation** display.

OK

Press the **OK** softkey to accept the new window split. The displayed elements do not occupy the full size of the window.

Abort

Press the **Abort** softkey to cancel the new window split. The original window split is displayed again.

2.15.3 Rotate picture

Prerequisite

The selected view must be a perspective view:

- “Traversing paths view” or
- “3D view” (“milling” technology).

Rotate picture

Pressing the **Rotate picture** softkey causes the selected view to display an auxiliary window with axis intersection and unmachined contour.

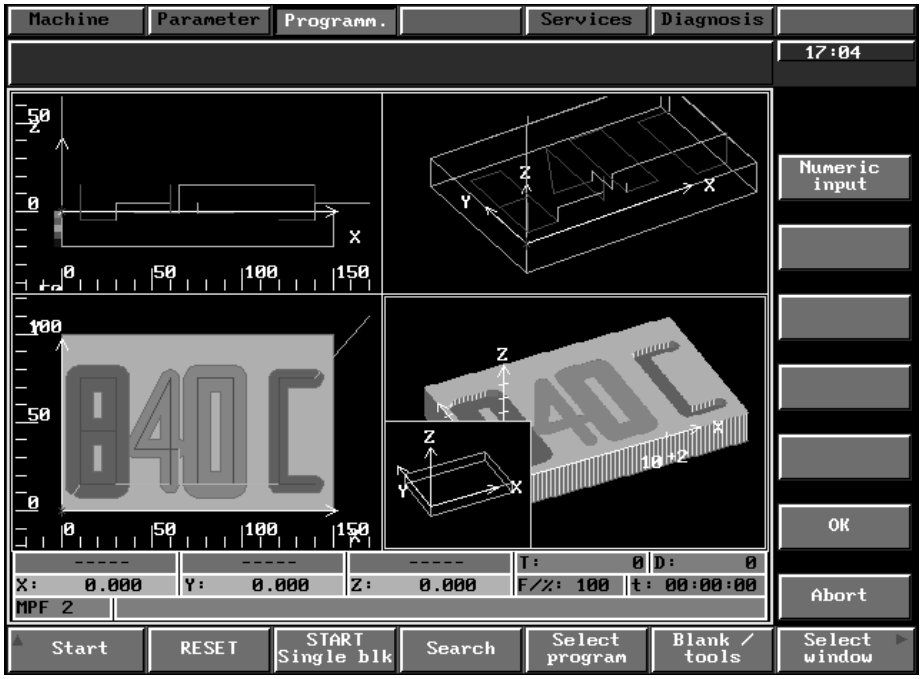
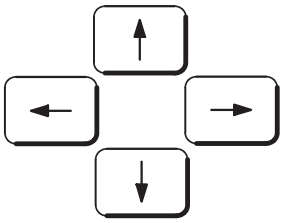


Fig. 2.17 Rotate picture

Key sequence



Use the **arrow keys** to change the viewpoint of the workpiece.

OK

Press the **OK** softkey to accept the new workpiece viewpoint. The auxiliary window disappears and the workpiece is displayed with the new viewpoint.

The new viewpoint only applies to the selected window (perspective view).

Abort

Pressing the **Abort** softkey causes the auxiliary window to disappear. The workpiece is displayed with the old viewpoint.

Numeric input

Pressing the **Numeric input** softkey allows you to enter the viewpoint directly.

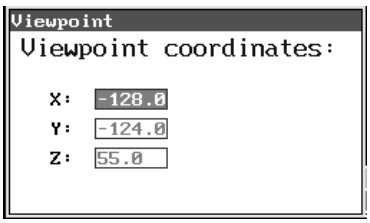
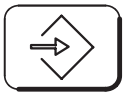


Fig. 2.18 Numeric input for setting the viewing angle for the 3D workpiece

. . .

Use the **alphanumeric keyboard** to enter the values in the input boxes and



Press the **input key** to confirm each entry.

OK

Press **OK** to accept the coordinates for the rotation of the perspective display. The numeric input window disappears and the display is updated to incorporate the changes.

Notes

- The entered coordinates are viewpoint coordinates, i.e. the reference point is the position of the observer.
- You can enter a viewpoint change in degrees in the **basic settings/ representations** parameter display. Pressing an **arrow key** causes the picture to be rotated through the set value.

2.16 Screen

Screen

Press the **screen** softkey to display a submenu for zooming, reducing and moving the contents of the selected window.

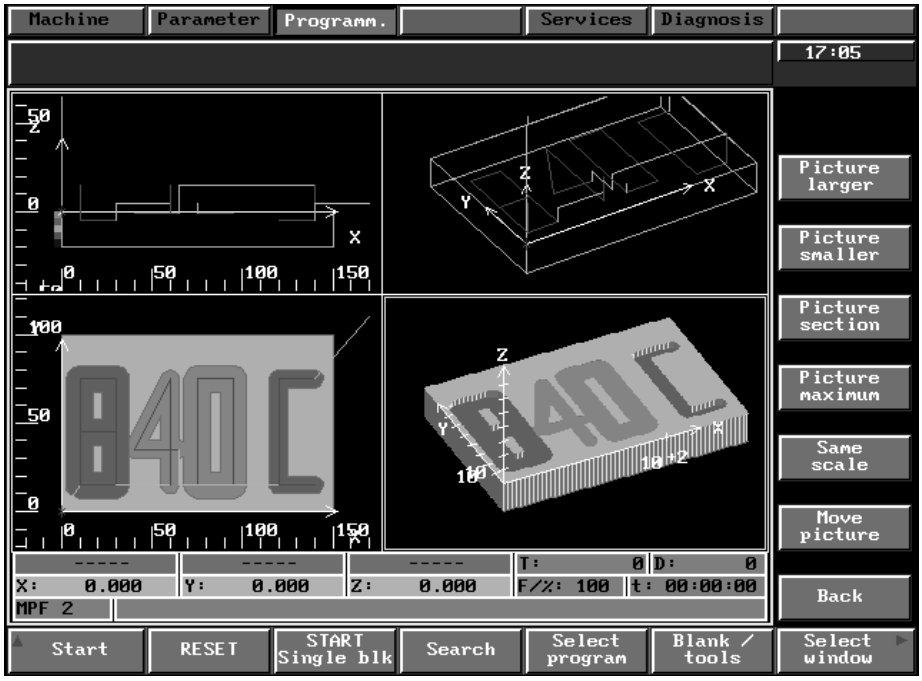


Fig. 2.19 Screen, e.g. for milling

The functions only apply to the selected simulation window.

2.16.1 Zoom functions

Picture larger

Press the **Picture larger** softkey to zoom in on the contents of the selected window.

Picture smaller

Press the **Picture smaller** softkey to make the contents of the selected window smaller.

Note

The reference point for zooming is always the axis origin.

2.16.2 Picture section

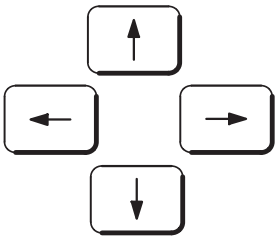
Picture section

Press the **picture section** softkey to display a frame in the selected window.

You can move, enlarge or reduce the frame.

Use this function to select the section of the display to be enlarged.

Key sequence



Use the **cursor keys** to move the frame in the desired direction. Moving the frame to a window boundary causes the contents of the window to be scrolled.
You can increase the amount of displacement by pressing the **shift** and **arrow keys** simultaneously. You can define the amount of displacement by changing the “Scroll value” setting in the basic settings/representation display.

Frame larger

Press the **Frame larger** softkey to increase the size of the frame. If the frame touches a window boundary, the contents of the window become smaller.

Frame smaller

Press the **Frame smaller** softkey to reduce the size of the frame. The frame is reduced to a minimum size defined by the system software, after which the contents of the frame are enlarged.

OK

Pressing the **OK** softkey accepts the display area set with the frame as the full window display. The frame disappears. You automatically return to the vertical basic menu.

Abort

Pressing the **Abort** softkey causes the frame to disappear.

2.16.3 Picture maximum

Picture maximum

Press the **Picture maximum** softkey for a display the size of the selected window.

2.16.4 Standardizing the scale

Prerequisite

Check that no perspective views are displayed in the selected window.

Same scale

Press the **Same scale** softkey to apply the size of the workpiece in the selected window as the scale for all of the windows.

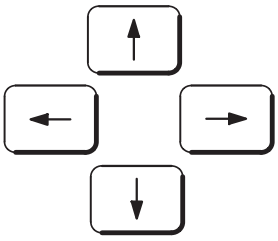
The function has no effect if a perspective traversing paths view or a **3D view** (“Milling” only) is displayed in the selected window.

2.16.5 Move picture

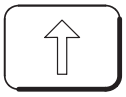
Move picture

Press the **Move picture** softkey to move the contents of the selected window.

Key sequence



Use the **arrow keys** to move the contents of the window in the desired direction.
The contents of the window are displaced by exactly one pixel.

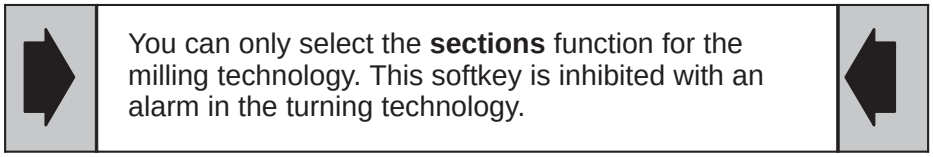


Pressing the **shift key** in conjunction with the **arrow keys** increases the size of the displacement. You can define the size of the displacement by changing the “Scroll value” setting in the **basic settings/representation** display.

Back

Press the **Back** softkey to switch back to the vertical softkey bar of the **screen** display. The picture displacement is retained.

2.17 Sections



You can only select the **sections** function for the milling technology. This softkey is inhibited with an alarm in the turning technology.

Prerequisite

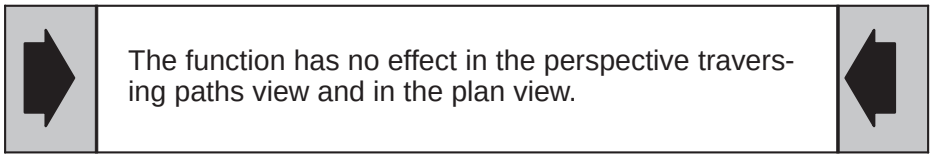
No “plan view” or perspective ”traversing paths view” should be displayed in the selected window (a section is not possible with these views).

Sections

Press the **sections** softkey to display the corresponding section view in the selected window with the cutting values defined for this view.

You can now change the cutting values using the construction lines and quadrants in the windows provided for this purpose.

You can also enter the cutting values direct in the **numeric input** screen.



The function has no effect in the perspective traversing paths view and in the plan view.

2.17.1 Defining the section in a 3D view

Prerequisite

You must first select a **3D view** (not the perspective traversing paths view) with the **Select window** function.

Key sequence

Sections

Press the **Sections** softkey.

Select window

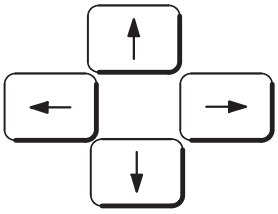
Use this softkey to select a window displaying the quadrant areas for defining the cuboid to be removed (quadrant intersection). In the corresponding 2D view a crosshair divides the display into 4 quadrants. Each selected quadrant area defines the volume to be removed in the 3D display. Consequently, 8 cuboids to be removed can be defined (plan view, side view front).

Select quadrant

Press the **Select quadrant** softkey. The next section area on the cuboid to be removed is selected; the intersection of the crosshair represents the zero point of the quadrant intersection. You select one of the four possible quadrants to define the volume to be removed. The quadrants are selected in clockwise direction. This function only applies to the 3D section display.

Note

The **Select quadrant** function can only be activated in windows displaying the quadrant areas for defining the cuboid to be removed.



Use the **arrow keys** to move the origin of the quadrant intersection in order to define the volume to be removed.

Numeric input

Pressing the **Numeric input** softkey allows you to enter the section position on the corresponding axis directly.

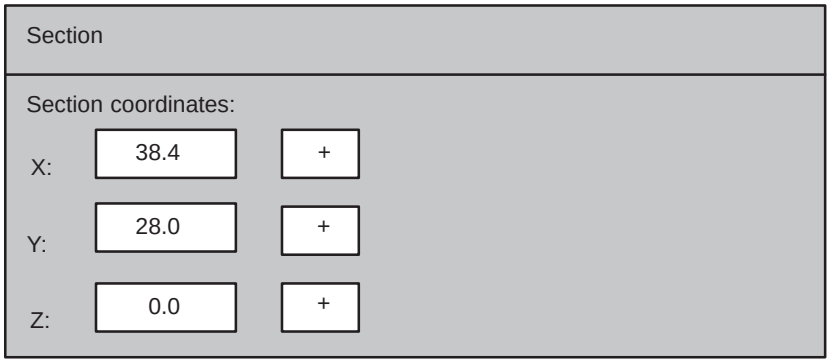


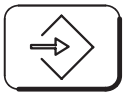
Fig. 2.20 Numeric input for 3D section position

It is necessary to define the section numerically if:

- The graphical definition is not sufficiently accurate
- No “plan view” or “side view front” is available for graphical definition.

. . .

Enter the positions using the **alphanumeric keyboard** and



confirm your entry with the **INPUT key**.

You can use the toggle fields after each input box to define the volume to be removed according to the quadrant.
A “+” sign means: The volume to be removed is generated in the positive direction of the current axis.
The “–” sign means: The volume to be removed is generated in the negative direction of the current axis.
In the 3D display, “*” represents the “–” sign.

OK

Press the **OK** softkey to accept the defined section values. The 3D view originally selected automatically becomes active and the workpiece is displayed according to the newly defined solid image.

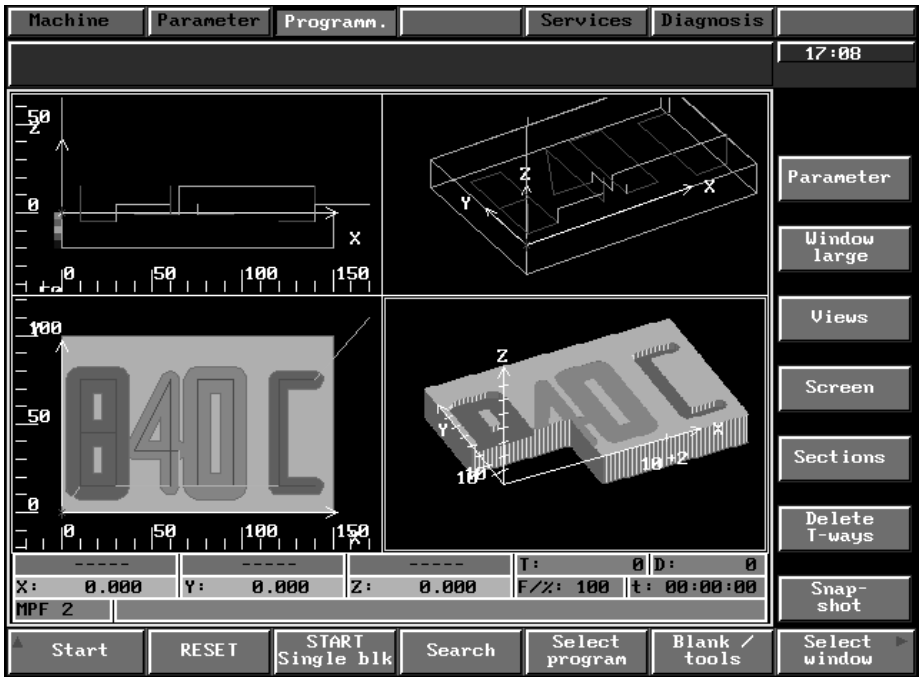


Fig. 2.21 Section for a 3D view

Abort

Press the **Abort** softkey to cancel the new volume to be removed. The original 3D view is displayed again with the old volume to be removed.

Section off

Press the **Section off** softkey to display the selected view without allowance for the section parameters.
The vertical softkey bar is reset to the status of the basic menu.

2.17.2 Section definition of a 2D view

Prerequisite
You must first select a side view with the **Select window** function.

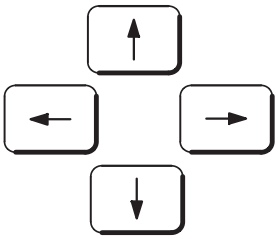
Key sequence

Sections

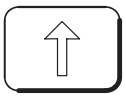
Press the **Sections** softkey.

Select window

Press the **Select window** softkey to display a window containing a graphical construction line for setting the section position.



Use the **arrow keys** to move the construction line into the new section position.
Every time an arrow key is pressed, the construction line moves by one pixel.



Press the **shift key** in conjunction with the **arrow keys** to increase the size of the displacement. You can define the displacement size by changing the “Scroll value” entry in the **basic settings/representation** display.

Note

The following is an example of a display with selected side view:

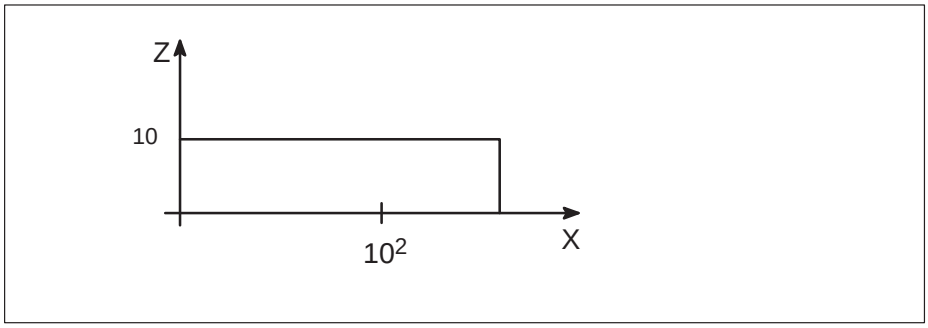


Fig. 2.22 Side view

In this view it is not possible to define the intersection line with the aid of the **arrow keys**. The intersection line for this side view lies in the X–Y plane. Use **Select window** to switch to the window where the X–Y plane is displayed. You can position the intersection line graphically in this plan view.

It is necessary to define the section numerically if:

- The graphical definition is not sufficiently accurate
- It is not possible to define the section graphically in the selected display.

Numeric input

Pressing the **Numeric input** softkey allows you to enter the section position of the corresponding axis directly.

An input window is displayed where you can enter the section position:

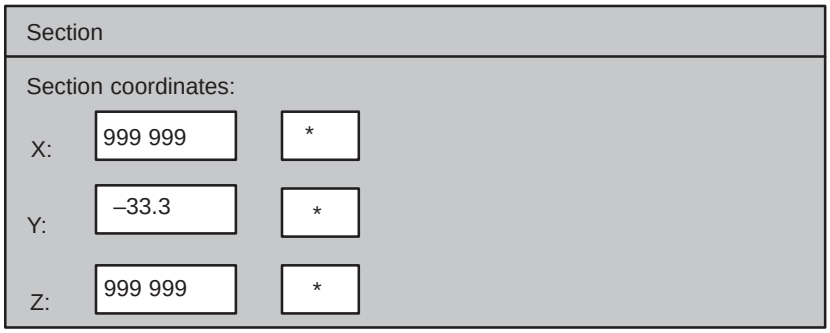


Fig. 2.23 Numeric input for 2D section position

. . .

Use the **alphanumeric keyboard** to enter the positions and

confirm each entry with the **INPUT key**.

You can enter the settings “+”, ”–” and “*” in the selection fields (toggle fields) after each input field. This allows you to define the coordinates according to each quadrant.
The leading signs in these toggle fields **only** apply to the 3D display.

The system enters “*” in these toggle fields in the **2D display** as a default setting. These defaults can be changed but this has no effect in the 2D display.

Note

Only **one** axis applies in the **2D display**. The other two axes contain the default setting 999 999 which cannot be overwritten.

If no view is available where the construction lines for defining the section position can be displayed, you can press the **Numeric input** softkey to display a window for entering the values manually.

Abort

Press the **Abort** key to cancel the new section position.
The system switches from the window with the section construction lines to the selection window.

OK

Press the **OK** softkey to accept the defined cutting value. The construction lines are removed and the current cutting section is displayed.

Section off

Press the **Section off** softkey to display the selected view without allowance for the cutting parameters.

The vertical softkey bar is reset to the status of the basic menu.

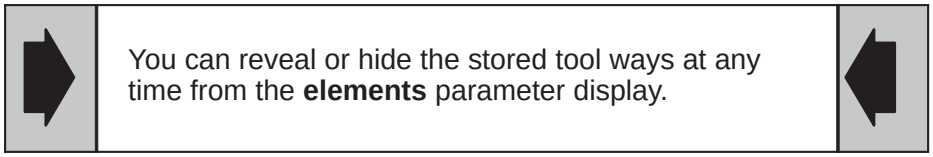
2.18 Delete T-ways

Key sequence

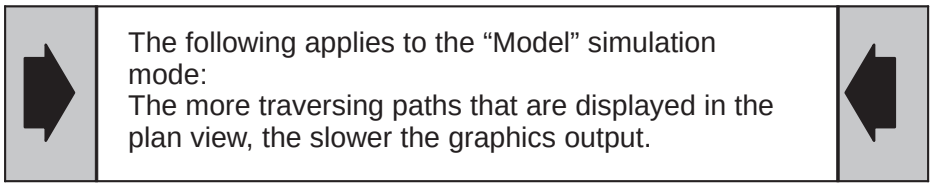
Delete T-ways

Press the **Delete T-ways** softkey to delete the tool ways created and saved up to that point.

The tool ways are even deleted if they are not visible in the current view.



You can reveal or hide the stored tool ways at any time from the **elements** parameter display.



The following applies to the “Model” simulation mode:
The more traversing paths that are displayed in the plan view, the slower the graphics output.

2.19 Snapshot

Key sequence

Snapshot

Press the **Snapshot** softkey to display the current workpiece views on the screen. The simulation is subsequently continued as set, i.e. the settings, block search with calculation, block search etc. are still active.

Note on milling

If the simulation sequence is run from beginning to end without advance and without suppressing the screen display, the 3D display can be updated with the snapshot function.
If the entry “Autom. update 3D” has been set in the **Parameter** screen, the 3D display is updated when the end of the program is reached.

Note on turning

Visualization of face end machining (view from the top) and peripheral surface machining is updated with the **Snapshot** function.

2.20 Thread display

Thread machining is only visualized in the turning technology.

Thread machining operations are indicated by outlining the area of the thread in a side view section. Tapping is not represented.

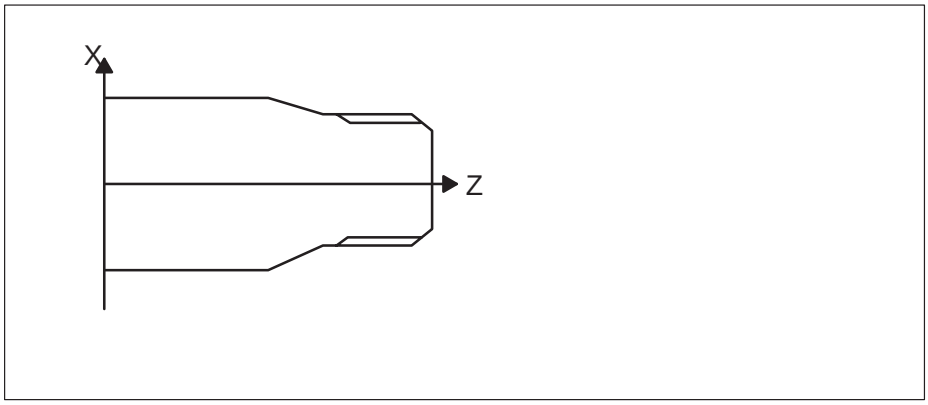


Fig. 2.24 Side view of external thread

2.21 Simulating chucks and chucking scenarios

You can hide or show chucks in the display.

Chucks are displayed

- in a frame and
- filled in in 3D representation.

A total of 10 chucks can be described. The individual chucking scenarios are described from the selection of chucks available.

A total of 99 chucking scenarios are possible.

The parameter screen **Parameter/Basic settings/System** offers the following M command default settings for chucking:

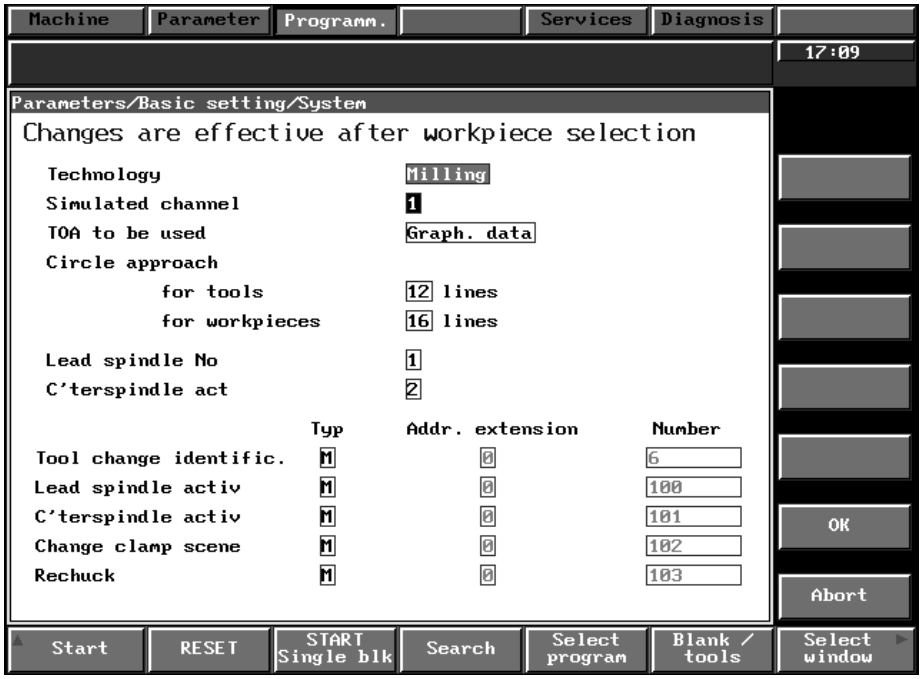


Fig. 2.25 Parameter/Basic settings/System

You can change the default settings.

You can simulate

- rechucking and
- manual rechucking.

2.21.1 Rechucking

Rechucking is always performed with a counterspindle and **not** with a tapping spindle. It is therefore **not** possible to remove stock by moving the workpiece against a stationary tool.

The operation can be performed with one slide, i.e. rechucking can be programmed with the same address letters (e.g. X, Z) for both chuck positions.

The rechucking operation is visualized by showing the stationary chucking position 2 (POS_P).

You can activate this function in the **Parameter/System** screen by entering ”Leading spindle active/counterspindle active” (see Section 3.2.1 “System”).

This operation is loaded in the shopfloor sheet under the identifier POS_P. The offset ZO2 must be entered in the shopfloor sheet under POS_P. “Normal” zero offsets such as ZO1 are allowed for automatically.

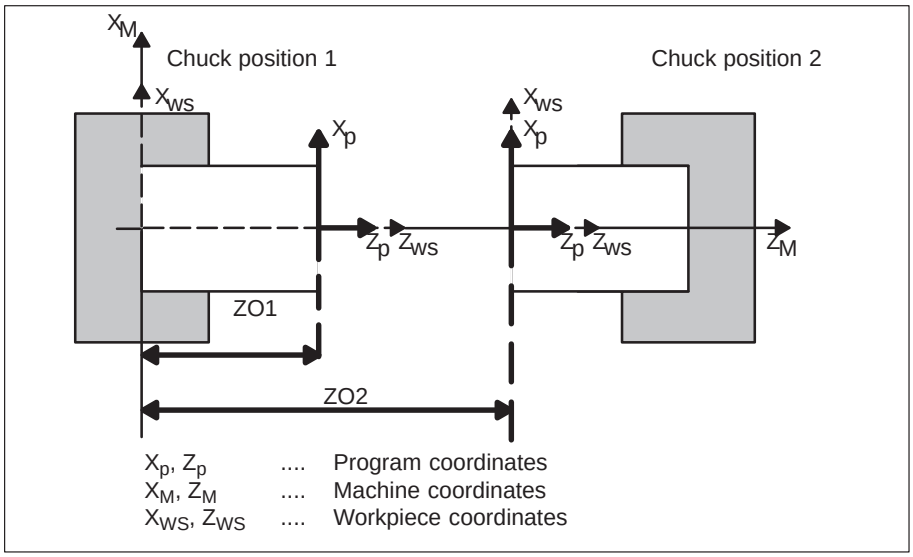


Fig. 2.26 ZO2 parameter (rechucking position POS_P)

Fig. 2.27 below shows three chucking scenarios with three different chucks.

The M commands have the following meanings:

M100 Activate main spindle

M101 Activate counterspindle

M102 Continue function for chucking scenarios

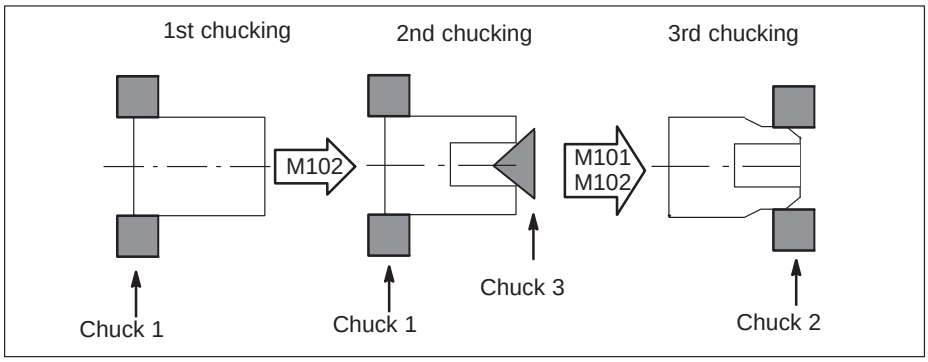


Fig. 2.27 Rechucking

2.21.2 Manual rechucking

Manual rechucking is triggered by a configurable PLC function. An angle offset is not taken into account in rechucking.

Fig. 2.28 below shows a rechucking operation using auxiliary function M103 from the side and from the face end. Both chucking scenarios must be defined, the scenarios are continued with M102.

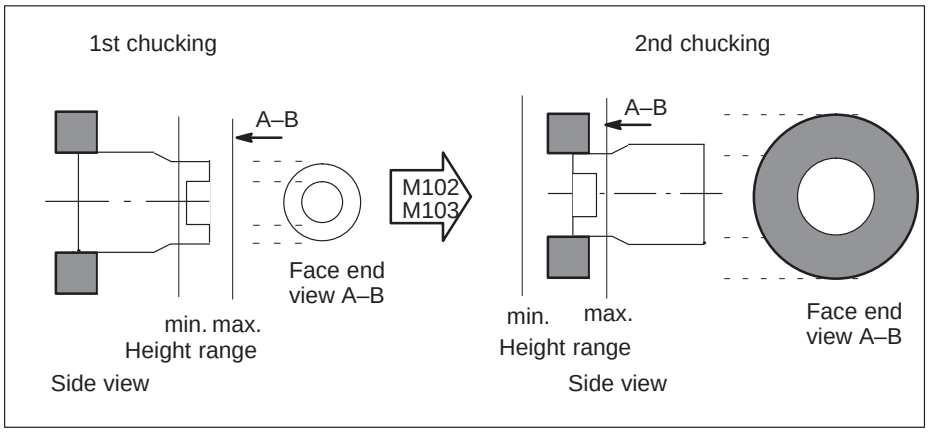
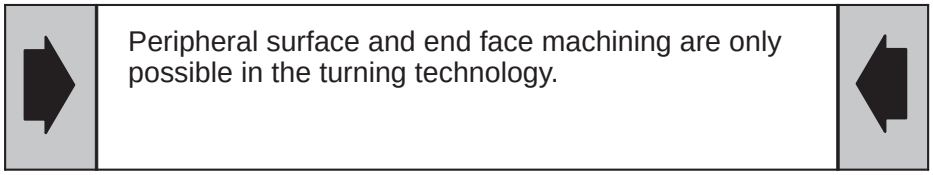


Fig. 2.28 Manual rechucking

2.22 Simulation of peripheral surface and end face machining



Peripheral surface and end face machining are only possible in the turning technology.

Select the views:

“Top”
“Left”
“Peripheral surface”
“Traversing paths”

for the 4-window representation in the **Parameter/Basic settings/ Views selection** display to visualize peripheral surface and end face machining.

The turning tool is displayed statically in the traversing paths display, the traversing paths are generated by a polymarker.

End face machining with milling paths can be visualized in the “Top” view.

The height of the color spectrum of the milling paths is defined in the **Parameter** display with the setting “Height range: Maximum/Minimum” (see Section 3.1.1).

If a milling cutter is used, the paths are traversed with a circle (milling cutter diameter).

Peripheral surface machining with milling paths is displayed in the “Peripheral surface” view for which the generating radius must be set in the **Parameter** display (see Section 3.1.1).

When simulating C axis operation, the C axis movement of the workpiece is represented as a relative movement of the tool, i.e. the tool is visualized as moving around the workpiece.

2.23 Broken–line graphics when changing part program (double–slide simulation)

During a visual inspection of double slides, the simulation graphics (traversing path, workpiece, workholder) are retained, even if another program is selected.

When a workpiece is selected again, the graphics are retained in accordance with the ”Retain graphics after channel switchover” toggle field.

The traversing paths in channel 1 to be simulated stop moving when a change is made to channel 2 to be simulated which is then traversed.

Delete T paths

Press this sofkey to delete **all** traversing paths.

RESET

Press this softkey to reset the machining steps in **all** channels.

2.24 Terminating the simulation function

Prerequisite
Key sequence

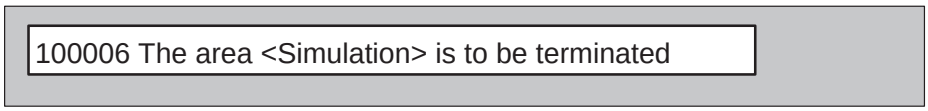
The basic screen of the **Simulation** area must be selected.

Parameter

Press the **Parameter** softkey.

End simulation

The **End simulation** option is provided in the vertical softkey bar. If you press this softkey, the following dialog box is displayed:



Abort

The operation is cancelled. The dialog box disappears.

OK

Press the horizontal **OK** softkey to terminate simulation. The system exits the **Simulation** user area and returns to the operating area last displayed.

The complete set of parameters is either saved or discarded according to the entry in the “Store parameters autom.” toggle field in the “Parameter” screen when simulation is terminated. You can save the set of parameters to the hard disk before aborting simulation using the **Save parameters** softkey function.

Notes

The operating areas

- Graphic programming
- Machine data
- Drive-servo-startup
- Programmer software and
- Simulation

cannot be run simultaneously. This means that if you switch to the **Programm. WOP** (graphic programming) area without terminating the simulation function, the system prompts you to exit simulation. These 5 areas use a large amount of memory space and can therefore not be run in parallel.

Before you close down the **Simulation** area, you must make sure that the parameter set is saved or you must save the parameters with **Save parameters**. Otherwise the data are lost.

■

3 Parameter Settings

If you wish to change

- **parameters** or
- **basic settings**

in the simulation, you can make various settings under the **Parameter** function.

3.1 Parameter

Parameter

Press the **Parameter** softkey. You can now change the simulation parameters.

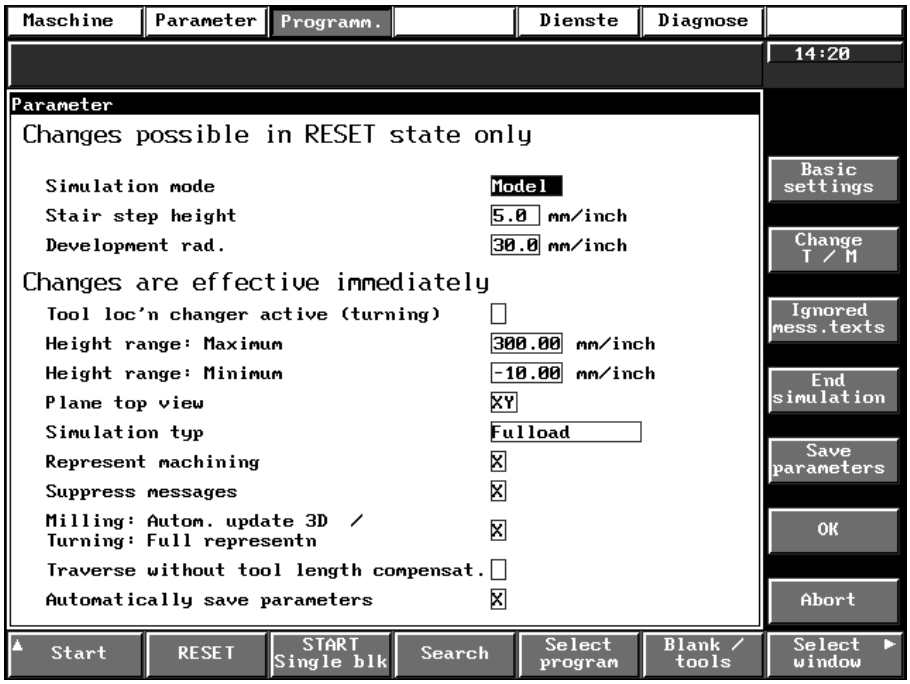


Fig. 3.1 Parameter

End simulation

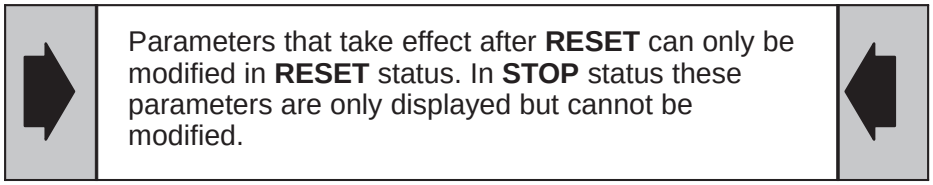
Depending on the “Store parameters autom.” entry in the toggle field, the complete set of parameters is either saved to hard disk or discarded.
If you press the **End simulation** softkey the system exits the **Simulation** operating area (see “Terminating the simulation” section).

Save parameters

Press the **Save parameters** softkey to write the current parameters to the hard disk.
The following are saved:

- All of the parameter displays
- The settings in the “Rotate Picture” window, etc.
- Modify program, block search settings
- Load list, as well as
- Message numbers to be suppressed

Save parameters writes these parameters to the SIMCONF or S_CONF_M (configuration file) files.
After installation of the simulation software, the SIMCONF or S_CONF_M files located in the Siemens branch only. **Save parameters** copies SIMCONF to the user branch with the new parameters.
You can delete SIMCONF from the directory simulation in the user branch in the **Services/Data management** operating area. File SIMCONF in the Siemens branch is again valid.



Parameters that take effect after **RESET** can only be modified in **RESET** status. In **STOP** status these parameters are only displayed but cannot be modified.

Notes

- **RESET** refers to the status invoked by pressing the **RESET** softkey in the **Simulation** area.
- Storage of the parameters is technology but not workpiece-specific, i.e. two sets of parameters exist.
- When you select a workpiece, the parameter set last saved and not the parameter set last active is used.

3.1.1 Settings in the Parameter screen that can only be modified in RESET status

Prerequisite

You have initiated the **RESET** status and the **Parameter** screen is displayed**.**

Simulation mode

Effective: After **RESET**
Parameter: Selection (toggle field)

Possible selections:

- "Model" (default)
  - The machining operations are saved internally.
  - The plan view shows the machining operations whose main level corresponds to the selected visualization level.
  - If the main machining level is a side view, the machining in this view is displayed in a frame.
  - Different colors are used to distinguish between the various machining depths.
  - Machining in the main level with simultaneous infeed in the 3rd axis is resolved in "steps". Step resolution does not take place for the traversing paths however.
  - The machining of all main levels is included in the 3D view.
  - The workpiece is displayed in the plan view with a fill pattern.
- "T paths" (traversing paths)
  - Only the traversing paths are stored.
  - In the 3D view only the unmachined part is displayed.
  - The workpiece is displayed inside a frame.

Stair step height

Effective: After **RESET**
Parameter: Value input
Value range: "0.01 – 99.99 mm/inch".

The parameter is only effective in the **Model** simulation mode.

- "5 mm/inch" (default)
  - The parameter defines the raster for converting a "inclined" machining path into a step movement.

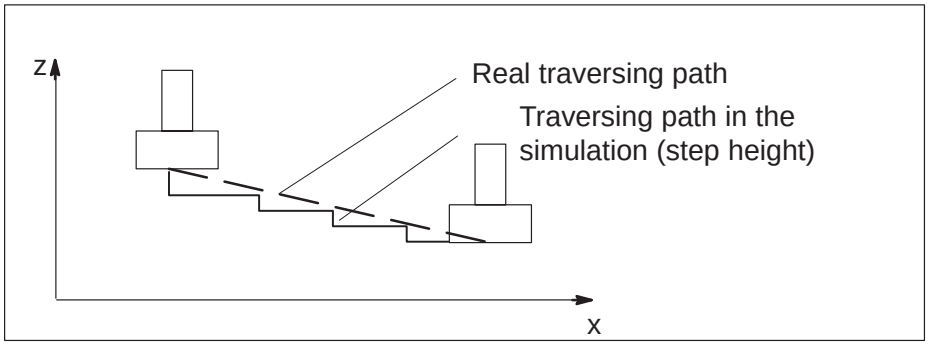


Fig. 3.2 Step resolution for "Inclined tool movement"

In the **Model** simulation mode:
The smaller the stair step height selected, the larger the computing time required for “inclined” traversing movements.

Generating radius

The setting for generating radius is of relevance to the turning technology only.

Parameter: Value input
Value range: “0.01 ... 9999 mm/inch”

This parameter defines the height of the generated surface which is displayed in the “Peripheral surface” view.

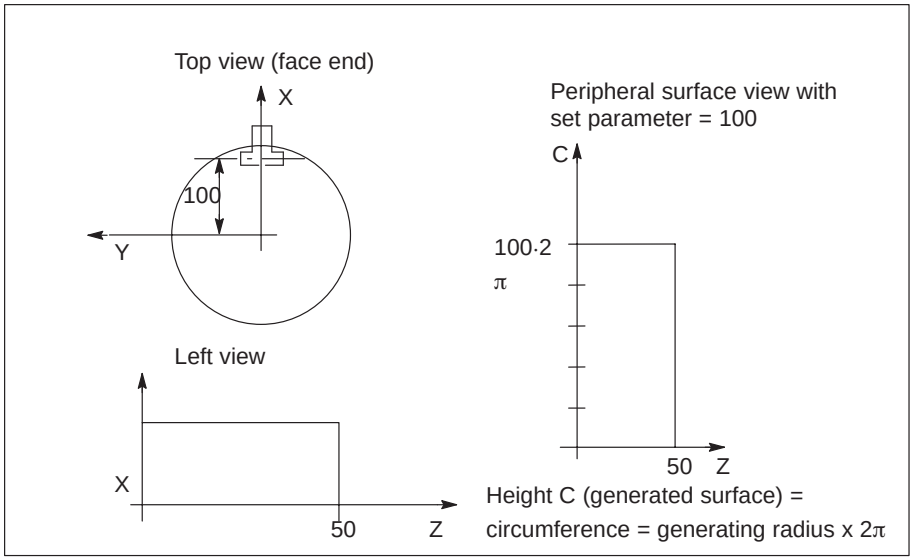


Fig. 3.3 Degree of generation parameter

Please note:
The generating radius must always be smaller than or equal to the workpiece radius.

If the generating radius > workpiece radius, nothing appears in the peripheral surface view.

Note

If a machining operation is performed on the peripheral surface, the machining path is automatically displayed in the “Machining plane 10” color setting (see “Color” Section).

The “Height range: Maximum/Minimum” settings are only active in the turning technology and after **RESET** (or when the simulation function is in the **RESET** state).

An explanation of the entries for the milling technology is given in Section 3.1.2.

Height range: Maximum

Effective: After **RESET** (only with turning)
Parameter: Value input
Value range: “–500.00 mm/inch to +500.00 mm/inch”

Height range: Minimum

Effective: After **RESET** (only with turning)
Parameter: Value input
Value range: –500.00 mm/inch to +500.00 mm/inch

Notes for turning technology

The color spectrum automatically adapts itself to the height range, no settings have to be entered. Machining is visualized with the color spectrum in the ”TOP VIEW” (face end) for machining operations performed in this plane.

Caution!
If a height range that is outside the workpiece is defined in turning, no color spectrum is visualized.

Example of a height range definition for turning (the height range is linked to the workpiece):

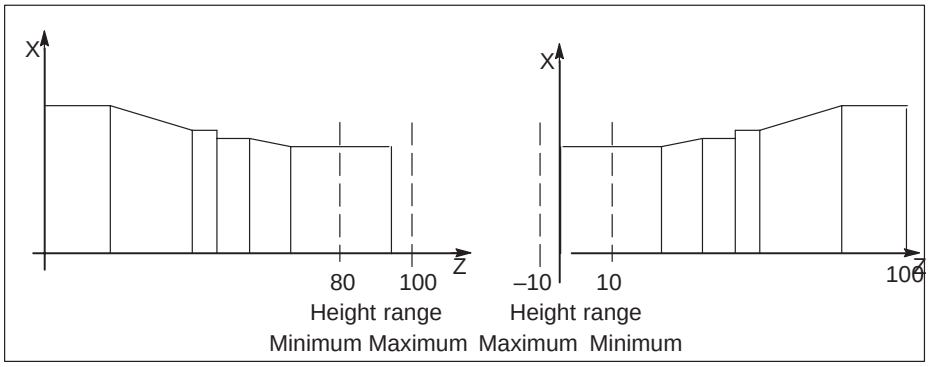


Fig. 3.4 Height range definition

3.1.2 Settings in the Parameter screen that take effect immediately

The Height range: Maximum/Minimum settings are effective immediately only with the milling technology. An explanation of the entries for the turning technology is given in Section 3.1.1.

Height range: Maximum

Effective: Immediately (only with milling)
Parameter: Value input
Value range: “–500.00 mm/inch to +500.00 mm/inch”

- “300.00 mm/inch” (default “turning”)
- “00.00 mm/inch” (default “milling”)

  Upper limit for the color identification of machining.

Height range: Minimum

Effective: Immediately (only with milling)
Parameter: Value input
Value range: “–500.00 mm/inch to +500.00 mm/inch”

- “–10 mm/inch” (default “turning”)
- “–20 mm/inch” (default “milling”)

Notes

In the **Parameter/Basic settings/Color** display, you can set the colors for the machining levels.

The **minimum value** input for the **height spectrum** must be smaller than the **maximum value**.

The colors are divided evenly over the selected height spectrum. The color scale is only displayed within the height range of the workpiece. If the bottom workpiece coordinate is less than the minimum value for the color spectrum, the lowest color range is used until the end of the workpiece. The same applies to the top coordinate of the workpiece.

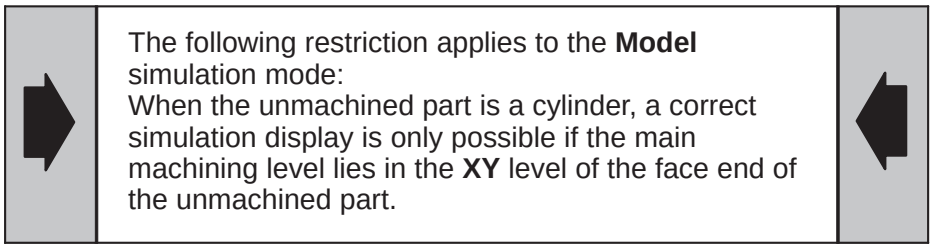
Top view

Effective: Immediately
Parameter: Selection (toggle field)

The default setting “XY” **cannot** be changed for the turning technology.

Possible selections:

- “XY” (default)
  - Selection of the level to be displayed in plan view.
  - Machining operations are displayed whose main level corresponds to the selected level. If no machining is visible, then no machining took place in this main level or it was simulated in the traversing paths mode.
- “YZ”
- “ZX”



The following restriction applies to the **Model** simulation mode:
When the unmachined part is a cylinder, a correct simulation display is only possible if the main machining level lies in the **XY** level of the face end of the unmachined part.

Simulation type

Effective: Immediately
Parameter: Selection (toggle field)

Possible selections:

- “Full load” (default)
  - The simulation is performed at the maximum speed possible, i.e. completely independently of the programmed feedrate.

  The speed can be influenced by the function **Override+** or **Override–** or by **Parameter** “Representation precision” (for 100% override).
- “Timed”
  - The simulation is timed, i.e. it is performed at the speed defined by the programmed feedrate and the selected override.
  - If, in timed mode, the machining speed is too high for the correct calculation of the simulation output, the screen output may be suppressed in order to maintain the timing accuracy of the simulation. In extreme cases, only the end result of the simulated machining is visualized. You can synchronize the calculation and the output by reducing the override value.

Display machining

Effective: Immediately
Parameter: Selection (toggle field)

Possible selections:

- "Y" (default)

  The visualization frequency is adjusted with the override.

- "N"

  The finished machined workpiece is displayed after the program has been executed.

Note

The setting "Yes" is represented by an "X" in the toggle field.
The setting "No" is represented by an empty field in the toggle field.

Suppress messages

Effective: Immediately
Parameter: Selection (toggle field)

Possible selections:

- "N" (default)

  All messages such as "No tools", "No blank" etc. are displayed and must be acknowledged.

- "Y"

  All messages in the list **Ignored message texts** are suppressed but listed in the log (see Section 3.4 "Ignored message texts").

If you have set the **Suppress messages** field in the **Parameter** screen, all the messages that are marked to be suppressed are not displayed on the screen.

All messages, even those that are suppressed, are listed in a log by the simulation function.

Automatic refresh 3D

Effective: Immediately (only of relevance for milling)
Parameter: Selection (toggle field)

Possible selections:

- "Y" (default)

  The 3D view is updated after every simulation **STOP**.

- "N"

  If the simulation switches to **STOP** status, **no 3D refresh** takes place.

Full display

Effective: Immediately (only of relevance for turning)
Parameter: Selection (toggle field)

Possible selections:

- “Y” (default)

  Rotational symmetrical full representation of the view left

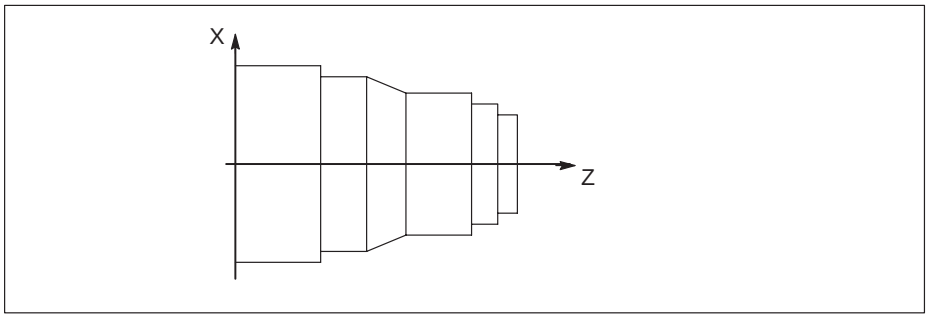


Fig. 3.5 Rotational symmetrical full representation of the view left

- “N”

  Rotational symmetrical half representation of the view left

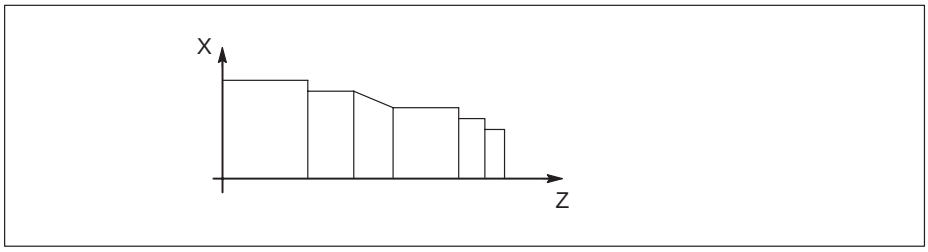


Fig. 3.6 Rotational symmetrical half representation of the view left

Traversing without TLC

Effective: Immediately
Parameter: Selection (toggle field)

Possible selections:

- “No” (default)

  Tool traverses with tool length compensation.

- “Y”

  Tool traverses without tool length compensation.

Store parameters automatically

Effective: Immediately
Parameter: Selection (toggle field)

Possible selections:

- “Y” (default)

  The parameters are saved when simulation is terminated.

- “N”

  The parameters are not saved automatically when simulation is terminated.

3.2 Basic settings

Basic settings

Press the **Basic settings** softkey to change the settings for simulation:

- **System**
- **Representation**
- **Elements**
- **Colors**
- **Views choice**
- **Load list**

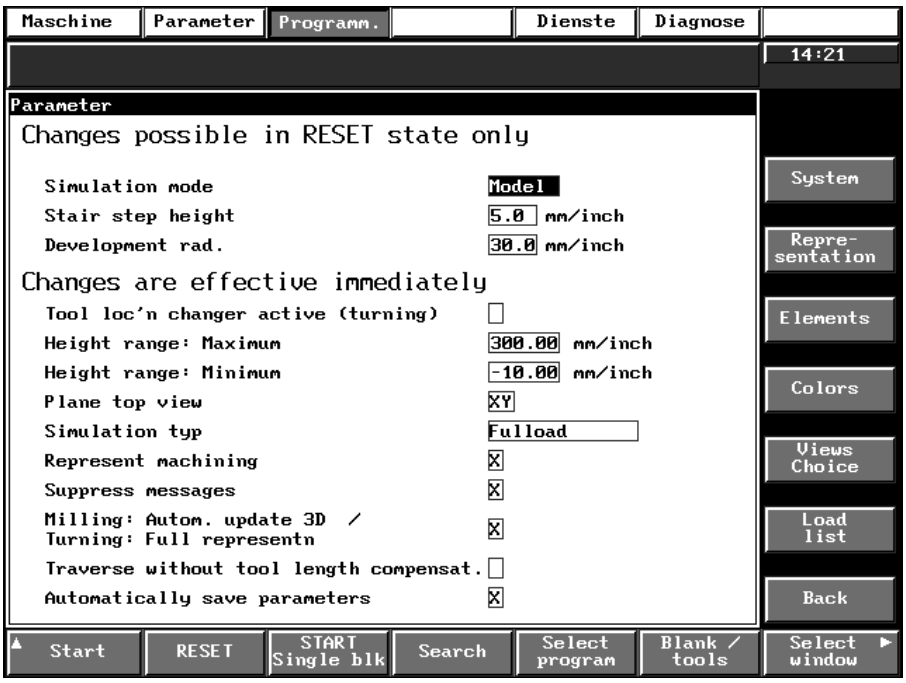


Fig. 3.7 Basic settings

3.2.1 System

System

Press the **System** softkey. You can now change the simulation parameters.

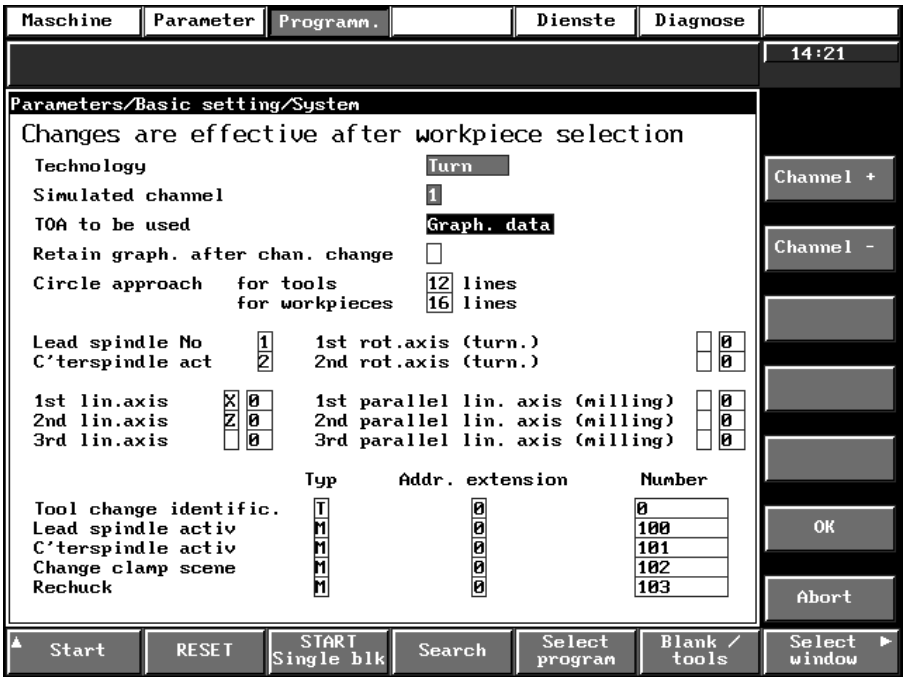


Fig. 3.8 SYSTEM settings in the simulation area

The modifications in the **Parameter/Basic settings/ System** display only take effect when the next workpiece is selected. The modified parameters are displayed, but only take effect after the modification when you call up a workpiece.

The parameters in the **Parameter/Basic settings/ System** display are saved to hard disk immediately after every modification, regardless of the setting in the "Store parameters automatically" toggle field in the "**Parameter**" display.

Technology

Parameter: Display field

- "Milling" or
- "Turning"

Note:

Selection possible under function **Change T/M**.

Axes to be simulated

Use the ”Channel+” and ”Channel–” softkeys to select the NC channel (value range ”1 to 6”) whose data are to be used for simulating.

The NC axes to be simulated are stored according to the channel.

TOA to be used

Parameter: Selection (toggle field)

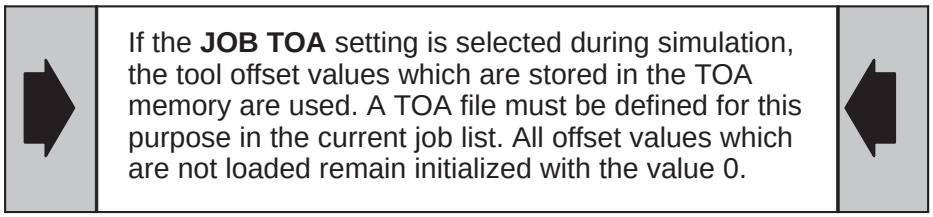
Possible selections:

- “Graphic data” (default)

  The offset values are generated automatically from the graphical description of the tools.
- “JOB TOA”

  The offset values of the TOA memory are used.

  The effect of the settings is described in Section 4.3.4.



If the **JOB TOA** setting is selected during simulation, the tool offset values which are stored in the TOA memory are used. A TOA file must be defined for this purpose in the current job list. All offset values which are not loaded remain initialized with the value 0.

Circle approach for ***tools***

Parameter: Selection (toggle field)

Possible selections:

- “12 lines” (default)
  - The number of lines defines the representation of a circle according to circle secants. The larger the number of lines, the better the approach of the secant generation to a circle and consequently the longer the processing time required by the simulator for a computing task.
  - The setting is of relevance for the drill and mill tools.
- “8 lines”
- “16 lines”
- “20 lines”
- “24 lines”

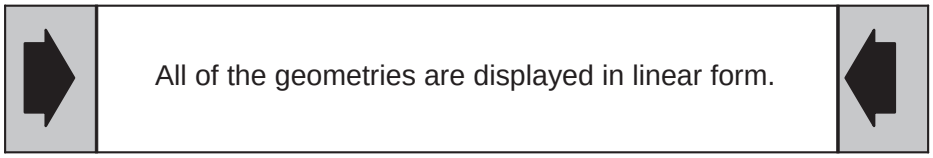
Circle approach for ***workpieces***

This setting only applies to the selection of the unmachined part (cylinder, contour).

Parameter: Selection (toggle field)

Possible selections:

- “16 lines” (default)
- “20 lines”
- “24 lines”
- “28 lines”



Tool change identifier
type

Parameter: Selection (toggle field)

Possible selections:

Selection of the identification letters for the tool change.

- “M” (default for “milling”)
- “T” (default for “turning”)
- “H”

“M”**,** “H” ... Evaluation of the address extension and number

“T” ... Instant tool change

Note: Tools are represented graphically only if the tool change identifier in the part program is identical with the identifier set here.

Tool change identifier
address extension

Parameter: Value input (input field)
Value range: “0 to 9”

- “0” (default)

Selection of the address extension for the tool change.

Tool change identifier
number

Parameter: Value input (input field)
Value range: “0 to 99999999”

- “6” (default for “milling”)
- “0” (default for “turning”)

Selection of the number for the tool change.

Leading spindle active

Parameter: Selection (toggle field)

Possible selections:

- “M”
- “H”

Value range for number: “0 to 99999999”

- “M100” (default)

Counterspindle active

Parameter: Selection (toggle field)

Possible selections:

- M
- H

Value range for number: “0 to 99999999”

- “M101” (default)

Continue chucking

Parameter: Selection (toggle field)

Possible selections:

- “M”
- “H”

Value range for number: “0 to 99999999”

- “M102” (default)

Counter chucking

Parameter: Selection (toggle field)

Possible selections:

- “M”
- “H”

Value range for number: “0 to 99999999”

- “M103” (default)

3.2.2 Representation

Representation

Press the **Representation** softkey. You can now modify the simulation parameters.

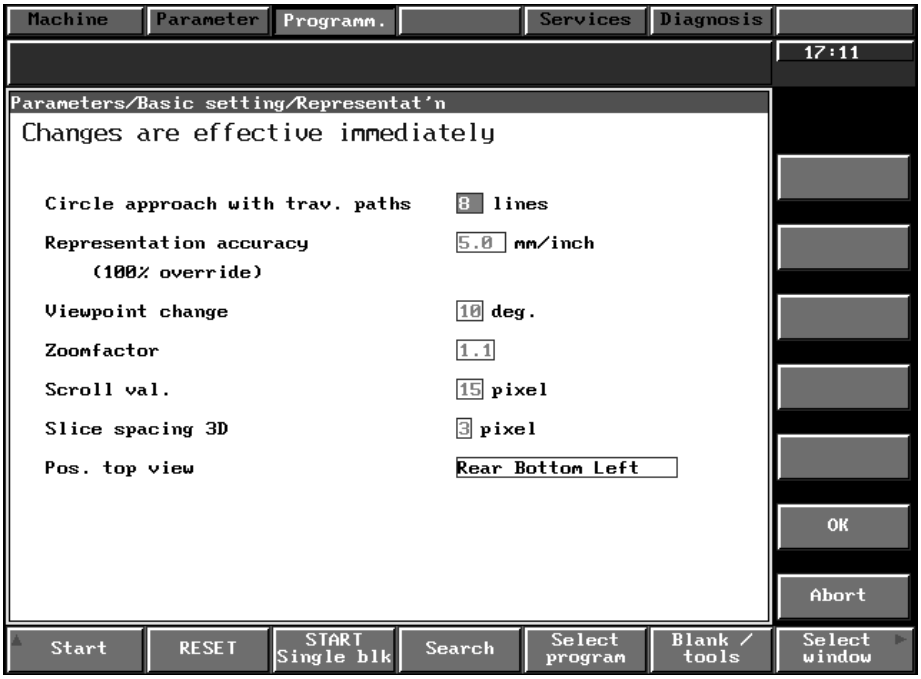


Fig. 3.9 Representation settings in the simulation area

Changes to “Representation parameters” take effect immediately. You accept with **OK**. If you want to retain the modifications after terminating simulation, you must save the parameters to the hard disk.

Circle approach for traversing paths

Parameter: Value input (input field)
Value range: “8 to 64 lines” (values can be input in steps of 1)

- “16” (default)

  This value defines the accuracy with which the arc of a traversing path is approached using lines.

Representation accuracy (100% override)

Parameter: Value input (input field)
Value range: “0.01 mm/inch to 99.99 mm/inch”

The parameter only applies to the **full load** simulation type.

- “5 mm” (default)

  The representation accuracy is a measurement of the representation frequency of the simulation graphics. For example, a representation accuracy of 2 mm means that the simulation graphics are updated (the display is refreshed) when the tool has traversed through 2 mm.

Viewpoint change

Parameter: Value input (input field)
Value range: “1 to 90 degrees”

- “15 degrees” (default)

  The angle for the **Rotate picture** function is entered in the **Viewpoint Change** parameter.

Zoom factor

Parameter: Value input (input field)
Value range: “1.1 to 1.5”

- “1.1” (default)
  - The factor for the **Picture larger/smaller** functions is entered in the “Zoom Factor” parameter.
  - The reciprocal of the input value is used for the **Picture Smaller** function.

Scroll value

Parameter: Value input (input field)
Value range: “1 to 50 pixels”

- “15 pixels” (default)

  The value for scrolling with the **Shift** and cursor keys in the **Window split**, **Sections** and **Move picture** functions is entered in the “Scroll value” parameter.

Slice spacing 3D This setting applies to milling only

Parameter: Value input (input field)
Value range: “1 to 9 pixels”

- “3 pixels” (default)
  - The 3D display is constructed from a series of slices. The “Slice spacing 3D” entry defines the pixel spacing of the slices.
  - The larger the entry, the faster the construction of the 3D display, because fewer slices are required.

Position top view

Parameter: Selection (toggle field)

Definition of the position of the axis intersection origin.

Possible selections:

- “Rear bottom left” (default for milling)
- “Rear top left”
- “Rear top right”
- “Rear bottom right” (default for turning)
- “Front bottom left”
- “Front top left”
- “Front top right”
- “Front bottom right”

The axis origin is defined with the aid of a cube model:

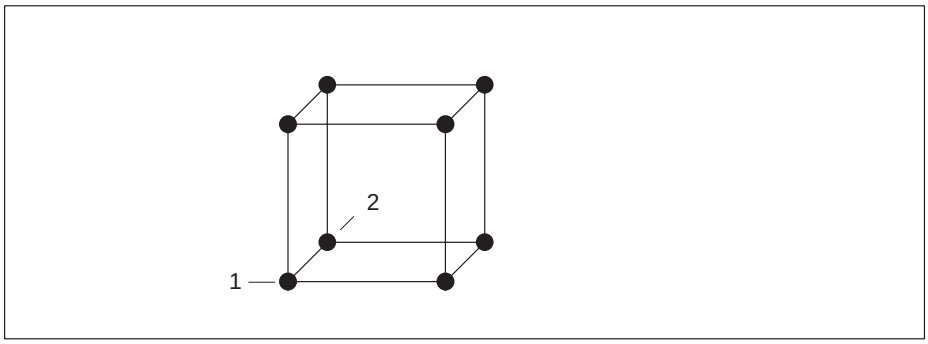


Fig. 3.10 Cube model

The corner points of the cube represent the possible selections.

For example, point **1** “Front bottom left” (see Fig. 3.10) defines the axis intersection for the plane of “Plan view XY” as follows:

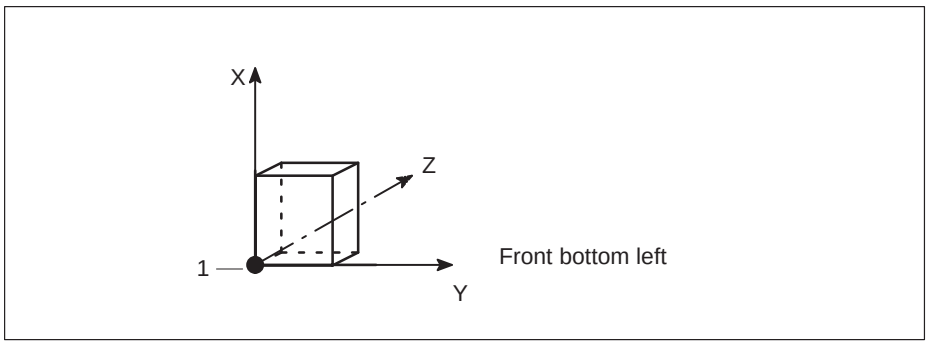


Fig. 3.11 Front bottom left

Point **2** “Rear bottom left” (see Fig. 3.10) defines the axis intersection for the plane of ”Plan view XY” as follows:

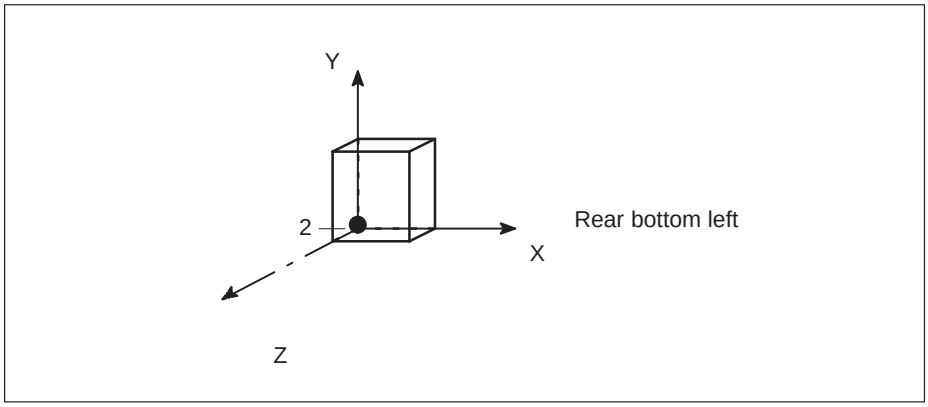


Fig. 3.12 Rear bottom left

3.2.3 Elements

Elements

Press the **Elements** softkey. You can now modify the settings of the visible elements for.

- “Workpiece”
- “Finished part”
- “Chuck”
- “Tool”
- “T paths” and
- “Axis intersection”

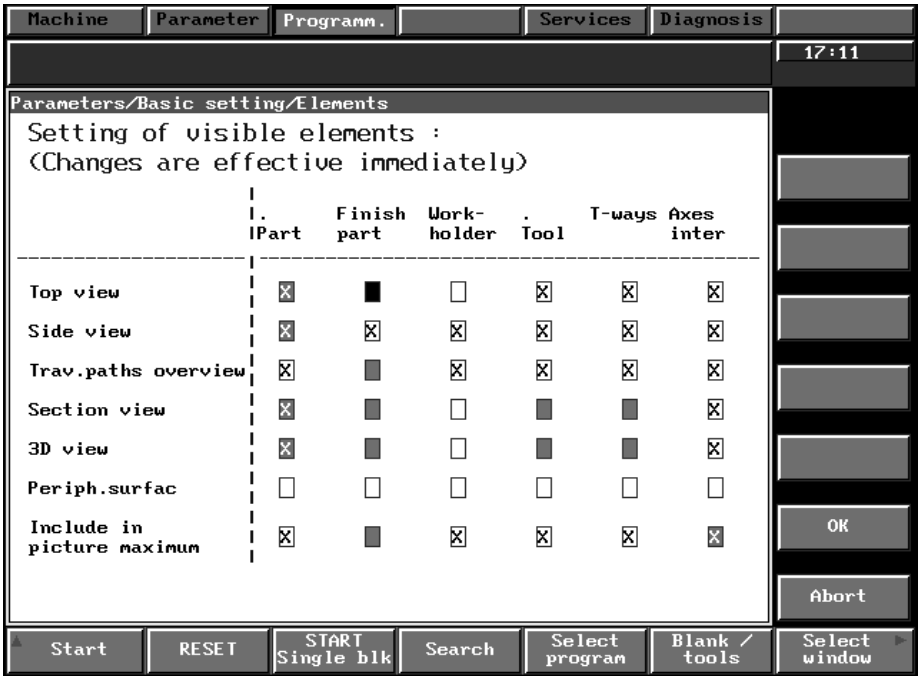


Fig. 3.13 Elements settings in the simulation area

Modifications in the elements parameters take effect immediately after you accept with **OK**. If you want the modifications to be retained after terminating simulation, you must save these parameters to hard disk.

In the **elements** parameter display, you can define which elements are to be displayed in the specified views. Boxes skipped by the input cursor are only provided for information purposes (fixed settings).

You can also define which geometries are included in the **picture maximum** function**.** The settings apply to all the views, but are only effective if the geometries are visible in the corresponding views. Allowance is generally not made for the axis intersection in **picture maximum,** i.e. the axis intersection is always adapted to the window.

3.2.4 Color

Color

Press the **Color** softkey. You can now modify the color settings.

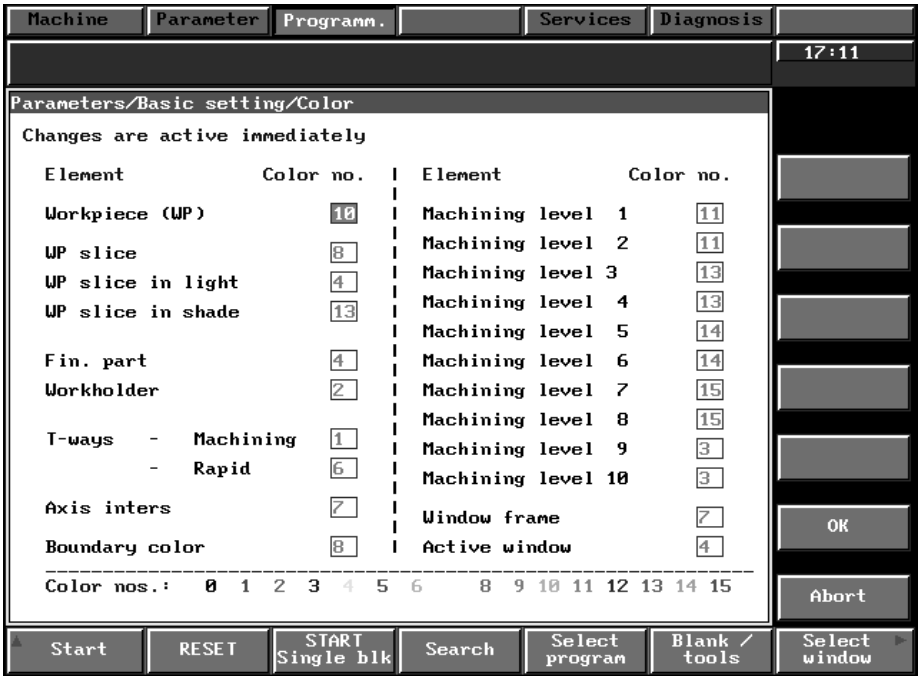


Fig. 3.14 Color settings in the simulation area

Changes to the **Color** parameters take effect immediately after you accept with **OK**. If the modifications are to be retained after the simulation is terminated, you must save these parameters to hard disk.

The color settings are defined according to the color numbers shown.

The colors that the numbers represent are shown in the color scale under “color numbers”.

3.2.5 Views choice

Views choice

Press the **Views choice** softkey. You can now define the window views for the functions

- **1 window**
- **2 windows**
- **4 windows**

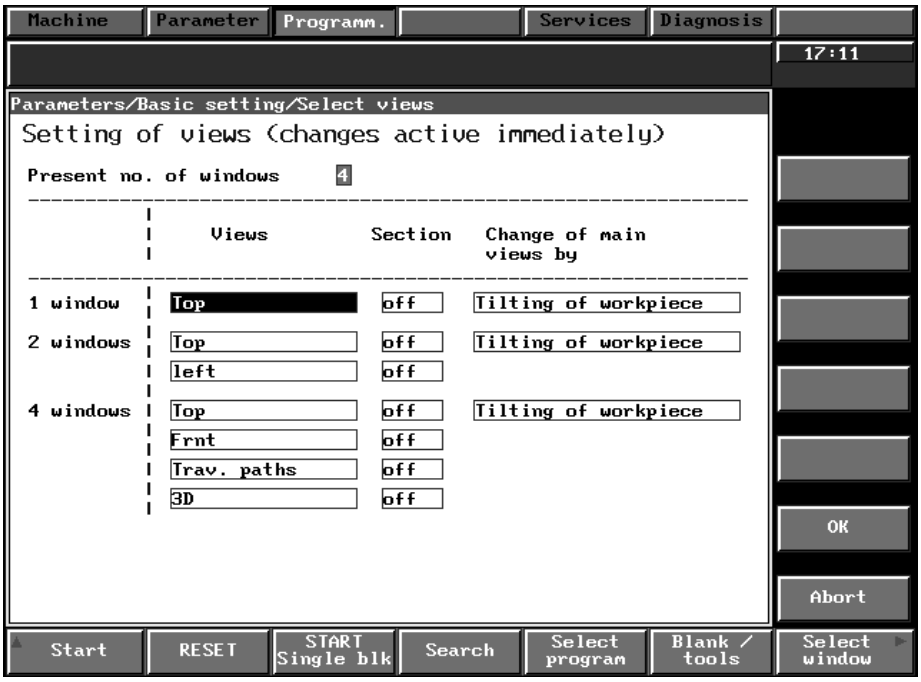


Fig. 3.15 View settings in the simulation area

The **Section (On/Off)** parameter is only effective if a section is possible in the corresponding view.

Changes to the **Views choice** parameters take effect immediately after you accept with **OK**. If the modifications are to be retained after simulation is terminated, you must save these parameters to hard disk.

Present number of windows

This parameter is displayed only .
Value range: “1, 2, 4”.

It is only possible to change the number of windows with the softkey functions **1, 2** and **4 windows**.

Views

Parameter: Selection (toggle field)

Possible selections:

- “Top”, “Bottom”
- “Front”, “Rear”, “Left”, “Right”
- “Traversing paths”
  (Display of the contours of the unmachined part, the traversing paths and the tool as a polymarker)
- “3D” (for “milling” technology only)
  (Momentary picture **without** the tool, machining result only)
- “Peripheral surface” (for turning technology only)
  (Display of the generated surface of the turned part, see Section 3.1.1).

Notes

The technological-specific views are displayed in the other technology as “traversing paths” views:
“3D” corresponds to “traversing path view for technology turning” and “peripheral surface” corresponds to “traversing path view for technology milling”.

1 window default

- “Top” (“milling”)
- “Left” (“turning”)

2 windows default

- “Top (1)”
- “Left (2)”

The display is as follows:

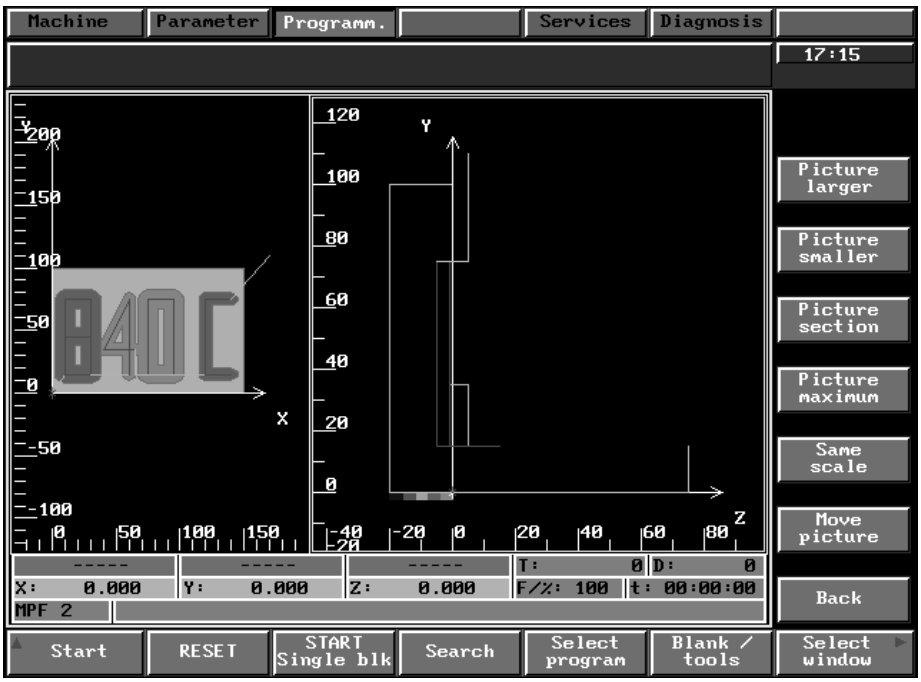


Fig. 3.16 2 windows default

4 windows default:

– “Top (1)”

– “Front (2)” (“milling”), “Left” (“turning”)

– “Traversing paths (3)” (“milling”), “peripheral surface” (“turning”)

– “3D” (“milling”), “traversing path (“turning”) (4)”

The display is as follows:

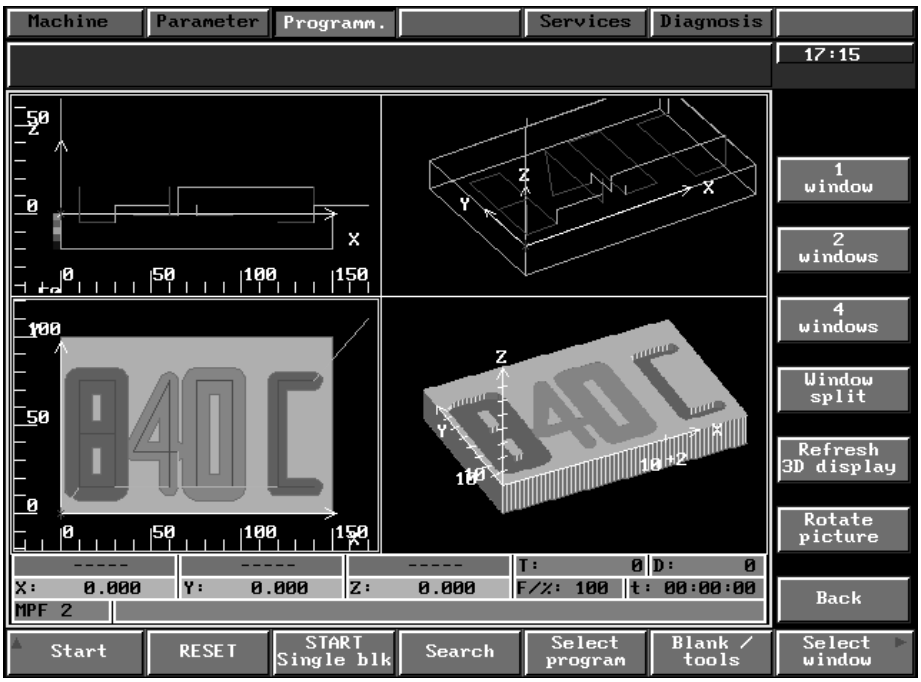


Fig. 3.17 4 windows default

The **Views** parameter allows you to define which displays are active in the **1, 2** or **4 windows settings**. For example, if you use the “3D” selection in the 1 window setting, the 3D display always appears when you press the **1 window** softkey (in the vertical views menu).

Section, milling technology

Parameter: Selection (toggle field)

Possible selections:

- “Off” (default)
- “On”
- “Top/Bottom” is of relevance to “turning” only.

You can also change this parameter for milling using the **Section / Section off** softkey function. If a section is active, it is specified in the section parameter of that display.

If a section is selected for display with the “On” setting and a section is possible in this display, the screen output is updated regardless of the **Sections** softkey function.
The settings made here are overwritten with the **Section/Section off** softkey functions.

Section, turning technology

Parameter: Selection (toggle field)

Possible selections:

- “Off” (outside view)
- “On” (section display of a slice)
- “Top” (half section display)
- “Bottom” (half section display)

For example, the following can be visualized in view “Left (1)”:

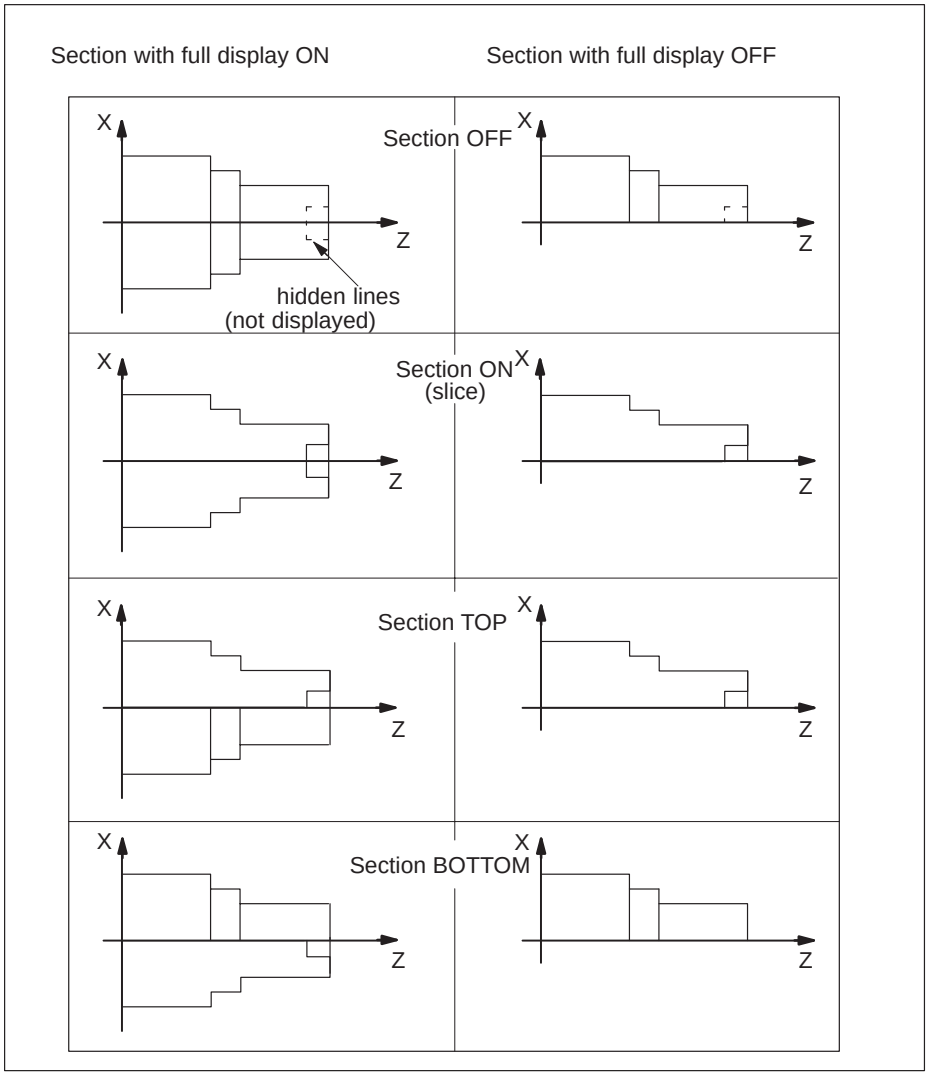


Fig. 3.18 View left

Change of main views by

Parameter: Selection (toggle field)

Possible selections:

- “Change in viewing dir.”
- “Tilting of workpiece” (default)

The setting only applies to the views

- “Left”
- “Right”
- “Rear”.

The “Change in viewing dir.” selection generates the screen output of a view from the viewpoint of the observer.

For the “left” view this means:
The view of the workpiece appears as if the observer is “walking around the workpiece” to the left. The workpiece is not tilted. In this case the aspect of the workpiece does not change.

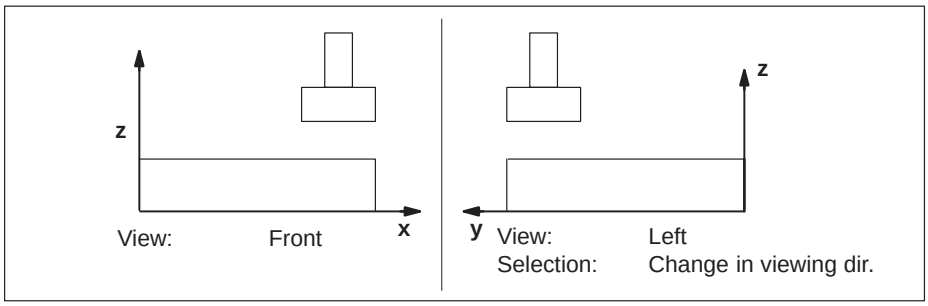


Fig. 3.19 Change in viewing direction

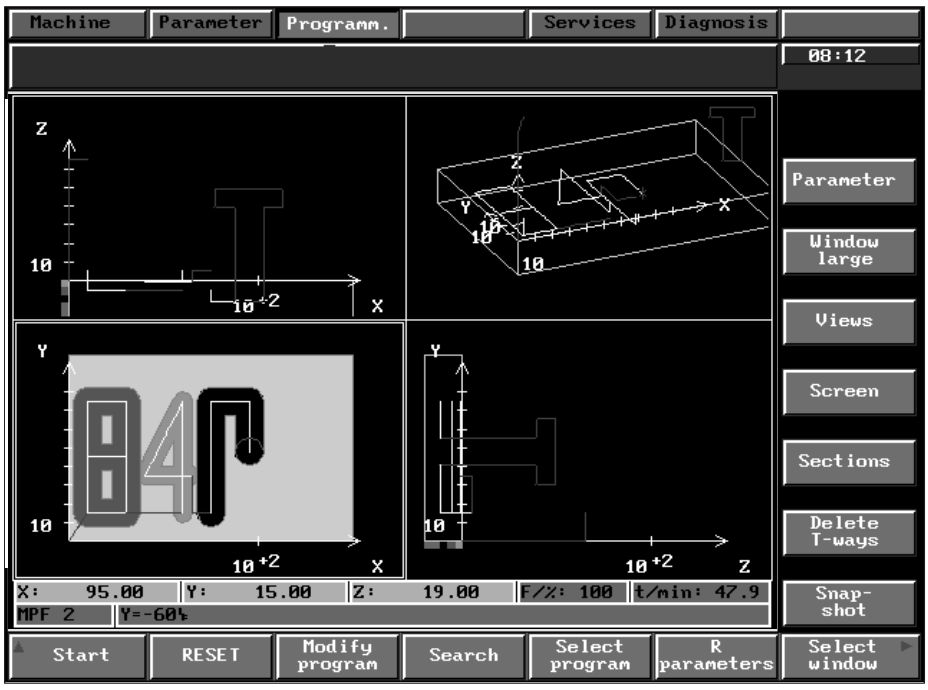


Fig. 3.20 Change in viewing direction in the 3rd window, milling

Selecting “Tilting of workpiece” generates the screen output of a view from the “viewpoint” of the workpiece.

For the “Left” view this means:
The workpiece is tilted to show a view from the left-hand side. The position of the tool in the display changes:

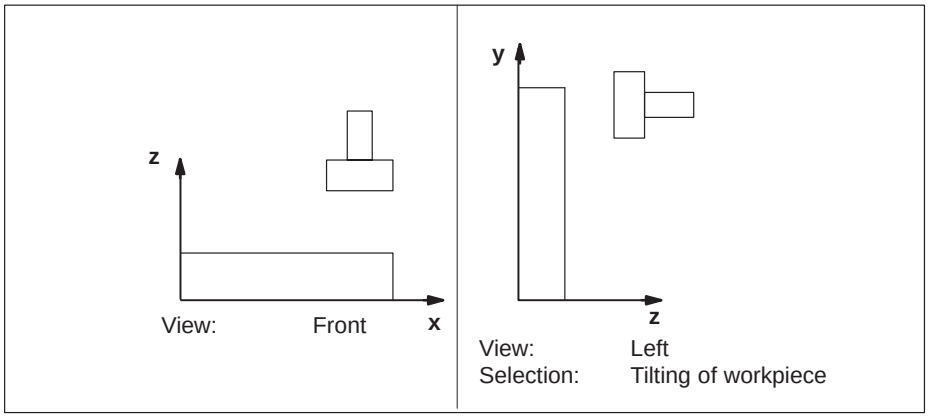


Fig. 3.21 Tilt workpiece

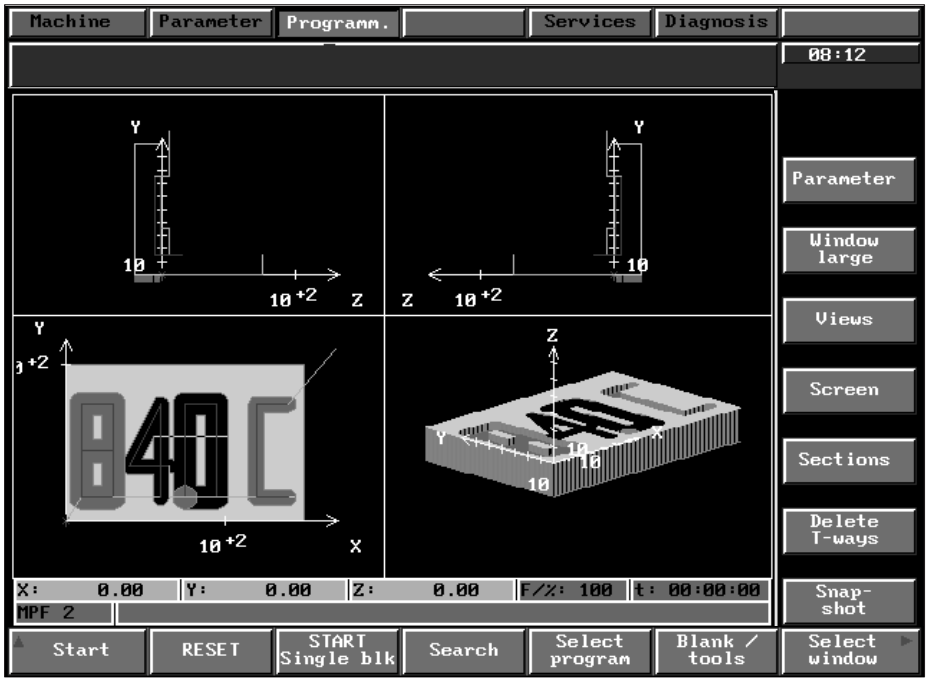


Fig. 3.22 Left-hand tilting in the 4th window, milling

3.2.6 Load list

Load list

Press the **Load list** softkey. A selection of files (e.g. machine data) appears which are included in the simulation if accepted in the load list.

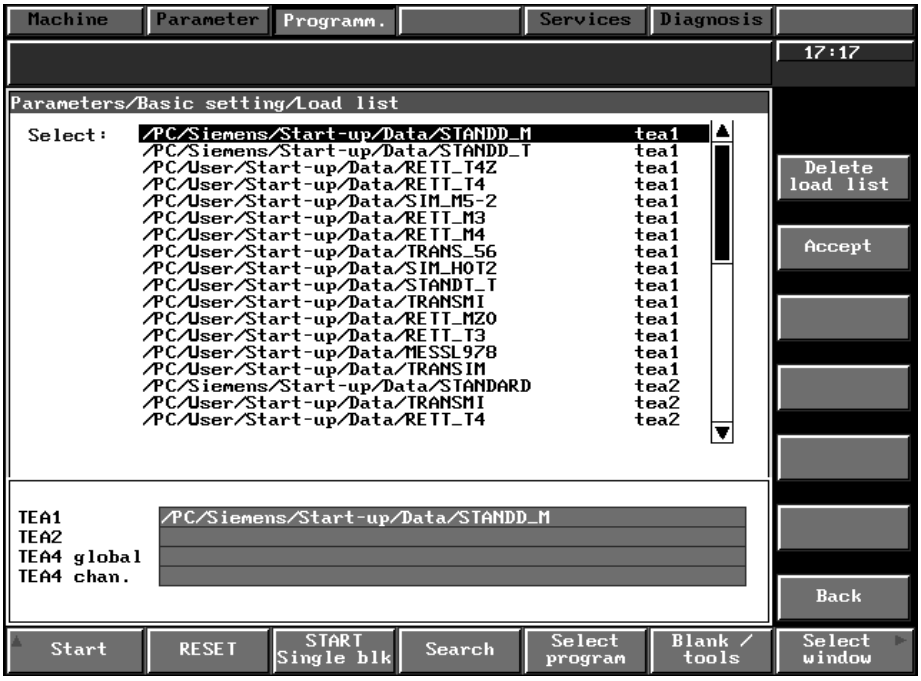


Fig. 3.23 Load list in the simulation area

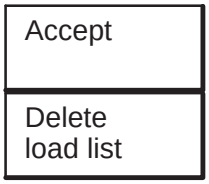
Changes to the load list only take effect when the simulation is rebooted. If the modifications are to be retained after the simulation is terminated, you must save the parameters to hard disk.

The **Load list** display is divided into two areas:

- Selection
- Load list

The upper area of the display provides you with a selection of files on the hard disk which can be copied into the load list.

Use the **Cursor keys** to select a file.



Press the **Accept** softkey to copy the entry selected by the cursor into the load list.

Press the **Delete load list** softkey to delete the entire load list.

You can delete the entire load list or overwrite individual entries.

Notes

TOA, ZOA, RPA, SEA and SEA4 data can be transferred with a job list for the tool in question.

Any files not loaded are assigned the value zero.
The load list consists of the data of Start-up/Data of the user and Siemens area.

The load list is only loaded when the simulation is booted and **not** when a workpiece is selected.

3.3 Change T/M

See Section “Changing the technology”, page 2-3.

3.4 Ignored message texts

The “840C Diagnostics Guide” lists all the message texts that can be displayed.

You can acknowledge such message with

OK

OK or

Abort

Abort

Note

If you set the “Suppress messages” field under the **Parameter** function (see Fig. 3.24), the simulation function will

- no longer output the message when you press **Abort**
- continue to output the message when you press **OK**.

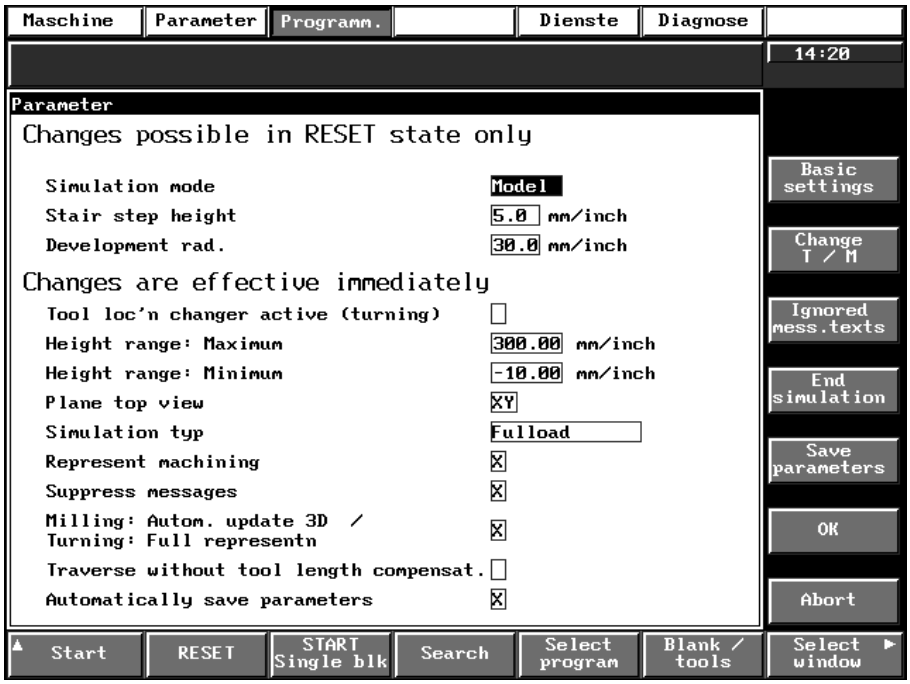


Fig. 3.24 ***Parameter*** *function with* ***Ignored message texts*** *softkey*

If you want to start displaying the ignored messages again, proceed as follows:

Sequence of operations

Parameter

Call the **Ignored message texts** function under the **Parameter** function (see Fig. 3.24)

Ignored message texts

When you activate the **Ignored message texts** function, the following display appears:

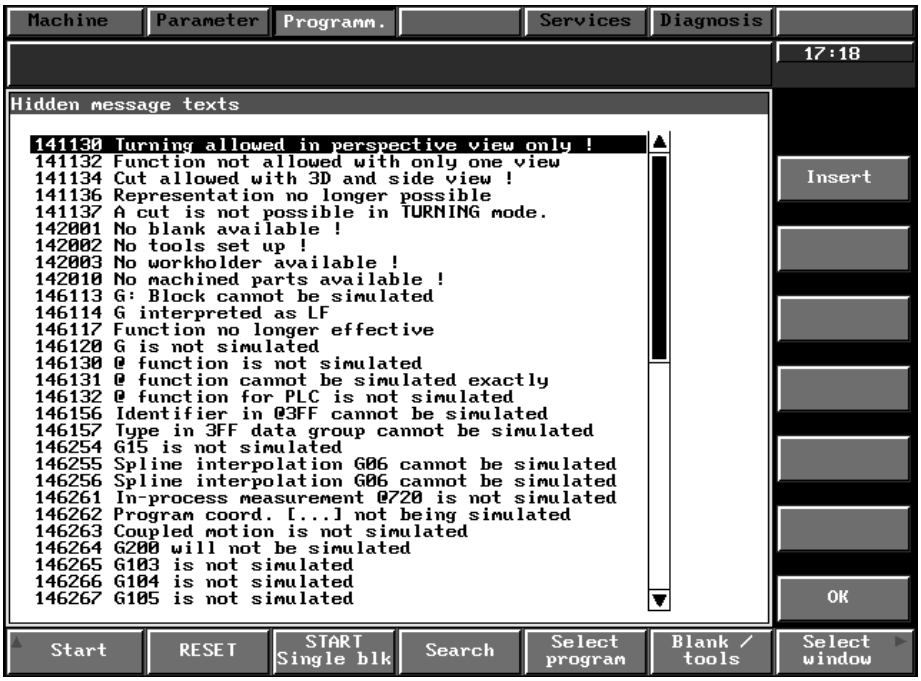


Fig. 3.25 ***Ignored message texts*** *function*

Select the messages that you wish to display again with the **cursor keys**.

Insert

Then press the **Insert** softkey and the ignored text is displayed again.

OK

Once you have selected all the messages you want, close the function with **OK** and the **simulation** function displays the messages that were previously ignored.

Note

If you save the parameters, the simulation function stores the current list of suppressed messages.

Extend the horizontal softkey bar with the **ETC key**

Message log

and you can then read all the displayed or suppressed messages (see Section 2.12 “Message logs”) in the simulation by activating the **Message log** function.

■

4 Setting up a Workpiece

All you need is a part program to simulate a workpiece.

Simulation also evaluates

- A **job list** (from which the simulation evaluates the loading instructions for system data and programs as well as the channel)
- A **shopfloor sheet** (from which the simulation evaluates the blank and finished part information)
- A **tool list** (from which the simulation evaluates the tool and chuck information).

The files are to be found in the workpiece directory.

Fig. 4.1 below shows an example for the workpiece EXEMP_M with the

- part program MPF2
- job list JOB
- shopfloor sheet WERKSTA
- tool list WERKZEUG.

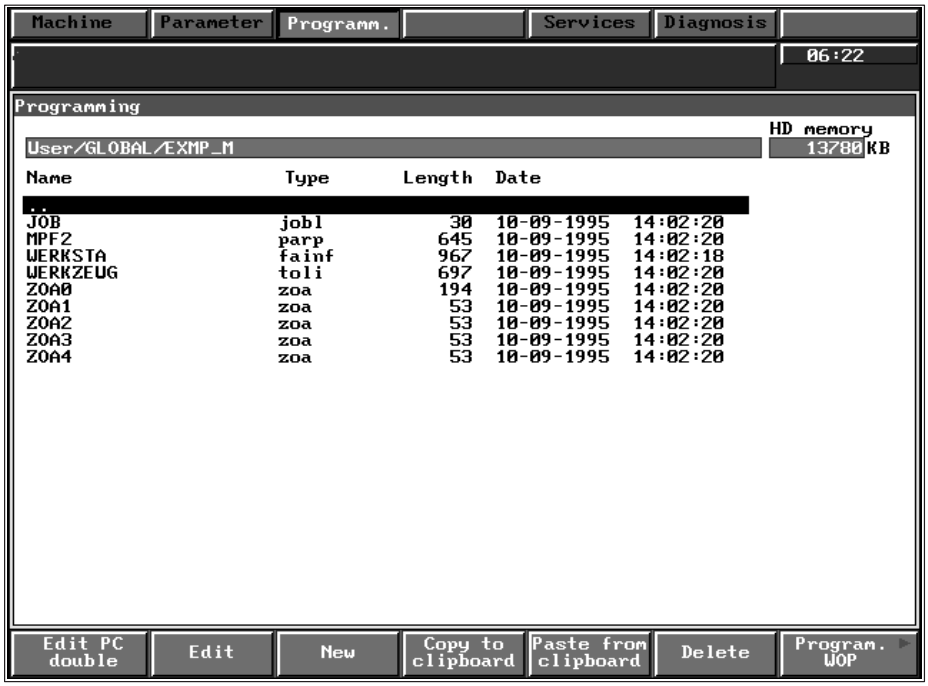


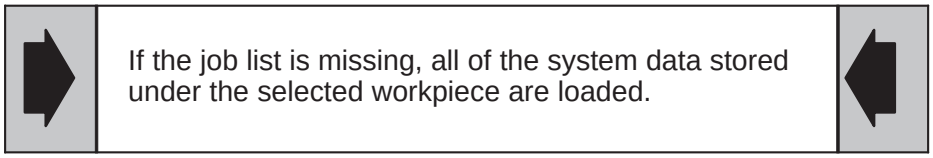
Fig. 4.1 Workpiece directory

The actual machine conditions are taken into account by the ZOA data that are loaded from the NCK.

If you create the workpiece to be simulated with the **programm.WOP** function in the Graphic Programming System, the file is automatically stored in the workpiece directory.

If you create the workpiece to be simulated in the conventional way and the simulation program is to include information from a job list, a shop-floor sheet or a tool list, you must create these files in the conventional manner, too.

You can copy existing files of this type from another workpiece and configure them as required with an editor.



If the job list is missing, all of the system data stored under the selected workpiece are loaded.

The following sections describe the

- Job list
- Shopfloor sheet and
- Tool list

and give the ID and syntax for each.

4.1 Job list

All of the loading instructions are followed. This means that job-specific system data can be loaded. All of the main programs and subroutines required for machining the workpiece can be defined in the JOB LIST. If several main programs exist, the main program with which machining is to begin can be identified.

The channel number in the JOB LIST must be identical to the simulation channel number setting. If the channel numbers are different, the program is not automatically loaded when a workpiece is selected.

Job list exists

The loading instructions for system data are executed (the previous system data are overwritten).

The system determines the program with which machining is to begin as well as the channel number.

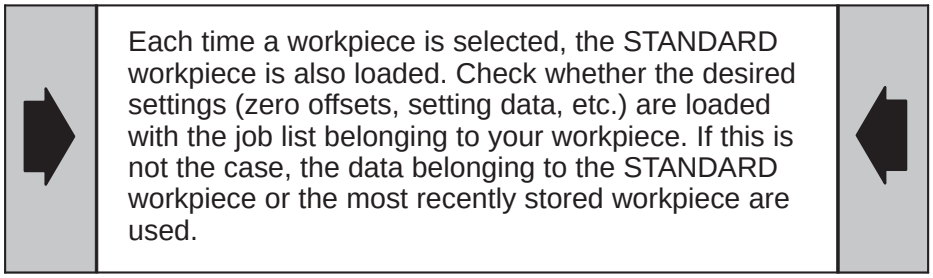
All of the main programs and subroutines specified in the job list are entered in the program selection list.

Job list missing

All of the system data stored under the selected workpiece are loaded.

No program is selected.

All of the main programs and subroutines stored under the selected workpiece are entered in the program selection list.



Each time a workpiece is selected, the STANDARD workpiece is also loaded. Check whether the desired settings (zero offsets, setting data, etc.) are loaded with the job list belonging to your workpiece. If this is not the case, the data belonging to the STANDARD workpiece or the most recently stored workpiece are used.

4.2 Shopfloor sheet (work schedule)

The shopfloor sheet (WERKSTA) contains the data for the

- blank (ID RAW_#)
- finished part (ID OUT_#)
- rechucking position (ID POS_P, see Section 2.21).

The actual machine conditions are taken into account by the ZOA data that are loaded from the NCK.

The simulation evaluates the information on the blank for a

- contour (turning technology) or
- cuboid (milling technology) or a
- cylinder.

Fig. 4.2 below shows a shopfloor sheet for workpiece EXMP_T:

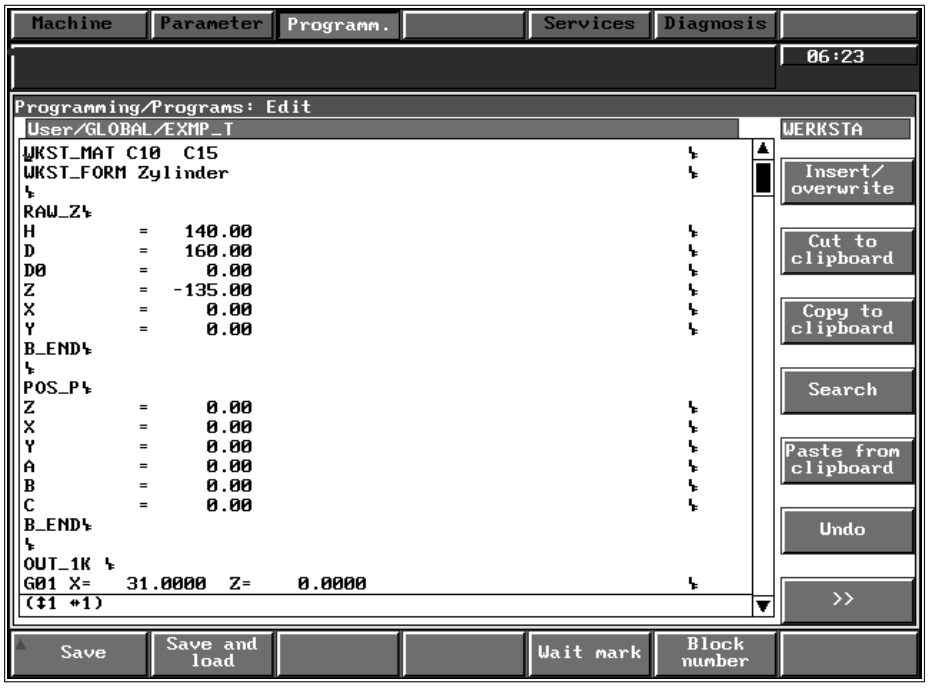


Fig. 4.2 Shopfloor sheet WERKSTA

4.2.1 Defining the blank

The simulation evaluates the unmachined part data from the shopfloor sheet. The shopfloor sheet supports the simulation.

The initial point parameter in the shopfloor sheet defines the clamping position of the workpiece. The clamping position and the zero offsets must be coordinated so that the desired final machining point is reached.

The blank contour lies in the ZX plane.

The ID for a blank definition is

- RAW_K For a blank contour in turning
- RAW_Q For a blank contour that is a cuboid for technology milling and
- RAW_Z For a cylindrical blank contour.

Blank contour technology turning

The syntax that follows the ID is described in Section 4.2.3.

RAW_K	For a blank contour in turning

B_END	End ID

Blank contour that corresponds to a cuboid for milling

The syntax that follows the ID has the following meaning:

RAW_Q	For a blank contour that is a cuboid for technology milling
L	Length of the cuboid Data format: Floating point
W	Width of the cuboid Data format: Floating point
H	Height of the cuboid Data format: Floating point
Position-related information	
X	Offset of the cuboid in the X direction of the graphical workpiece Data format: Floating point
Y	Offset of the cuboid in the Y direction of the graphical workpiece Data format: Floating point
Z	Offset of the cuboid in the Z direction of the graphical workpiece Data format: Floating point
C	Rotation of the cuboid around the Z axis in the workpiece zero Data format: Floating point
B_END	End identifier

Cylindrical blank contour

The syntax that follows the ID has the following meaning:

RAW_Z	For a cylindrical blank contour
H	Height of the cylinder Data format: Floating point
D	Outer diameter of the cylinder
D1	Inner diameter of the cylinder

Position-related information

X	Offset of the cylinder in the X direction of the graphical workpiece Data format: Floating point
Y	Offset of the cylinder in the Y direction of the graphical workpiece Data format: Floating point
Z	Offset of the cylinder in the Z direction of the graphical workpiece Data format: Floating point
B_END	End identifier

Position description of workpiece zero ZO2

The syntax for the position description of the workpiece zero with reference to machine zero ZO2 for rechucking (see Section 2.21) is:

POS_P	Position description block of workpiece zero with reference to machine zero (ZO2, see Section “Rechucking”)
X	Coordinates of the zero offsets
Y	
Z	
B_END	End ID

Cuboid

Fig. 4.3 below describes the parameters for a cuboid:

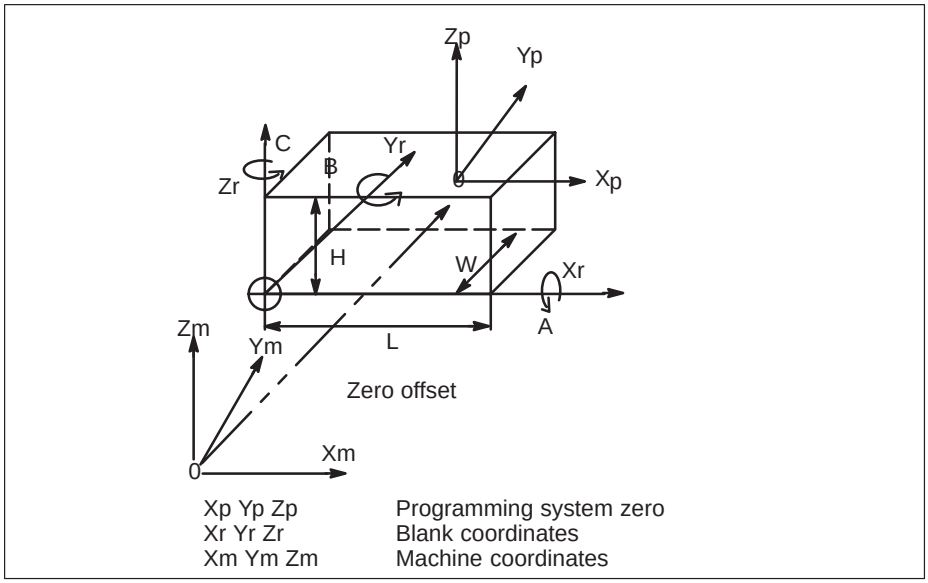


Fig. 4.3 Parameters for cuboid

Programming example

RAW_Q	ID
L =	Length of the cuboid
W =	Width of the cuboid
H =	Height of the cuboid
X =	Offset of the cuboid in the X direction of the graphical workpiece
Y =	Offset of the cuboid in the Y direction of the graphical workpiece
Z =	Offset of the cuboid in the Z direction of the graphical workpiece
C =	Rotation of the cuboid around the Z axis
B_END	End ID of blank description

Fig. 4.4 below explains parameter “C”:

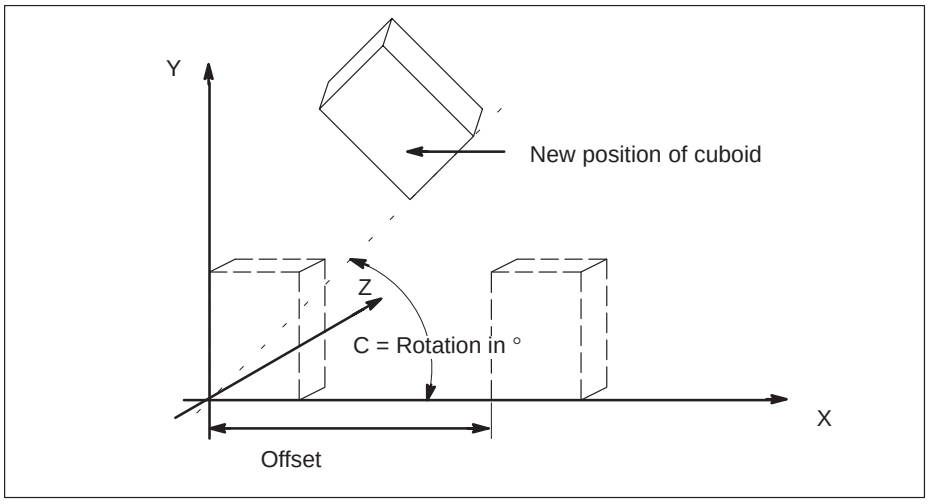


Fig. 4.4 Parameter C – Rotation of cuboid around Z axis

Cylinder

Fig. 4.5 below explains the parameters for a cylinder:

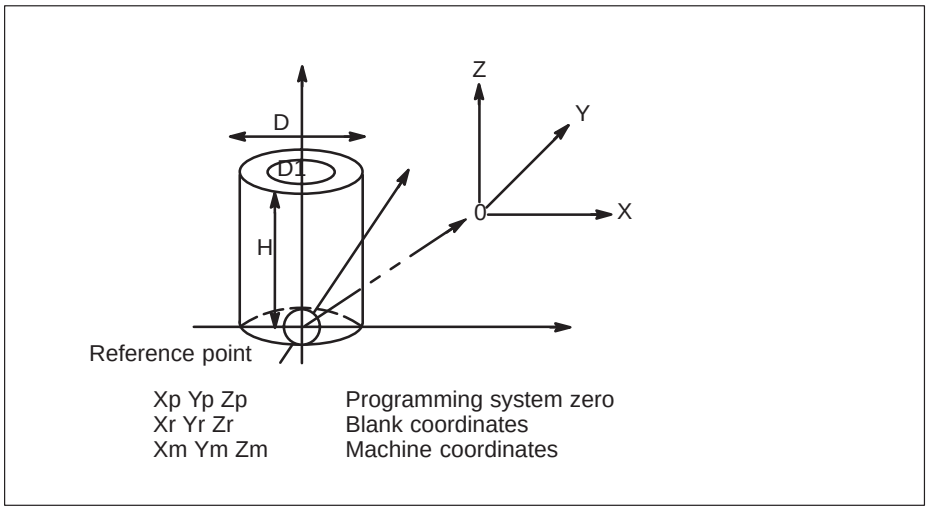


Fig. 4.5 Parameters for cylinder

Programming example

RAW_Z	ID
H =	Height of the cylinder
D =	Outer diameter of the cylinder
D1 =	Inner diameter of the cylinder
X =	Offset of the cylinder in the X direction of the graphical workpiece
Y =	Offset of the cylinder in the Y direction of the graphical workpiece
Z =	Offset of the cylinder in the Z direction of the graphical workpiece
B_END	End ID

Please see Section 4.2.3 “Defining the contour” for the contour definitions.

Notes

If a shopfloor sheet **exists,** the blank information is evaluated by the simulation.

If there is **no** shopfloor sheet, the simulation does not show any machining operations. Only the traversing paths of the tools are visualized.

4.2.2 Finished part definition

The finished part information is introduced by the ID OUT_# K (contour description).

For a description of the parameters please refer to the following Section “Defining the contour”.

4.2.3 Defining the contour

A finished part can be defined so that a contour that is known before simulation can be displayed on the screen.

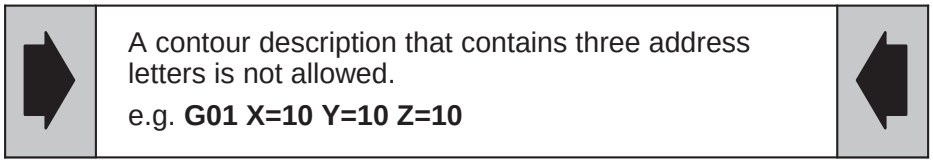
This defined contour can then be compared directly with the contour definition in the simulation process.

The finished part contour is defined in the XY, YZ, ZX or ZC plane and must be addressed with the correct address letters.

The blank/chuck contour is defined in the ZX plane.

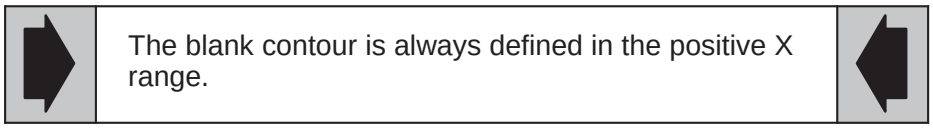
The ID for the contour definition is

- RAW_K Blank contour turning technology
- OUT_#[1] K Finished part contour turning and milling
- CHUCK_#[1] K Chuck contour



A contour description that contains three address letters is not allowed.

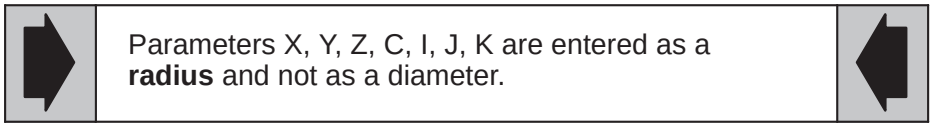
e.g. **G01 X=10 Y=10 Z=10**



The blank contour is always defined in the positive X range.

The contour is applied using G01 to G03 blocks in a fixed format. All positional parameters must be written:

Position parameter	Description
X	Absolute coordinates X axis Data format: Floating point
Y	Absolute coordinates Y axis Data format: Floating point
Z	Absolute coordinates Z axis Data format: Floating point
I	Increm. coordinates center point X axis Data format: Floating point
J	Increm. coordinates center point Y axis Data format: Floating point
K	Increm. coordinates center point Z axis Data format: Floating point
C	Absolute coordinates C axis Data format: Floating point



Parameters X, Y, Z, C, I, J, K are entered as a **radius** and not as a diameter.

1) # is replaced by a serial number starting at 1

Position parameter	Description	Example
G01 [X=] [Z=] [Y=] [C=]	Linear movement to end point	XY plane: G01 X= Y= YZ plane: G01 Y= Z= ZX plane: G01 X= Z= ZC plane: G01 C= Z=
G02 [X=] [Z=] [Y=] [C=] [I=] [J=] [K=]	Clockwise circular movement to end point with circular interpolation parameters I, K	XY plane: G02 X= Y= I= J= YZ plane: G02 Y= Z= J= K= ZX plane: G02 X= Z= I= K= ZC plane: G02 C= Z= I= K=
G03 [X=] [Z=] [Y=] [C=] [I=] [J=] [K=]	Counterclockwise movement to end point with circular interpolation parameters I, K	XY plane: G03 X= Y= I= J= YZ plane: G03 Y= Z= J= K= ZX plane: G03 X= Z= I= K= ZC plane: G03 C= Z= I= K=
TRANS X= Z= Y=	Transformation block of contour with reference to workpiece zero	

Notes

The 1st block of a contour definition must be a G01 block.

The following applies to a blank contour definition:

- Maximum 40 blocks (contour elements)
  Every circular element reduces the maximum number by 2
- Values must be entered in the positive X range and
- it is always automatically closed and rotated around the Z axis.

The following applies to a finished part contour definition:

- Maximum 99 blocks (contour elements)
  Every circular element reduces the maximum number by 2
- It is not closed and
- it is displayed in the appropriate view.

Example

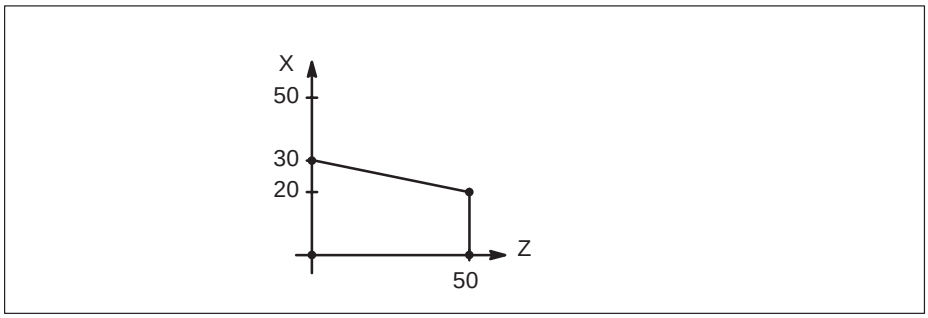


Fig. 4.6 Example

```
RAW_K
G01    X = 0   Z = 0
G01    X = 0   Z = 50
G01    X = 20  Z = 50
G01    X = 30  Z = 0
B_END
```

The contour of the blank is automatically closed, i.e., as the example shows, it is not necessary to program the last block

```
G01    X = 0   Z = 0.
```

4.3 Tool list

The tool list (WERKZEUG) contains the data for the

- tools,
- chucks and
- chucking scenarios

to be simulated.

The simulation evaluates the tool data and chuck data from the tool list.

The graphical tool list supports simulation and does not represent a complete tool description.

A tool can be defined quickly and simply with the aid of the tool list.

If no part program is loaded with the workpiece, no tools are set up.

A full check of the tool dimensions is only possible by using the TOA files (setting: JOB TOA).

The ID for a tool list is

- TOOL_#[1)] Tools,
- CHUCK_#[1)] Type (cuboid, cylinder or contour) of chuck and
- CHUCK_ID_#[1)] Chucking scenarios.

Fig. 4.7 below shows an example of a tool list for workpiece EXMP_T:

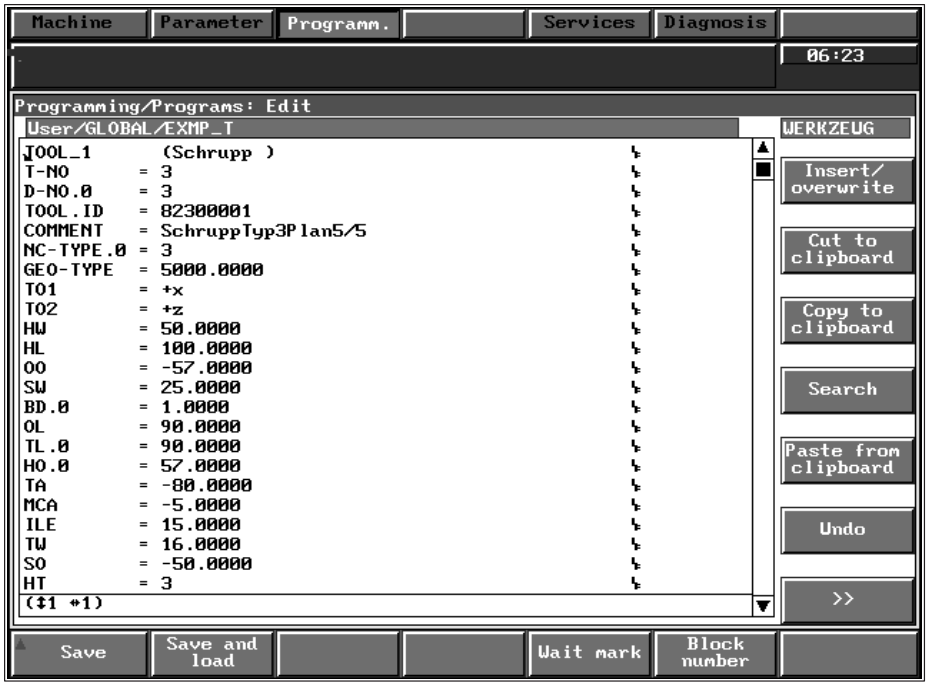


Fig. 4.7 WERKZEUG tool list

1) # is replaced by a serial number starting at 1

4.3.1 Tool data

The ID for the tools is

TOOL_#.

"#" is the serial number of the relevant block type. Numbering always starts at "1".

The following data are taken into account by the simulation:

Programming example

TOOL_1		
T–EXT	= 8	Address extension
NC–TYPE.0	= 10	Tool type
T–NO	= 45	Tool number (T number)
D–NO.0	= 5	Tool offset memory number
D–NO.1	= 6	Tool offset memory number
D–NO.2	= 3	Tool offset memory number
D–NO.3	= 5	Tool offset memory number
TO1	=+Z	Orientation of the infeed axis of the tool (dependent on the selected machining level)
HW	= 20	Diameter of the holder
HL	= 10	Length of the holder
BD	= 16	Diameter of the tool edge (P4)
OL	= 80	Total length of the tool (P2)
OO	= 10	Cross dimension
B_END		End identifier of the tool description

Notes

Additional data are required to represent the tools graphically. These are described separately for each technology in Sections 4.3.2 (milling) and 4.3.3 (turning).

Parameters which are not required do not need to be entered in the tool list.

Tool list exists

The tool information is evaluated by the simulation. However this does not occur if **no part program is specified in the job list.** In this case **no tool is set up.**

With the **TOA to be used: graphics data** setting the compensation block of the tool is determined from the graphics data and stored in the 4 defined offset numbers (existing information is overwritten). All of the tools entered in the tool list are loaded. If the same offset memory number has been specified for several tools, the last tool for which the same D number was specified overwrites the previous offset values.

With the **TOA to be used: JOB TOA** setting the system uses the loaded tool offset values.

Tool list missing

Since the tool information is not available in the simulation, the tools are represented by a polymarker. It is not possible to display the machining processes. Only the traversing paths of the tools are visible.

With the **TOA to be used: graphics data** setting, no offset values are used.

With the **TOA to be used: JOB TOA** setting the values entered in the offset memory are used.

T–EXT Address extension of the tool number (T number). For example, if the DIN program includes the call T1 = 25:
The address extension is 1.
Data format: Integer max. 2 digits.

NC TYPE.O NC type of the tool for the TOA data block.
Turning tools = 1 to 9
Drill = 10
Mill = 20
Angle mill = 30
Data format: Integer max. 2 digits.

T–NO Tool number (T number)
Data format: Integer max. 8 digits.

D–NO.0 Tool offset memory number. If the Siemens tool manager is active or no offset numbers are entered which are not equal to 0, the value is set to 0.
Data format: Integer max. 4 digits.

D–N0.1, D–N0.2, ... See D–NO.0 parameters for tool edge (exist for each of the 4 possible tool edges).

TO1, milling Orientation of the infeed axis of the tool dependent on the selected machining level.
Permissible orientations are ±x, ±y, ±z
Standard value –Z
Data format: Max. 2 characters.

Examples of tool orientation (TO1):

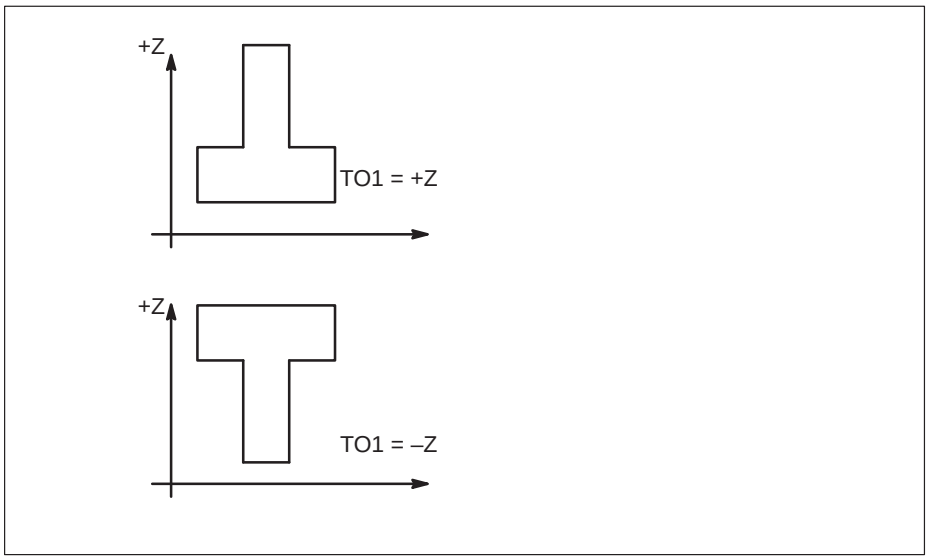


Fig. 4.8 Tool orientation

TO1, turning

With turning tools, the tool point direction is displayed with reference to value TO1.

0	+X
–90	+Z
180	–X
90	–Z

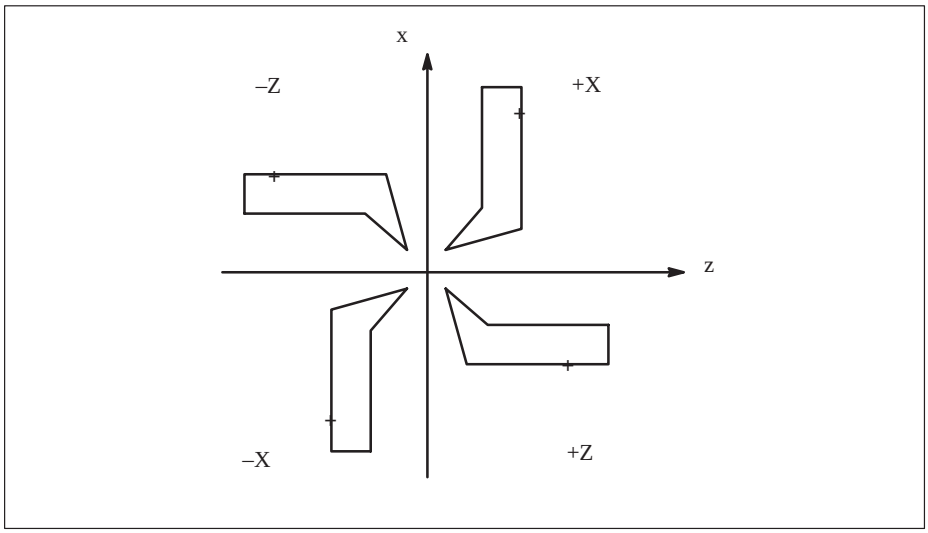


Fig. 4.9 Tool point direction

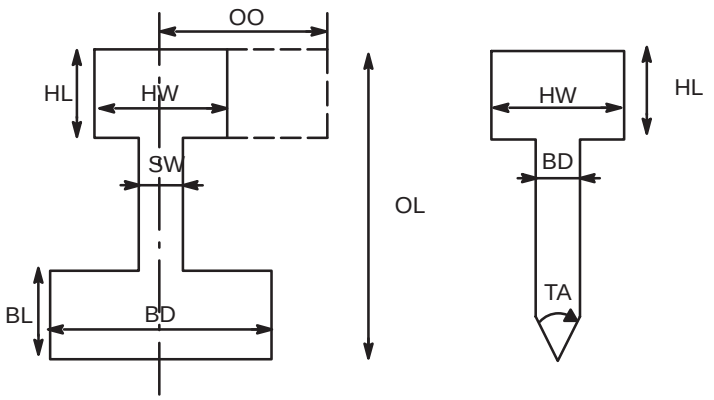
TO2

The orientation of the holder dependent on the selected machining level.
Permissible orientations are ±x, ±y, ±z
Standard value –Z
Data format: Max. 2 characters.

The tool list can also be generated using the operator guidance function and help pictures under **Blank tools**.

The TOOL file may not be larger than 64 Kbytes. If more tools are required, then 2 tool files of less than 64 Kbytes must be created and stored in Standard Workpiece and Actual Workpiece.

4.3.2 Tool definitions for milling



HW Diameter of the holder
Data format: Floating point

HL Length of the holder
Data format: Floating point

SW Diameter of the shaft
Data format: Floating point

BD (P4) Diameter of the cutting edge
Data format: Floating point

BL Length of the cutting edge
Data format: Floating point

OL (P2) Total length of the tool
Data format: Floating point

OO Cross dimension (for angle head cutter)
Data format: Floating point

TA Tip angle of the cutting edge
Data format: Floating point

Fig. 4.10 Tool definition for milling

4.3.3 Tool definitions for turning

You can program a turning tool geometry in two ways:

- minimum input of 8 parameters or
- full parameter input.

Minimum input

If the 8 parameters below are entered, a simplified tool is generated.

The eight parameters:

Programming example:

TOOL_1

T–NO	= 1	Tool number
D–NO.0	= 10	D offset memory
MCA	= 10.0	Main cutting edge angle
TA	= 80.0	Secondary cutting edge angle
NC–TYPE.0	= 4	Tool type
OL	= 100.0	Longitudinal dimensions in neutral length
OO	= –50.0	Cross dimension in neutral length
TO1	= +X	Tool point direction
B_END		End ID of the tool description

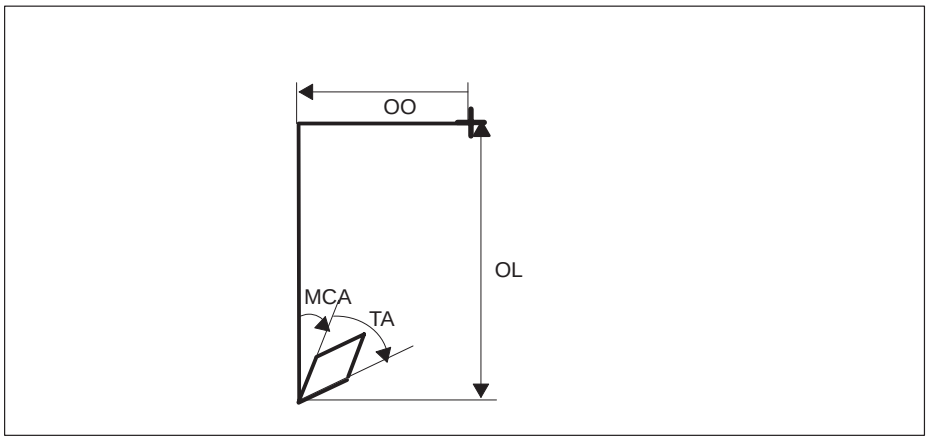


Fig. 4.11 Minimum definition

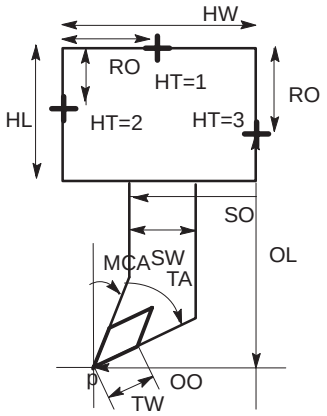
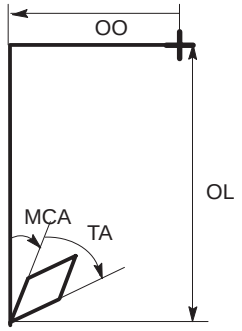
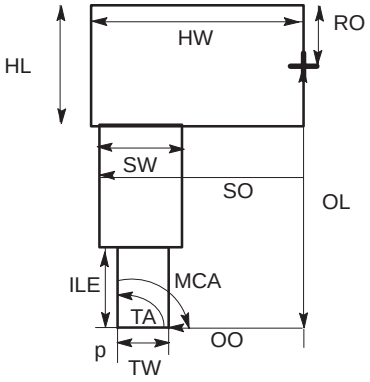
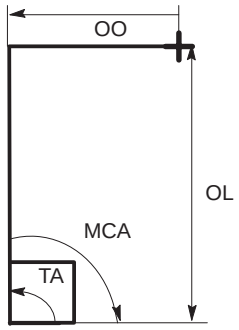
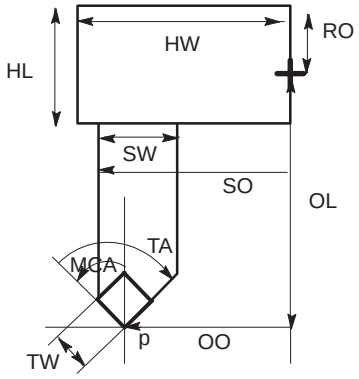
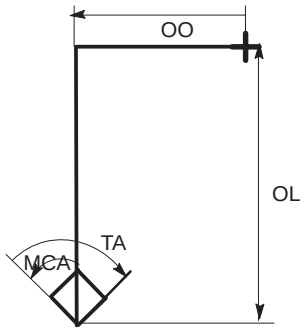
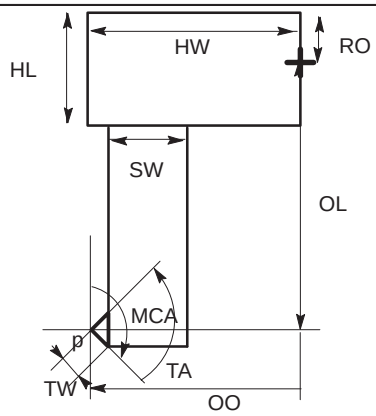
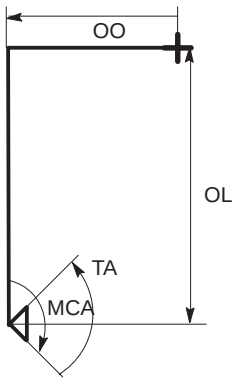
If more than these eight parameters are entered, the tool is output graphically with tool nose, holder and shank.

Full parameter input

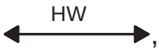
The tool is simulated with

- tool nose
- holder and
- shank.

Tool simulation for turning

	Detailed parameters	Minimum parameters
Roughing/ finishing		
Recessing		
Threading		
Threading		

Notes

- All dimensions with two dimension arrows, e.g. 

, are dimensions that have been generated, i.e. no sign has been included.
- All dimensions with one dimension arrow, e.g. 

, have a sign.

MCA Main cutting edge angle
(ccw is plus, cw is minus)
Data format: Floating point

TA Tip angle (ccw is plus, cw is minus)
Data format: Floating point

ILE Insert length for cutting edge of recessing tool
Data format: Floating point

TW Tool width for recessing tools,
cutting edge length for all other turning tools
Data format: Floating point

SO The distance between the holder point (HT) to the left shank edge in the neutral position (this value is **not** evaluated for internal thread cutting)
Data format: Floating point

SW Shank width
Data format: Floating point

HT Position of the reference point on the holder
Data format: 4 integers
Value range:
1 top
2 left
3 right

RO Distance between the reference point on the holder to the relevant edge. Where HT=2 or HT=3 to the upper edge, where HT=1 to the left edge.

OL Length dimension in the neutral position
Data format: Floating point

OO Cross dimension in the neutral position
OO is the distance between the holder point (HT) and the tool tip
Data format: Floating point

The tools are described geometrically in the neutral position. The insert position is defined by the orientation (±x, ±z).

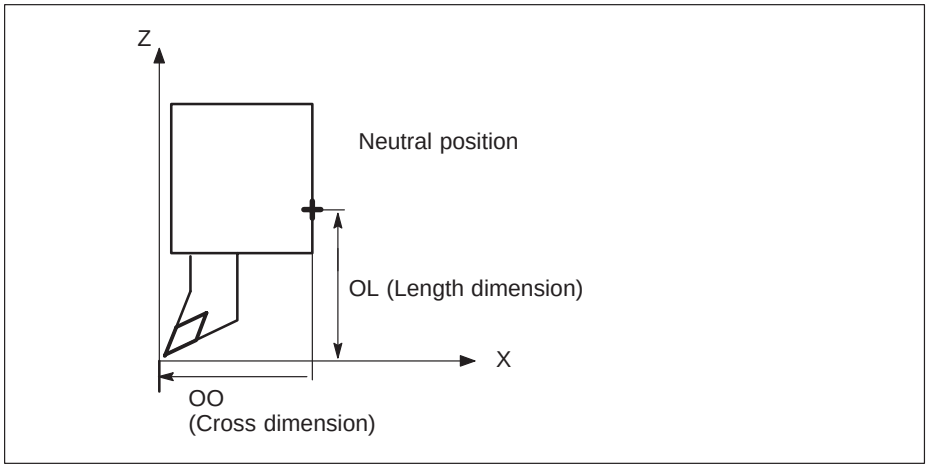


Fig. 4.12 Neutral position

The thickness of holder, shank and cutting edge are set internally by the simulation program and cannot be altered.

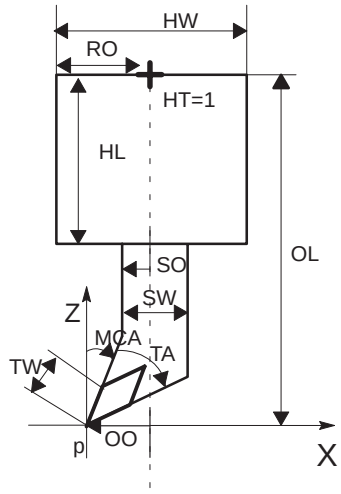
The cutting edge of the recessing tool is always represented by the simulation program as rectangular.

The simulation program uses ideal tools, i.e. without cutting edge radii.

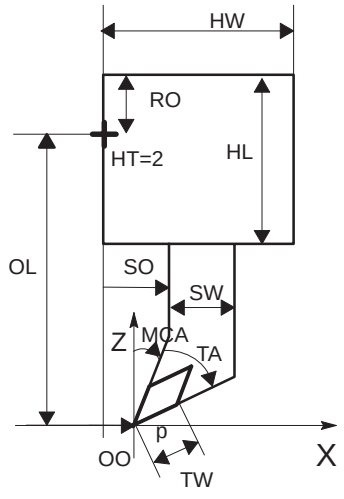
The position of the cutting edge is determined from the values entered for the main cutting edge angle (MCA) and the tip angle (TA).

Depending on the position of the reference point of the toolholder (HT) the following relationships result:

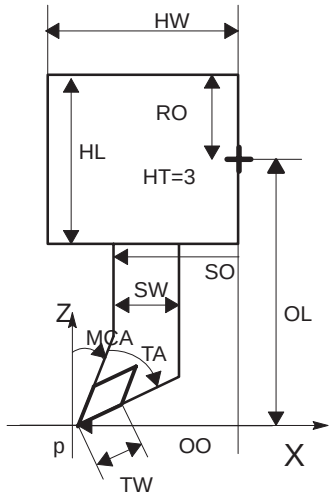
Roughing/finishing
Tool nose left
HT= 1



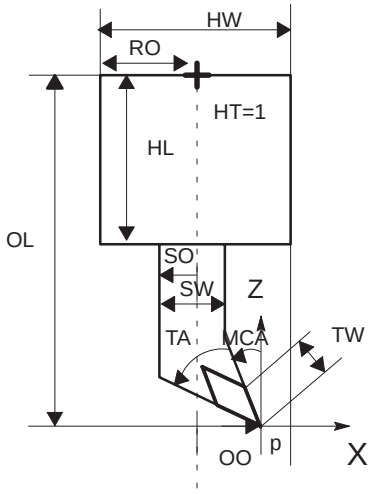
Roughing/finishing
Tool nose left
HT=2



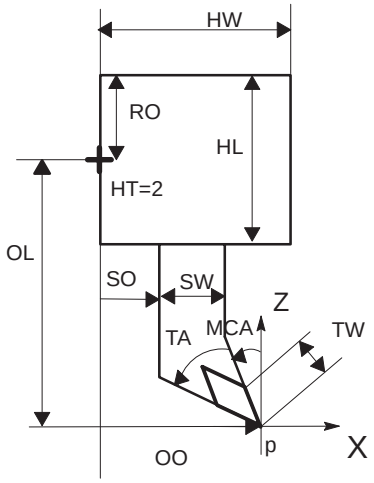
Roughing/finishing
Tool nose left
HT=3



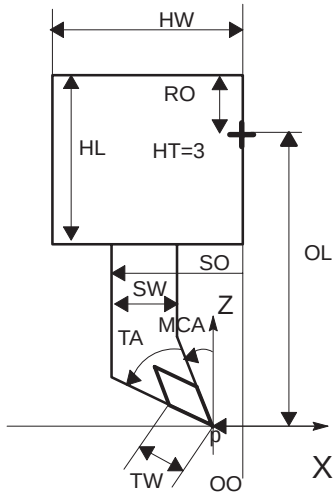
Roughing/finishing
Tool nose right
HT= 1



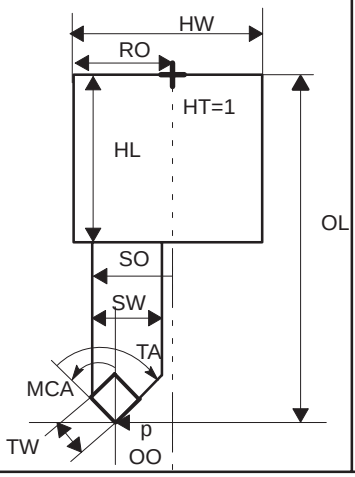
Roughing/finishing
Tool nose right
HT=2



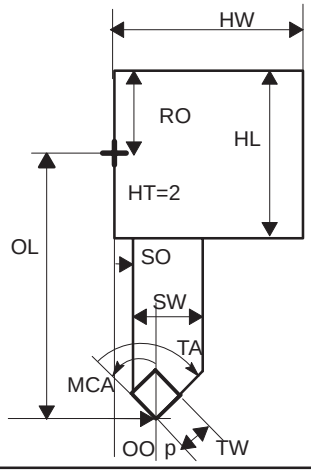
Roughing/finishing
Tool nose right
HT=3



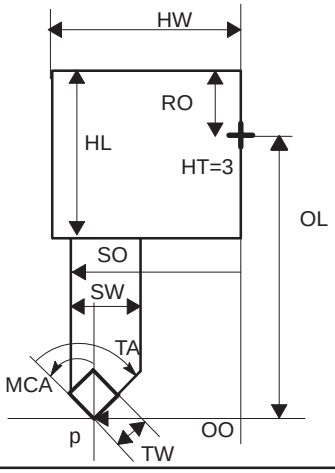
Outside thread
HT=1

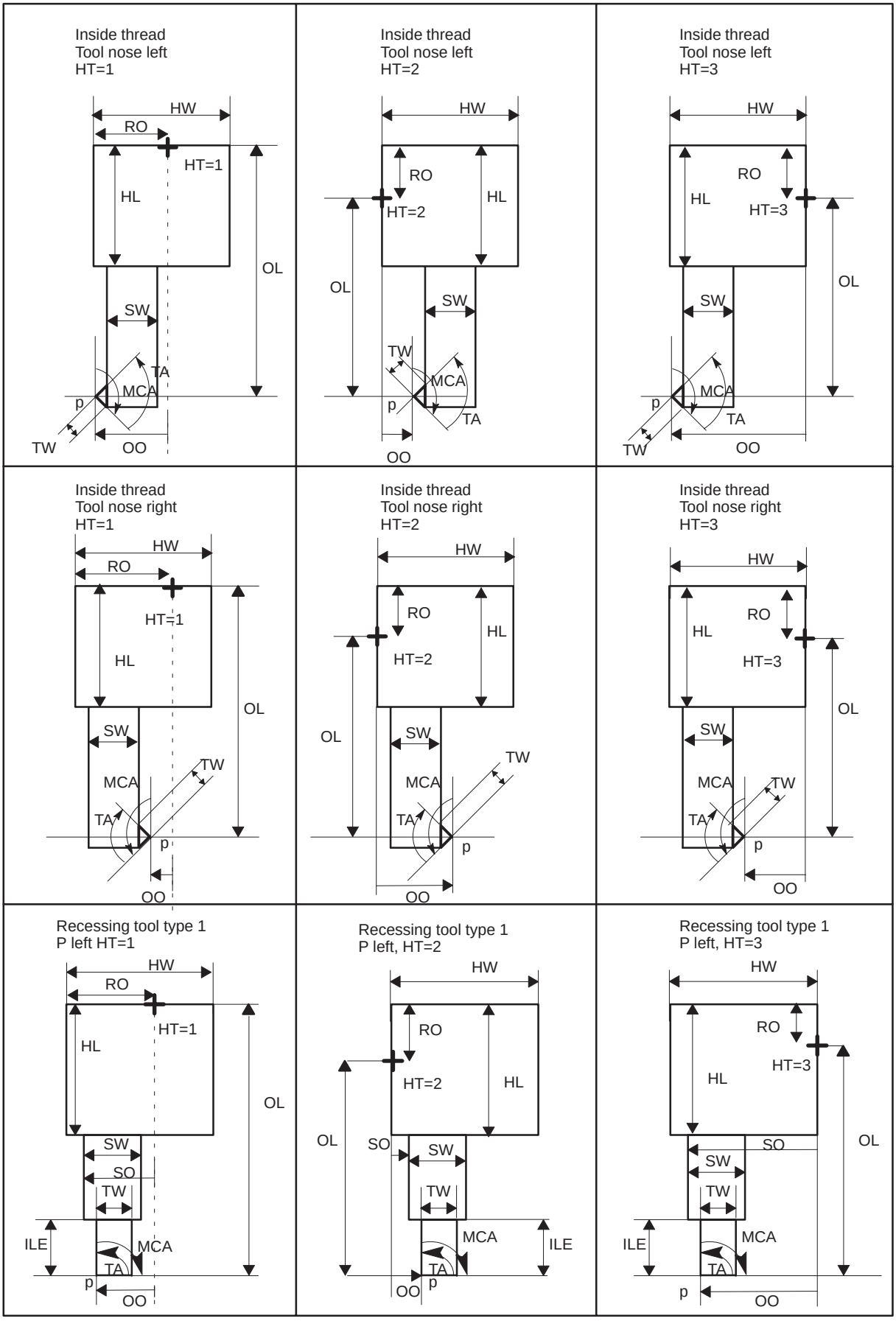


Outside thread
HT=2

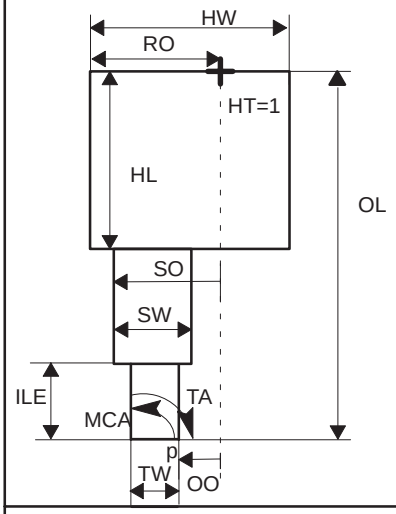


Outside thread
HT=3

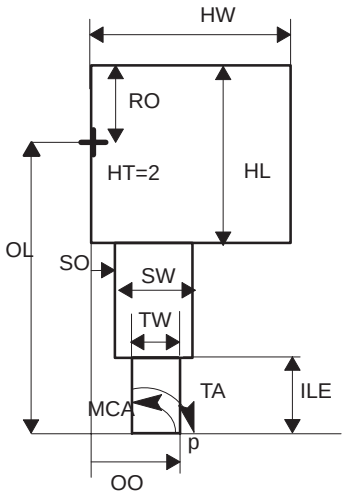




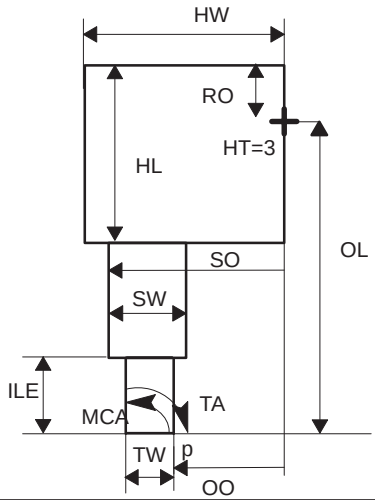
Recessing tool type 2
P right, HT=1



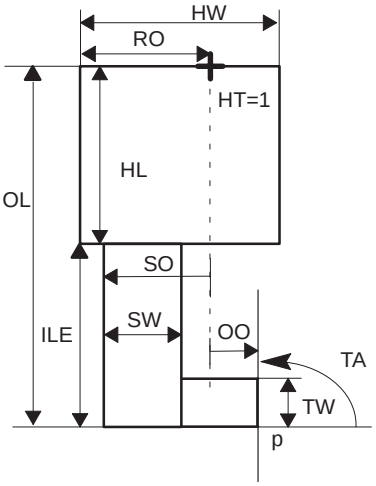
Recessing tool type 2
P right, HT=2



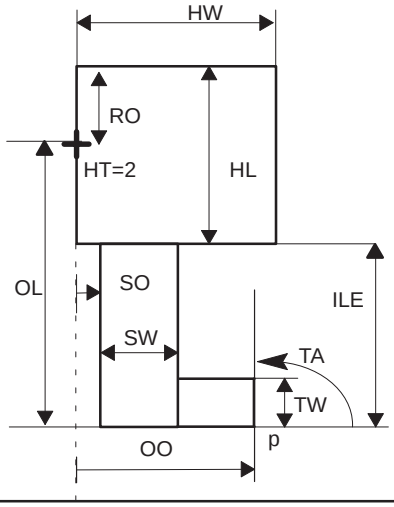
Recessing tool type 2
P right, HT=3



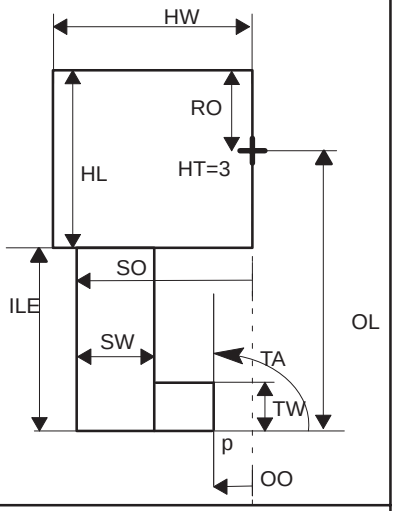
Recessing tool type 3 HT=1
MCA=0, TA=90



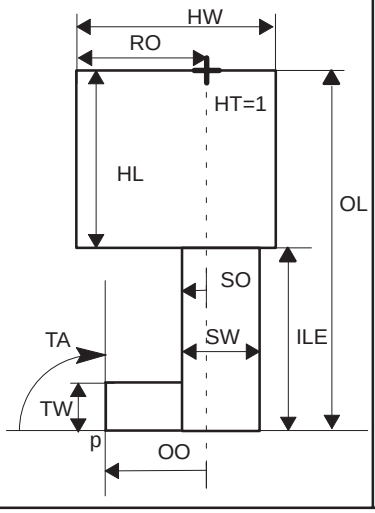
Recessing tool type 3 HT=2
MCA=0, TA=90



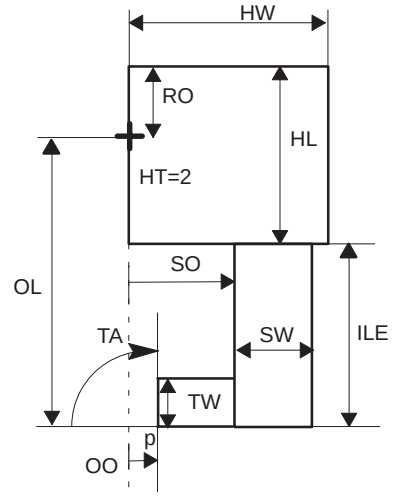
Recessing tool type 3 HT=3
MCA=0, TA=90



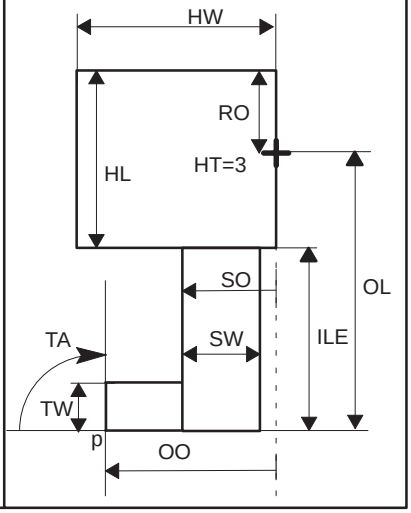
Recessing tool type 4 HT=1
MCA=0, TA= –90



Recessing tool type 4 HT=2
MCA=0, TA= –90



Recessing tool type 4 HT=3
MCA=0, TA= –90



Recessing tool

The recessing tools are identified by the tool edge width. They are distinguished by the values entered for the main tool edge and tool tip angle according to the following pattern:

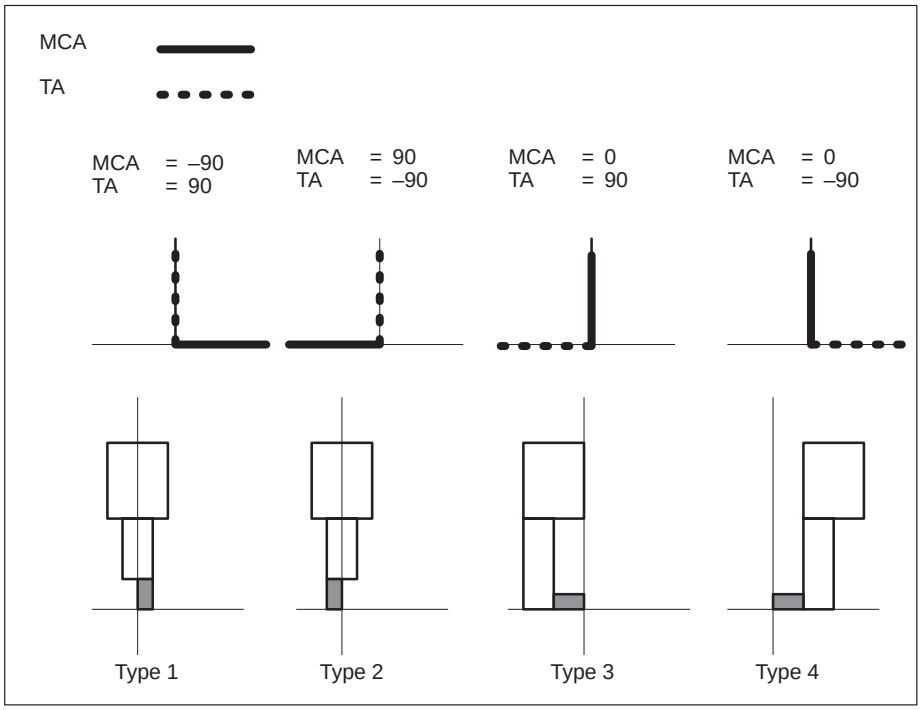


Fig. 4.13 Recessing tool types

The recessing tool path is always along the tool edge center.

Please note the following when defining the recessing tool:
Angle MCA (main tool edge) = 0, ±90°
Angle TA (tool tip) = ±90°
The recessing tool has a maximum of two edges.

The following applies:

Either
the value of the 1st tool edge is entered in D–NO.0 and,
the value of the 2nd tool edge is entered in D–NO.1 or
the value of the 1st tool edge is entered in D–NO.2 and
the value of the 2nd tool edge is entered in D–NO.3.
(D–NO.# is the tool offset memory number).

4.3.4 Effect of tool offsets and graphics data

Tool information can be included in the simulation by means of the workpiece-specific data (%TOA) and/or the tool list.
In the **Parameter/Basic settings/System** display the settings

- Graphics data and
- JOB TOA

can be selected in the “TOA to be used” toggle field.

Graphics data

The geometrical data and the graphics information are taken exclusively from the tool list.

If a workpiece-specific file (%TOA) also exists, it is ignored.

If no tool list exists, no tool information is output even if a workpiece-specific file (%TOA) exists.

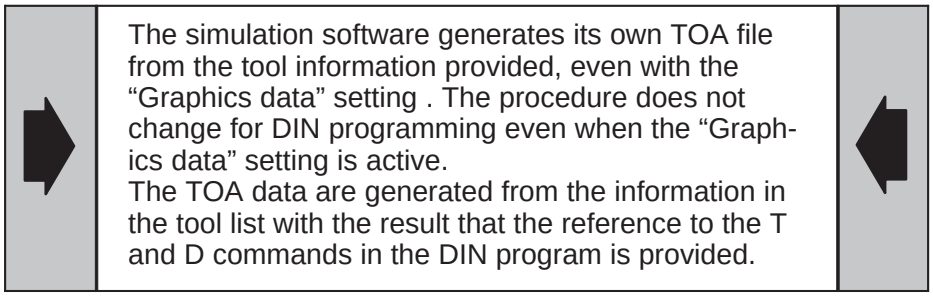
JOB TOA

Offset values are taken from the workpiece-specific file %TOA.

If a tool list file also exists only the graphical information for the screen output is taken from the tool list.

If only the workpiece-specific file (%TOA) exists and the tool list is missing, there is no graphical display of the tool. The offset values of the %TOA file affect the asterisk (*).

If only the tool list and no workpiece-specific file (%TOA) exists, i.e. there is neither a tool list nor a % TOA file, the TOA data are taken from the current TOA memory. These TOA data may not belong to the current workpiece, but may be loaded from the job list of another workpiece. This may result in the simulation of undesired tool offsets.



The simulation software generates its own TOA file from the tool information provided, even with the “Graphics data” setting . The procedure does not change for DIN programming even when the “Graphics data” setting is active.
The TOA data are generated from the information in the tool list with the result that the reference to the T and D commands in the DIN program is provided.

Note

The simulation does not provide for tool wear.

4.3.5 Describing the workholder and chucking scenarios

Chuck

The ID for a chuck is

- CHUCK_#[1)] type.

"Type" is replaced by "K" for contour, "Z" for cylinder or "Q" for cuboid.

The simulation program evaluates maximum 10 chuck geometries.

A maximum number of 40 linear geometrical elements can be entered when defining a chuck contour (a circular element reduces the number by 2).

Chucking scenarios

The ID for a chuck scenario is

- CHUCK_ID_#[1)].

The simulation program evaluates max. 99 chucking scenarios.

Programming example

```
CHUCK_1 K                          Chuck
G01 X = 140.00 Z = 60.0
G01 X = 140.00 Z = 25.0
G02 X = 160.00 Z = 5.00 I = 20.0
G01 X = 160.0 Z = 0.0
G01 X = 100.0 Z = 0.0
G01 X = 100.0 Z = 60.0
TRANS X = 0.0 Y = 0.0 Z = –10.0
B_END

CHUCK_ID_1                         Chucking scenario description
1                                  Name (number) of chuck
B_END
```

Notes

When describing the chucking scenarios, enter the numbers of the chuck in question.

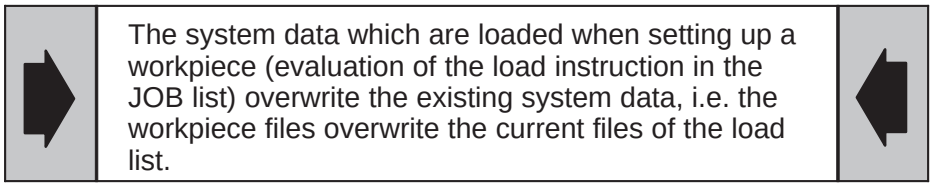
More than 1 chuck can be entered for each scenario.

You can continue the chucking scenario with a configurable M/H function (default M102), see Section 3.2.1.

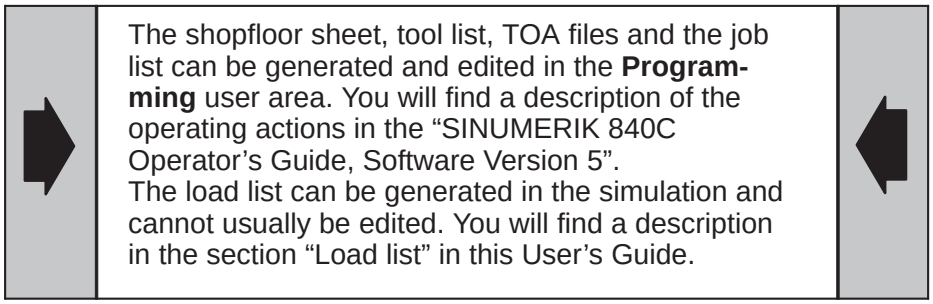
1) # is replaced by a serial number starting at 1

4.4 Including the load list

The system data defined in the load list are loaded when the **simulation is started or rebooted.** System data which are not entered in the load list are initialized
with 0.



The system data which are loaded when setting up a workpiece (evaluation of the load instruction in the JOB list) overwrite the existing system data, i.e. the workpiece files overwrite the current files of the load list.



The shopfloor sheet, tool list, TOA files and the job list can be generated and edited in the **Programming** user area. You will find a description of the operating actions in the “SINUMERIK 840C Operator’s Guide, Software Version 5”.
The load list can be generated in the simulation and cannot usually be edited. You will find a description in the section “Load list” in this User’s Guide.

4.5 Axis names

A maximum of 3 linear axes can be defined. The axis names defined in the machine data for the 1st, 2nd and 3rd axis are permitted within the programs to be simulated. The 1st, 2nd and 3rd axes must comprise an orthogonal coordinate system.

If only 2 axes are defined in the machine data, then only two axes are visualized (e.g. in turning).

- The assignment of the axis names to the workpiece axes X, Y and Z is fixed within the simulation:
  - Traversing movements of the 1st NC which is identified by the address names X or U axis in the direction of the workpiece axis X,
  - Traversing movements of the 2nd NC which is identified by the address names Y or V axis in the direction of the workpiece axis Y,
  - Traversing movements of the 3rd NC which is identified by the address names Z or W axis in the direction of the workpiece axis Z.

  In each case one address extension (e.g. X5, V2, Z3) is possible.
- In turning, two rotary axes with identifier C + extension (e.g. C1 and C2) can be defined. The rotary axes refer to spindles 1 and 2. The function **Rechuck** always switches to the other rotary axis, i.e. only one spindle can be visualized at a time.

■

5 Examples

5.1 Milling example

Preparing a DIN program for simulation

If a DIN program is simulated without the tool list and shopfloor sheet files, only the traversing paths of the tool are displayed without the unmachined part information. You can add the following files to a DIN program:

- Job list
- Shopfloor sheet and
- Tool list

In this way you can set up a workpiece completely.

This section uses a simple DIN program to describe how to set up a workpiece for the graphical simulation.

The following workpiece is to be simulated:

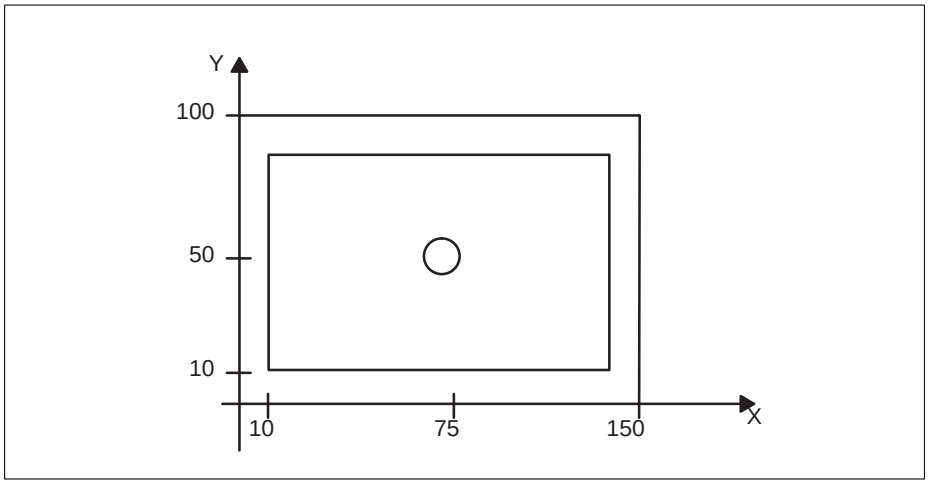
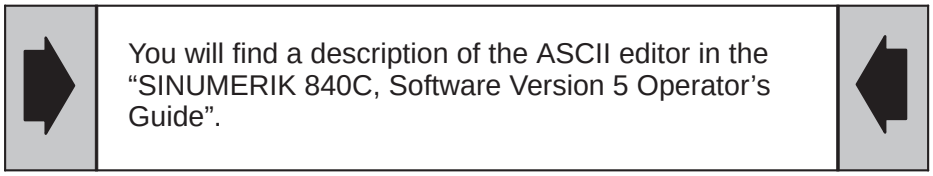


Fig. 5.1

The infeed depth for the recess is 10 mm.
The height of the unmachined part is 20 mm.
The diameter of the through-hole is 10 mm.



1st step
Workpiece program

You program the workpiece in the **Programming** area. You edit the program blocks using the ASCII editor. For example, generate a workpiece with the name TEST. Store all the files for the example workpiece under this workpiece.

The edited part program might look like this:

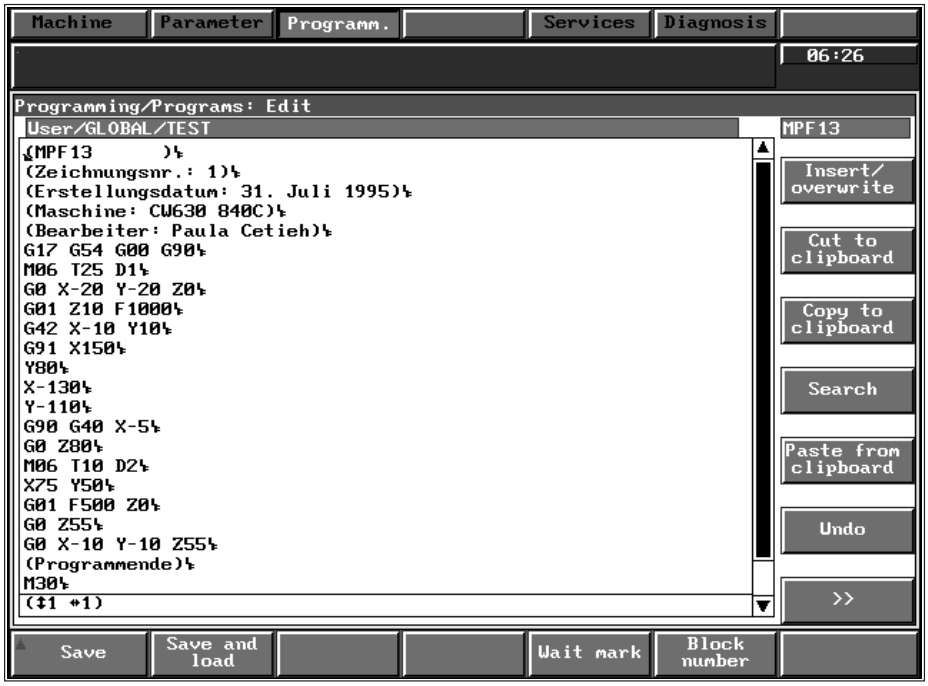


Fig. 5.2 Part program MPF13 for TEST workpiece

2nd step
Job list

Produce a joblist:

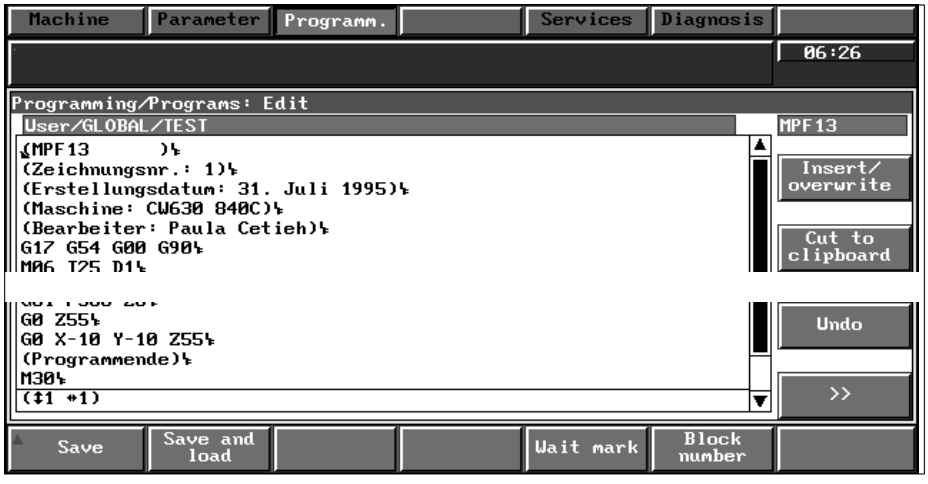


Fig. 5.3 JOB joblist for TEST workpiece

The job list assures that

- all the files of the workpiece are loaded and
- the program MPF13 is immediately selected for machining.

3rd step
Shopfloor sheet (work schedule)

Create a new file WERKSTA and enter the unmachined part information according to the drawing:

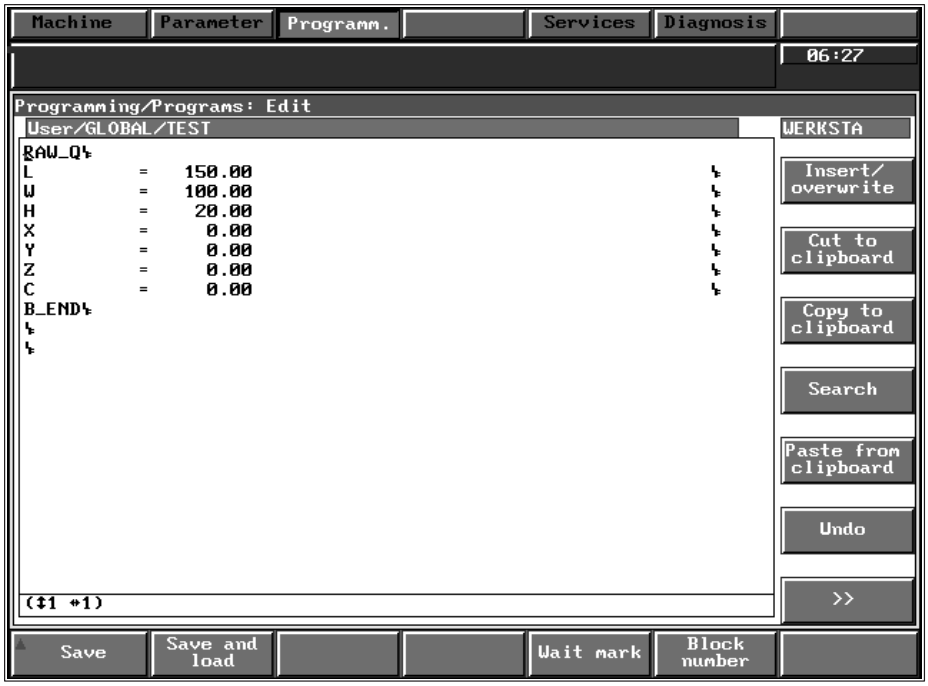


Fig. 5.4 WERKSTA shopfloor sheet for TEST workpiece

The reference point of the clamping position is identical to the machine zero (entry 0 in X, Y, Z).

4th step
Tool list

For the TEST workpiece you require two tools, a mill and a drill. The tool list might look like this:

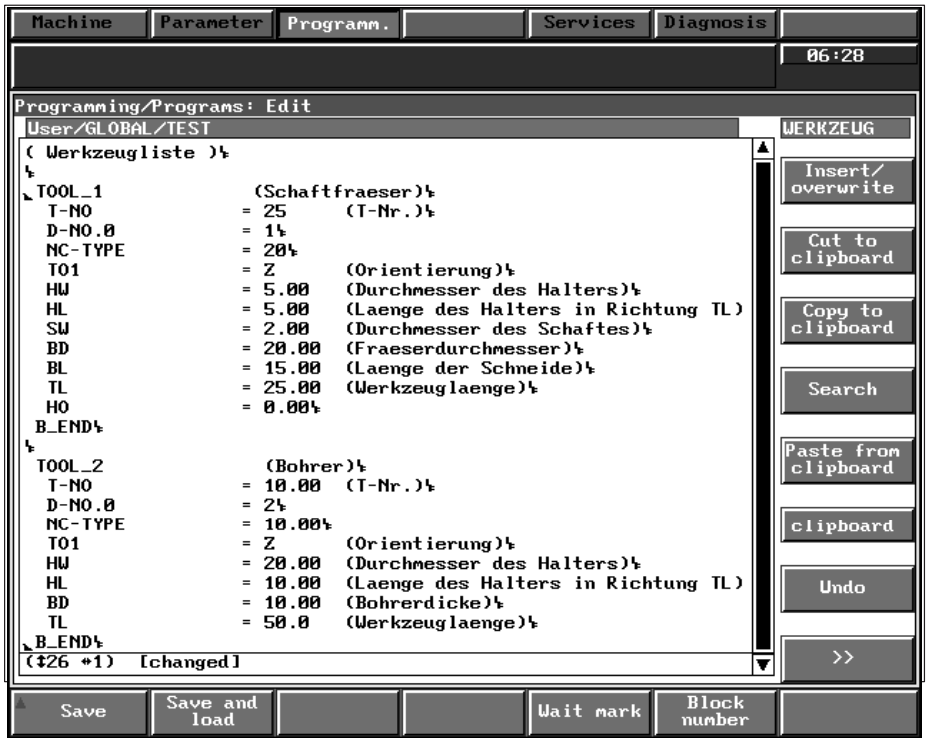


Fig. 5.5 WERKZEUG tool list for TEST workpiece

5th step
Start simulation

The workpiece is now set up. Provided that the workpiece and its files have been correctly generated and saved to the hard disk, the program can be simulated completely in the user area.

Switch over to simulation and load the TEST workpiece with **Select program/Select workpiece**.

If the standard set of parameters is used (e.g. no zero offsets are active), the following simulation output is generated:

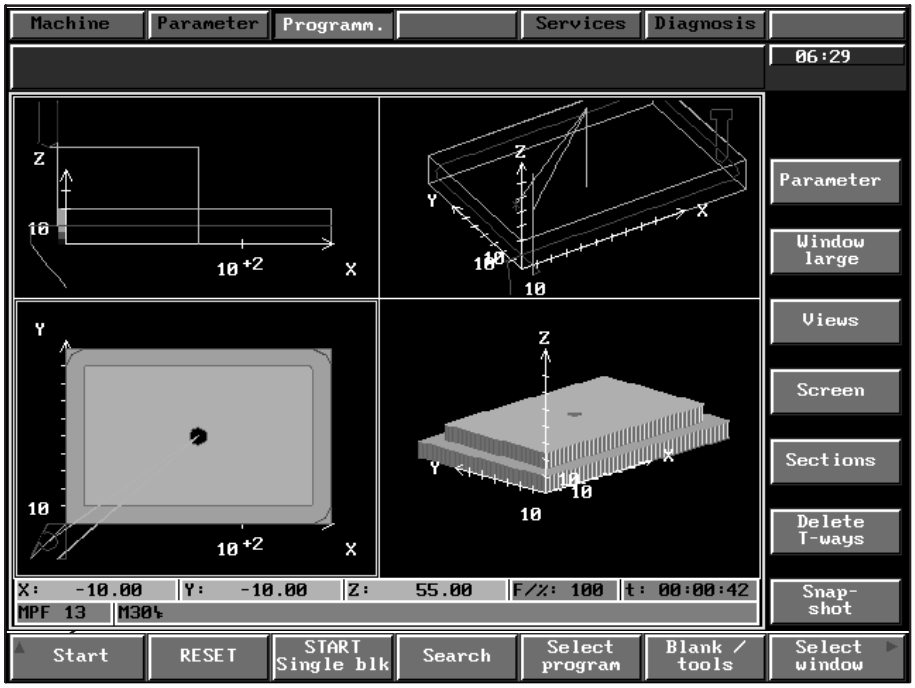


Fig. 5.6 Simulation output for the milling example

5.2 Turning example

The following turned part is to be simulated:

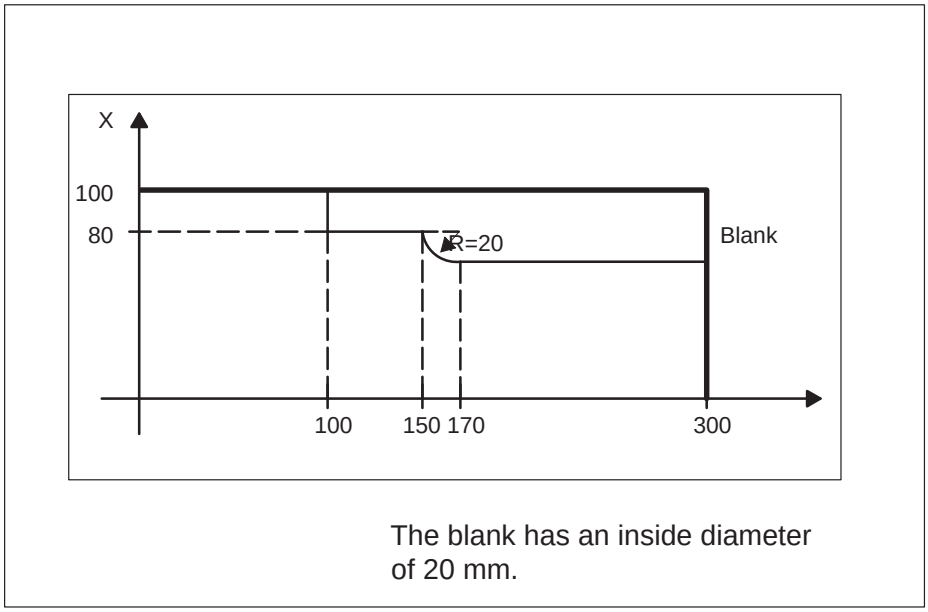


Fig. 5.7 Programming example for turning

1st step
Workpiece program

Diameter programming is used.

Write the following main program with the part program name “MPF123”:

```
N5 G0 X0 Z390
N10 T2 D21 X220
F0.1 S1000 M3
R20=123 (L)
R21=200 (Initial point vertical)
R22=0 (Initial point horizontal)
R24=1 (Final mach. allowance vertical)
R25=2 (Final mach. allowance horizontal)
R26=5 (Roughing depth)
R27=42 (Right of the contour)
R28=0.5
R29=41
R30=1
L95
M30
```

Write the following subroutine with the program name “SPF123”:

```
G01 X200 Z100
G01 X160 Z100
G01 X160 Z150
G03 X120 Z170 B20
G01 X120 Z300
M17
```

2nd step
Job list

Produce a job list:

```
LOAD.*
SELECT MPF123 CH=1
```

This job list assures that

- all the files of the workpiece are loaded and
- the program MPF123 is immediately selected for machining.

3rd step
Shopfloor sheet

Create a work schedule file and enter the information for the blank from the drawing. Then define the expected finished part contour:

Shopfloor sheet

Blank definition

```
RAW_Z
H                = 300.00
D                = 200.00
D1               =  20.00
X                =   0.00
Y                =   0.00
Z                =   0.00
C                =   0.00
B_END
```

Rechucking position (NPV2)

```
POS_P
X                =   0.00
Y                =   0.00
Z                =   200.00
A                =   0.00
B                =   0.00
C                =   0.00
B_END
```

Finished part contour

```
OUT_1 K
G01 X = 100 Z = 0
G01 X = 100 Z = 100
G01 X = 80 Z = 100
G01 X = 80 Z = 150
G03X = 60 Z = 170 I = 0 K = 20
G01 X = 60 Z = 300
B_END
```

4th step
Tool list

For the example turning operation you need a roughing/finishing tool. Then you must define the workholder

The tool list might look like this:

```
TOOL_1          (Roughing)
T–NO            = 2
D–NO.0          = 21
NC–TYPE.0       = 3
TO1             = +x
HW              = 50.0000       (Diameter, toolholder)
HL              = 100.0000      (Length, toolholder)
OO              = –57.0000      (Cross dimension)
SW              = 25.0000       (Shank width)
OL              = 85.0000       (Length dimension)
MCA             = 0             (Main tool edge
                                 angle)
TA              = –80.0000      (Tool tip angle)
TW              = 10.00         (Tool width)
SO              = –50.0000      (Distance to
                                 shank edge)
HT              = 3             (Position of
                                 reference point on
                                 toolholder)
RO              = 40.000        (Distance from
                                 reference point on
                                 toolholder)
B_END


CHUCK_1 K                       (Workholder)
G01 X = 140.00 Z = 60.0
G01 X = 140.00 Z = 25.0
G02 X = 160.00 Z = 5.00 I = 20.0
G01 X = 160.0 Z = 0.0
G01 X = 100.0 Z = 0.0
G01 X = 100.0 Z = 60.0
TRANS X = 0.0 Y = 0.0 Z = –10.0
B_END

CHUCK_ID_1                      (Chucking scenario)
1
B_END
```

5th step
Start simulation

The following simulation output is generated:

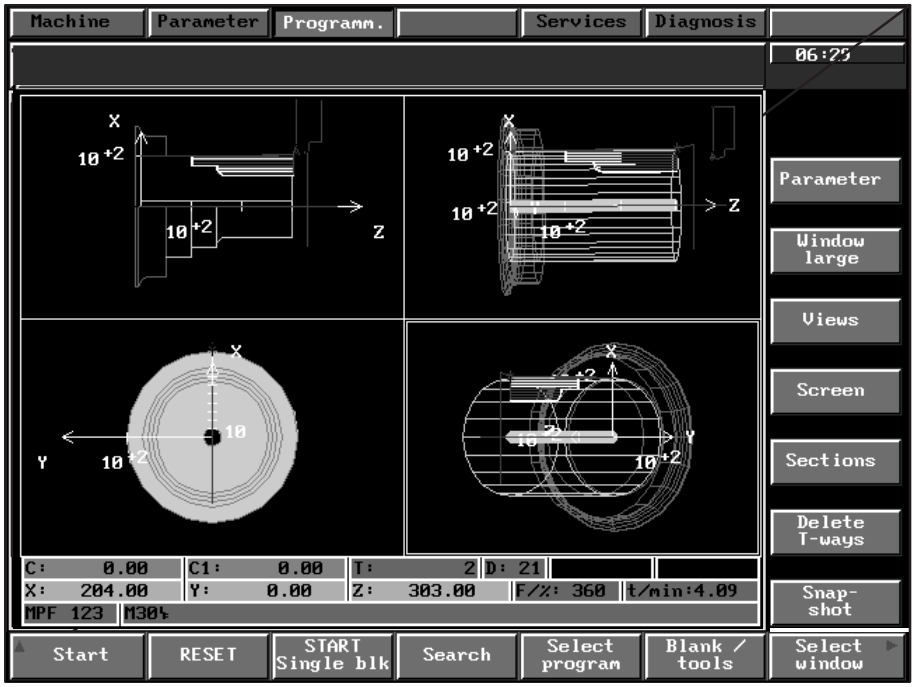


Fig. 5.8 Simulation output for the example

5.3 Multi-side machining

The following display illustrates some of the many simulation possibilities (relief cutting, multi-side machining, etc.).

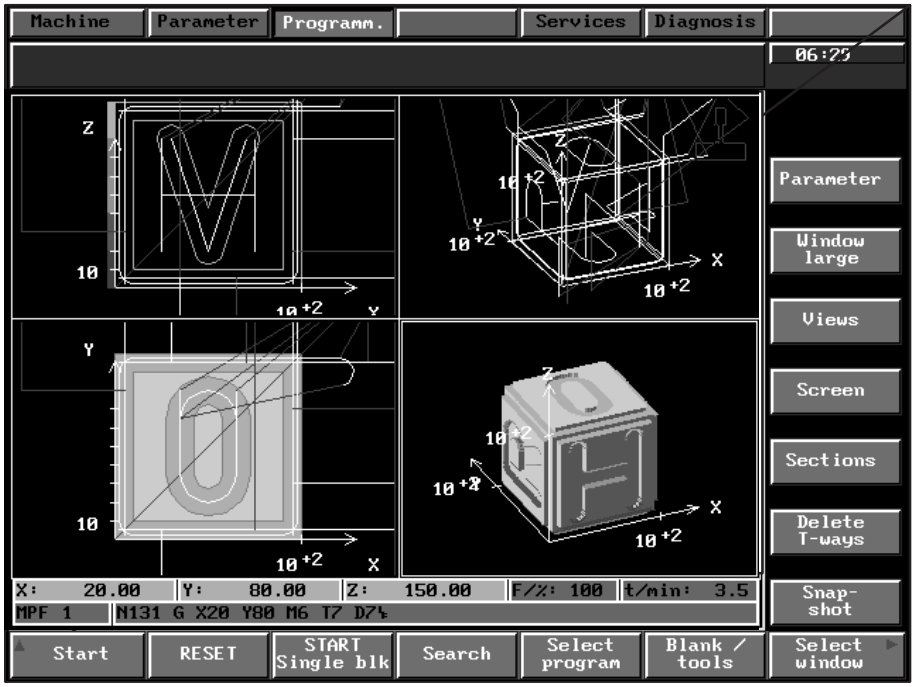
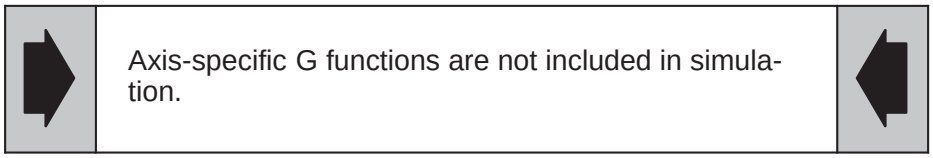


Fig. 5.9 Example for milling technology

■

6 List of DIN Programming Commands



Group	ISO	Code	Criterion	Function and meaning
1	G	00	s	Rapid traverse movement, cartesian programming
	G	01	s	Linear interpolation, cartesian programming; Basic setting after RESET, M02 and M30
	G	10	s	Rapid traverse movement, programmed in polar coordinates
	G	11	s	Linear interpolation, programmed in polar coordinates
	G	02	s	Circular interpolation, clockwise, cartesian programming
	G	03	s	Circular interpolation, counterclockwise, cartesian programming
	G	33	s	Thread cutting, lead constant
	G	34	s	Thread cutting, linear lead increase
	G	35	s	Thread cutting, linear lead decrease
	G	36	t	Thread cutting, position-controlled spindle
	G	06	s	Spline interpolation
	G	12	s	Circular interpolation clockwise, programmed in polar coordinates
	G	13	s	Circular interpolation counterclockwise, programmed in polar coordinates
2	G	09	s	Speed reduction, exact stop fine
3	G	17	s	Plane selection X – Y
	G	18	s	Plane selection Z – X
	G	19	s	Plane selection Y – Z
	G	16	s	Plane selection with free choice of axis

s ... Possible in simulation, included
e ... Cannot be simulated, error message is output
i ... Cannot be simulated, function is ignored with no error message
t ... Can only be simulated for turning

Group	ISO	Code	Criterion	Function and meaning
4	G	40	s	Deselection of tool-nose/cutter radius compensation
	G	41	s	Selection of tool-nose/cutter radius compensation to the left of the contour
	G	42	s	Selection of tool-nose/cutter radius compensation to the right of the contour
5	G	53	s	Suppression of zero offsets
6	G	54	s	Settable zero offset 1 / Coordinate rotation
	G	55	s	Settable zero offset 2 / Coordinate rotation
	G	56	s	Settable zero offset 3 / Coordinate rotation
	G	57	s	Settable zero offset 4 / Coordinate rotation
7	G	04	s	Dwell time in seconds or spindle revolutions
	G	14	e	See Programming Guide
	G	24	e	Dwell actual-value related to axes and spindles
	G	25	i	Minimum working area limitation
	G	26	i	Maximum working area limitation
	G	26S	s	Spindle speed limitation
	G	58	s	Programmable zero offset 1 / Coordinate rotation
	G	59	s	Programmable zero offset 2 / Coordinate rotation
	G	92S	s	**S..** Spindle speed setpoint limitation
	G	92T	i	**T..** Ramp-up time for thread cutting
	G	92P	t	Cylindrical interpolation
	G	92A	t	**A..** Start angle for multiple turn threads
	G	74	s	Reference point travel
	G	75	s	Set reference dimension
	G	200	e	Resynchronization of an axis on changing channel

s ... Possible in simulation, included
e ... Cannot be simulated, error message is output
i ... Cannot be simulated, function is ignored with no error message
t ... Can only be simulated for turning

Group	ISO	Code	Criterion	Function and meaning
8	G	60	i	Speed reduction, exact stop fine
	G	63	s	Tapping without encoder, feedrate 100%
	G	64	i	Contouring, block transition without speed reduction
	G	62	i	Contouring, block transition with speed reduction
9	G	70	s	Input measurement system in inches
	G	71	s	Input measurement system metric
10	G	80	s	Delete G81 to G89
	G	81	s	Cycle L81: Drilling, centering
	G	82	s	Cycle L82: Drilling, counterboring
	G	83	s	Cycle L83: Deep-hole drilling
	G	84	s	Cycle L84: Tapping with/without encoder
	G	85	s	Cycle L85: Bore 1
	G	86	s	Cycle L86: Bore 2
	G	87	s	Cycle L87: Bore 3
	G	88	s	Cycle L88: Bore 4
	G	89	s	Cycle L89: Bore 5
11	G	90	s	Dimensioning (absolute)
	G	91	s	Incremental dimensioning
	G	68	t	Absolute dimensioning along shortest path (rotary axis only)
12	G	94	s	Feedrate dimension: mm/min or inch/min
	G	95	s	Feedrate dimension: mm/rev or inch/rev
	G	96	s	Feedrate dimension: mm/rev or inch/rev and constant cutting speed in mm/min or feet/min
	G	97	s	Cancel G96, store last set speed
	G	98	t	C axis feedrate
	G	195	e	See Programming Guide
	G	295	e	Revolutional feedrate with reference to rotary axis/spindle (speed/feedrate actual value link)
13	G	147	s	Approach contour with straight line
	G	247	s	Approach contour in quarter-circle
	G	347	s	Approach contour in semi-circle
	G	148	s	Exit contour with straight line

s ... Possible in simulation, included
e ... Cannot be simulated, error message is output
i ... Cannot be simulated, function is ignored with no error message
t ... Can only be simulated for turning

Group	ISO	Code	Criterion	Function and meaning
13	G	248	s	Exit contour with quarter-circle
	G	348	s	Exit contour with semi-circle
	G	48	s	Exit contour as approach strategy
	G	110	s	Pole specification, relative to last set position
	G	111	s	Pole specification, absolute with respect to workpiece 0
	G	112	s	Pole specification, relative to last valid pole
	G	15	e	5D tool length compensation
14	G	50	s	Deselection of scale modification
	G	51	s	Scale modification
15	G	150	e	Deselection of coupled motion
	G	151	e	Coupled axis combination 1
	G	152	e	Coupled axis combination 2
	G	153	e	Coupled axis combination 3
	G	154	e	Coupled axis combination 4
	G	155	e	Coupled axis combination 5
	G	156	e	Coupled axis combination 6
	G	157	e	Coupled axis combination 7
	G	158	e	Coupled axis combination 8
	G	159	e	Coupled axis combination 9
16	G	130	s	Deselection of transformation
	G	131	s	Selection of TRANSMIT transformation, transmit combination 1
	G	133	e	Selection of 2D coordinate rotation transformation, transmit combination 1
	G	135	e	Selection of 3D coordinate rotation transformation, transmit combination 1
17	G	230	s	Deselection of transformation
	G	231	s	Selection of TRANSMIT transformation, transmit combination 1
	G	233	e	Selection of 2D coordinate rotation transformation, transmit combination 1
	G	235	e	Selection of 3D coordinate rotation transformation, transmit combination 1

s ... Possible in simulation, included
e ... Cannot be simulated, error message is output
i ... Cannot be simulated, function is ignored with no error message
t ... Can only be simulated for turning

Group	ISO	Code	Criterion	Function and meaning
18	G	330	s	Deselection of transformation
	G	331	s	Selection of TRANSMIT transformation, transmit combination 1
	G	333	e	Selection of DOUBLETRANSMIT transformation, transmit combination 1
	G	335	e	Selection of 3D coordinate rotation transformation, transmit combination 1
19	G	931	s	Contour definition: Circle-line-circle
	G	932	s	Contour definition: Line-circle with radius/chamfer
	G	933	s	Contour definition: Circle-line with radius/chamfer
	G	934	s	Contour definition: Circle with radius/chamfer
	G	935	s	Contour definition: 3-point definition with radius/chamfer
20	G	400... 403	t	G functions for rolling mill technology (for spindles only)
21	G	171	i	Fill up FIFO buffer
	G	172	i	See Programming Guide
22	G	175	s	Update of zero offsets, length compensation, angle of rotation in each block
	G	176	s	Freezing of zero offsets, length compensation and angle of rotation
	G	410	e	See Programming Guide
	G	411	e	See Programming Guide
	G	412	e	See Programming Guide
23	G	720	e	Deselect extended measuring
	G	721	e	Block change after completion of the programmed traverse movement
	G	722	e	Immediate block change and not influenced by measuring function
24	G	220– 222	i	Travel to fixed stop
25	G	450	s	TRC circular transitions
	G	451	s	Calculation of TRC intersection
26	G	455	s	Selection/deselection of TRC with transition
	G	456	s	Selection/deselection of TRC on direct path

s ... Possible in simulation, included
e ... Cannot be simulated, error message is output
i ... Cannot be simulated, function is ignored with no error message
t ... Can only be simulated for turning

Path information	ISO	Code	Criterion	Function and meaning
28	G	420	e	Switch off extended stop and retraction for axes and spindles or selectively for either axes or spindles
	G	421	e	Activate monitoring sources and enable responses
	G	422	e	Configure generator operation
	G	423	e	Configure stop by control
	G	424	e	Configure stop by drive alone
	G	425	e	Configure retraction
	G	426	e	Configure retraction for drive module
29	G	600	i	Exact stop coarse with G00
	G	620	i	Continuous-path mode with velocity reduction
	G	640	i	Continuous-path mode without velocity reduction
30	G	431	i	See Programming Guide
	G	432	i	See Programming Guide
	X		s	Path in mm, inch and degrees
	Y		s	Path in mm, inch and degrees
	Z		s	Path in mm, inch and degrees
	4.		s	Path in mm, inch and degrees (address A, B, C, E, U, V, W, Q including extension)
	A		s	Angle in degrees with contour definition
	A		s	Angle in degrees with polar coordinates
	U,B		s	Radius with circular and polar coordinate programming in mm/inch
	U,B		s	Corner with contour definition
	U,B		s	Chamfer with contour definition in mm/inch
	U,B		s	Radius with contour definition in mm/inch
	I		s	Circle parameter in mm/inch
	I		s	Thread lead in inch/mm
	J		s	Circle parameter in mm/inch
	J		s	Thread lead in inch/mm
	K		s	Circle parameter in mm/inch
	K		s	Thread lead in inch/mm

s ... Possible in simulation, included
e ... Cannot be simulated, error message is output
i ... Cannot be simulated, function is ignored with no error message
t ... Can only be simulated for turning

M Group	ISO	Code	Criterion	Function and meaning
1	M	00	s	Programmed stop; unconditional
	M	01	s	Programmed stop; conditional
2	M	02	s	End of program
	M	17	s	End of subroutine
	M	30	s	End of program
3	M	03	s	Spindle ON, rotation clockwise
	M	04	s	Spindle ON, rotation counterclockwise
	M	05	s	Spindle OFF
	M	19	s	Oriented spindle stop
	M	..	t	C axis on/off
4	M	36	i	Feed as programmed
	M	37	i	Feed reduced by 1:100
5	M	..9999	e	Machine-specific functions Exception: One M or H function for the tool change
	M	xx	s	M function for tool change (e.g. M06)
Cycles	**ISO**	**Code**	**Criterion**	**Function and meaning**
	L	81–89	s	See G functions: G81–G89
	L	93–99	t	Turning cycles
	L	900	s	Drilling pattern circle of holes
	L	901	s	Milling pattern groove
	L	902	s	Milling pattern elongated hole
	L	903	s	Rectangular pocket/circular pocket
	L	930	s	Mill circular pocket
	L	970	s	Cycle in process gauging

s ... Possible in simulation, included
e ... Cannot be simulated, error message is output
i ... Cannot be simulated, function is ignored with no error message
t ... Can only be simulated for turning

General	ISO	Code	Criterion	Function and meaning
	L		s	Subroutine call
	F		s	Feedrate in mm/min, inch/min, mm/rev, inch/rev, rpm
	F		s	Dwell time in seconds
	F		s	Thread lead increase and decrease in mm/rev and inch/ rev
	S		s	Spindle speed
	S		s	Constant cutting speed
	S		s	Spindle speed limitation
	S		s	Spindle stop in degrees
	S		s	Dwell time in spindle revolutions
	T		s	Tool number
	D		s	Select tool offset memory
	R		s	R parameter
	P		s	Pointer to R parameter
	P		s	Number of subroutine passes
	P		s	Relationship between working diameter and unit diameter for G92P cylindrical interpolation
	P		s	G51P scale factor
	+		s	Addition of R parameters
	–		s	Subtraction of R parameters
	*		s	Multiplication of R parameters
	/		s	Division of R parameters
	/		s	Skippable blocks
	()		s	Comment
	LF		s	End of block
	H		s	Help function (see M ... 999, page 6–8)
	K		s	Constant
	[ ]		e	Program coordination

s ... Possible in simulation, included
e ... Cannot be simulated, error message is output
i ... Cannot be simulated, function is ignored with no error message
t ... Can only be simulated for turning

Main Group	ISO	Code	Criterion	Function and meaning
0	@	040	s	Save number of R parameters to stack
	@	041	s	Save list of R parameters to stack
	@	042	s	Fetch number of R parameters from stack
	@	043	s	Fetch list of R parameters from stack
1	@	100	s	Absolute jump
	@	111	s	Case/Switch branch
	@	121	s	IF equal to THEN ELSE branch
	@	122	s	IF not equal to THEN ELSE branch
	@	123	s	IF greater than THEN ELSE branch
	@	124	s	IF greater than and equal to THEN ELSE branch
	@	125	s	IF less than and equal to THEN ELSE branch
	@	126	s	IF less or equal THEN ELSE branch
	@	127	s	IF true THEN ELSE branch
	@	128	s	IF not THEN–ELSE branch
	@	131.. 138	s	While loop with query conditions such as IF THEN ELSE
	@	141.. 148	s	Repeat loop with query conditions such as IF THEN ELSE
	@	151	s	FOR TO loop
	@	161	s	FOR DOWNTO loop
2	@	200	s	Delete R parameter
	@	201	s	Load R parameter with value
	@	202	s	Exchange R parameter values
	@	203	s	Bit scan in R parameter
	@	210	s	Clear machine input buffer (MIB)
	@	211	s	Load contents of MIB in R parameter
	@	212	s	Load value in MIB
3	@	300	s	Load machine data (0..4999) in R parameter
	@	301	s	Load machine data byte (5000..6999) in R parameter
	@	302	s	Load machine data bit (5000..6999) in R parameter
	@	303	s	Load cycle machine data in R parameter

s ... Possible in simulation, included
e ... Cannot be simulated, error message is output
i ... Cannot be simulated, function is ignored with no error message
t ... Can only be simulated for turning

Main Group	ISO	Code	Criterion	Function and meaning
	@	304	s	Load cycle machine data bytes in R parameter
	@	305	s	Load cycle machine data bits in R parameter
	@	306	s	Load PLC machine data (0..5999) in R parameter
	@	307	s	Load PLC machine data bytes (6000..8999) in R parameter
	@	308	s	Load PLC machine data bits (6000..8999) in R parameter
	@	310	s	Load setting data in R parameter
	@	311	s	Load setting data byte (5000..9999) in R parameter
	@	312	s	Load setting data bit (5000..9999) in R parameter
	@	313	s	Load cycle setting data in R parameter
	@	314	s	Load cycle setting data byte in R parameter
	@	315	s	Load cycle setting data bit in R parameter
	@	320	s	Load tool offset data in R parameter
	@	330	s	Load settable zero offsets in R parameter
	@	331	s	Load programmable zero offsets in R parameter
	@	332	e	Load external zero offsets in R parameter
	@	333	e	Load DRF offsets in R parameter
	@	334	e	Load PRESET offsets in R parameter
	@	336	e	Load total offsets in R parameter
	@	337	s	Load settable coordinate rotation angle in R parameter
	@	338	s	Load programmable coordinate rotation angle in R parameter
	@	345	s	Load programmed cutting speed in R parameter
	@	360	s	Load actual axis value (workpiece-specific) in R parameter
	@	361	s	Load actual axis value (machine-specific) in R parameter
	@	362	e	Load actual axis value (machine-specific) in R parameter with allowance for following error
	@	363	e	Load actual spindle position in R parameter

s ... Possible in simulation, included
e ... Cannot be simulated, error message is output
i ... Cannot be simulated, function is ignored with no error message
t ... Can only be simulated for turning

Main Group	ISO	Code	Criterion	Function and meaning
	@	364	e	Load actual spindle speed in R parameter
	@	367	s	Load G16 axis indices and leading spindle in R parameter
	@	36a	s	Load active D number in R parameter
	@	36b	s	Load active G function from group 0 in R parameter
	@	36e	s	Speed setpoint of spindle
	@	371		Acquisition of active signals:
			e	Block search active/dry run feedrate active
			e	Measurement input 1 or 2 active
			s	Simulation program yes/no
	@	372	s	Load current channel number in R parameter
	@	380	e	Load PLC input bit in R parameter
	@	381	e	Load PLC output bit in R parameter
	@	382	e	Load PLC flag bit in R parameter
	@	383	e	Load PLC data word bit in R parameter
	@	390	e	Load PLC input byte in R parameter
	@	391	e	Load PLC output byte in R parameter
	@	392	e	Load PLC I/O byte in R parameter
	@	393	e	Load PLC flag byte in R parameter
	@	394	e	Load left PLC data word byte in R parameter
	@	395	e	Load right PLC data word byte in R parameter
	@	3a0	e	Load PLC input word in R parameter
	@	3a1	e	Load PLC output word in R parameter
	@	3a2	e	Load PLC I/O word in R parameter
	@	3a3	e	Load PLC flag word in R parameter
	@	3a4	e	Load PLC timer in R parameter
	@	3a5	e	Load PLC counter in R parameter
	@	3b0	e	Load PLC data word (fixed-point value) in R parameter
	@	3b1	e	Load PLC data word (BCD) in R parameter
	@	3b2	e	Load PLC data word (floating-point value) in R parameter
	@	3c0	e	Load NC alarm in R parameter
	@	3d0	e	Load number of entered NC alarms in R parameter

s ... Possible in simulation, included
e ... Cannot be simulated, error message is output
i ... Cannot be simulated, function is ignored with no error message
t ... Can only be simulated for turning

Main Group	ISO	Code	Criterion	Function and meaning
4	@	400	s	Transfer R parameter to machine data (0..4999)
	@	401	s	Transfer R parameter to machine data bit (5000..6999)
	@	402	s	Transfer R parameter to machine data byte (5000..6999)
	@	403	s	Transfer R parameter to cycle machine data
	@	404	s	Transfer R parameter to cycle machine data byte
	@	405	s	Transfer R parameter to cycle machine data bit
	@	406	s	Transfer R parameter to PLC machine data (0..5999)
	@	407	s	Transfer R parameter to PLC machine data byte (6000..8999)
	@	408	s	Transfer R parameter to PLC machine data bit (6000..8999)
	@	410	s	Transfer R parameter to setting data
	@	411	s	Transfer R parameter to setting data byte (5000..9999)
	@	412	s	Transfer R parameter to setting data bit (5000..9999)
	@	413	s	Transfer R parameter to cycle setting data
	@	414	s	Transfer R parameter to cycle setting data byte
	@	415	s	Transfer R parameter to cycle setting data bit
	@	420	s	Transfer R parameter to tool offset memory
	@	423	s	Add R parameter to tool offset memory
	@	430	s	Transfer R parameter to settable zero offset
	@	431	s	Add R parameter to settable zero offset
	@	432	s	Transfer R parameter to programmable zero offset
	@	434	e	Transfer R parameter to DRF offset
	@	435	e	Transfer R parameter to PRESET offset

s ... Possible in simulation, included
e ... Cannot be simulated, error message is output
i ... Cannot be simulated, function is ignored with no error message
t ... Can only be simulated for turning

Main Group	ISO	Code	Criterion	Function and meaning
	@	437	s	Transfer R parameter to settable coordinate rotation
	@	438	s	Add R parameter to settable coordinate rotation
	@	439	s	Transfer R parameter to programmable coordinate rotation
	@	43a	s	Add R parameter to programmable coordinate rotation
	@	440	s	Transfer R parameter to axis position setpoint
	@	441	s	
	@	442	s	Transfer R parameter to spindle speed setpoint (not for turning)
	@	443	s	
	@	446	s	Transfer R parameter to radius setpoint
	@	447	s	Transfer R parameter to angle setpoint
	@	448	s	Transfer R parameter to interpolation thread lead parameter
	@	482	e	Transfer R parameter to PLC flag bit
	@	483	e	Transfer R parameter to PLC data word bit
	@	493	e	Transfer R parameter to PLC flag byte
	@	494	e	Transfer R parameter to left PLC data word
	@	495	e	Transfer R parameter to right PLC data word
	@	4a3	e	Transfer R parameter to PLC flag word
	@	4b0	e	Transfer R parameter (fixed-point value) to PLC data word
	@	4b1	e	Transfer R parameter (BCD value) to PLC data word
	@	4b2	e	Transfer R parameter (floating-point value) to PLC data word
	@	4c0	e	Display cycle alarm with R parameter setting
	@	4e1	e	Transfer R parameter to spindle acceleration constant
6	@	610	s	Determine value of R parameter
	@	613	s	Calculate square root of R parameter
	@	614	s	Find sum of the squares and then square root

s ... Possible in simulation, included
e ... Cannot be simulated, error message is output
i ... Cannot be simulated, function is ignored with no error message
t ... Can only be simulated for turning

Main Group	ISO	Code	Criterion	Function and meaning
	@	620	s	Increment R parameter
	@	621	s	Decrement R parameter
	@	622	s	Truncate R parameter
	@	630	s	Sine (R parameter)
	@	631	s	Cosine (R parameter)
	@	632	s	Tangent (R parameter)
	@	634	s	Arc-sine (R parameter)
	@	637	s	Angle between two vectors
	@	640	s	Natural logarithm of R parameter
	@	641	s	e^X with R parameter
	@	650	s	Logical OR of R parameters
	@	651	s	Logical XOR of R parameters
	@	652	s	Logical AND of R parameters
	@	653	s	Logical XOR and NOT of R parameters
	@	654	s	Logical NOT of R parameters
	@	655	s	Bit OR
	@	656	s	Bit XOR
	@	657	s	Bit AND
	@	658	s	Bit AND followed by NOT
	@	659	s	Bit NOT
	@	660	s	Reset bit
	@	661	s	Set bit
	@	671		Set bit if test values are equal
	@	672	s	Set bit if test values are not equal
	@	673	s	Set bit if Var2 greater than value
	@	674	s	Set bit if Var2 greater than/equal to value
	@	675	s	Set bit if Var2 less than value
	@	676	s	Set bit if Var2 less than/equal to value

Note Mathematical operations are performed without allowance for the NCK area limits.

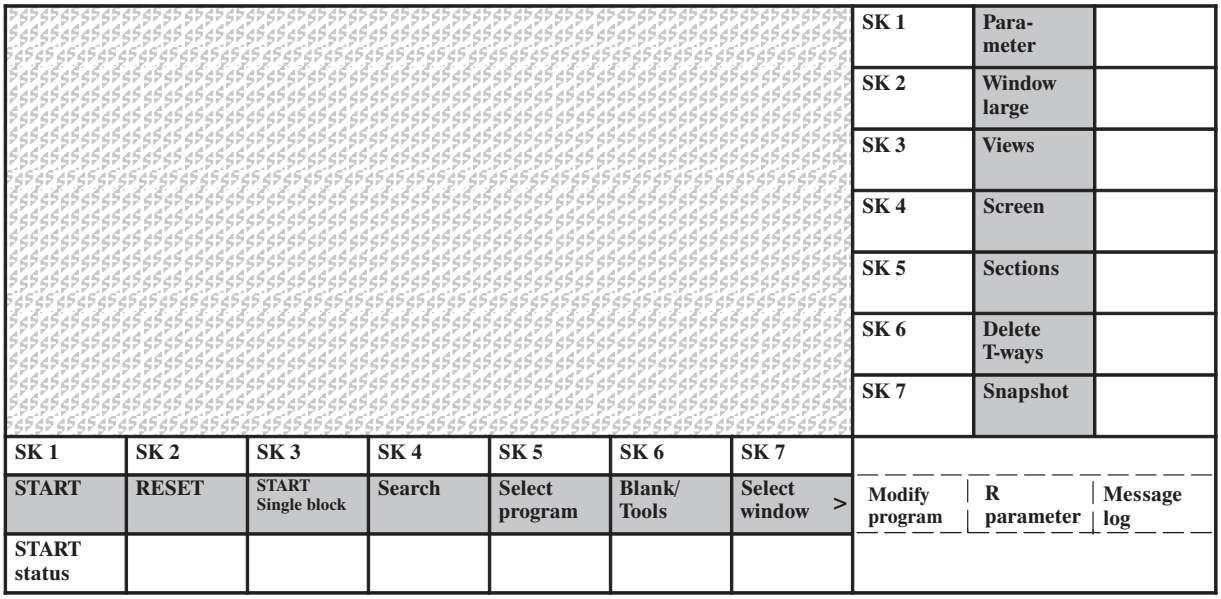
s ... Possible in simulation, included
e ... Cannot be simulated, error message is output
i ... Cannot be simulated, function is ignored with no error message
t ... Can only be simulated for turning

Main Group	ISO	Code	Criterion	Function and meaning
7	@	706	s	Approach setpoint position in machine coordinates, same as G53
	@	710	t	Break down subroutine into individual blocks
	@	711	t	Calculate intersection
	@	713	s	Define safety distance in current input format
	@	714	s	STOP decoding
	@	715	s	STOP decoding 2
	@	720	i	Determine actual values of travelling axes on measurement probe signal

■

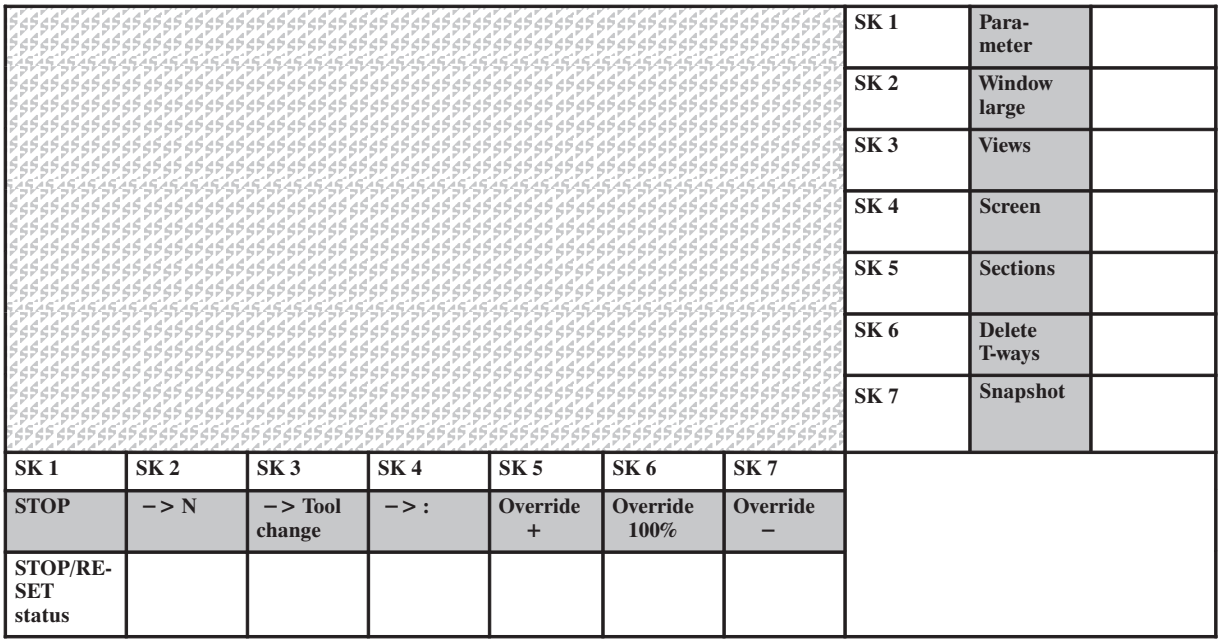
s ... Possible in simulation, included
e ... Cannot be simulated, error message is output
i ... Cannot be simulated, function is ignored with no error message
t ... Can only be simulated for turning

7 Menu Trees



							SK 1	Para-meter	
							SK 2	Window large	
							SK 3	Views	
							SK 4	Screen	
							SK 5	Sections	
							SK 6	Delete T-ways	
							SK 7	Snapshot	
SK 1	SK 2	SK 3	SK 4	SK 5	SK 6	SK 7			
START	RESET	START Single block	Search	Select program	Blank/ Tools	Select window >	Modify program	R parameter	Message log
START status									

Fig. 7.1 Basic status: STOP/RESET



							SK 1	Para-meter	
							SK 2	Window large	
							SK 3	Views	
							SK 4	Screen	
							SK 5	Sections	
							SK 6	Delete T-ways	
							SK 7	Snapshot	
SK 1	SK 2	SK 3	SK 4	SK 5	SK 6	SK 7			
STOP	–> N	–> Tool change	–> :	Override +	Override 100%	Override –			
STOP/RE-SET status									

Fig. 7.2 Status: START

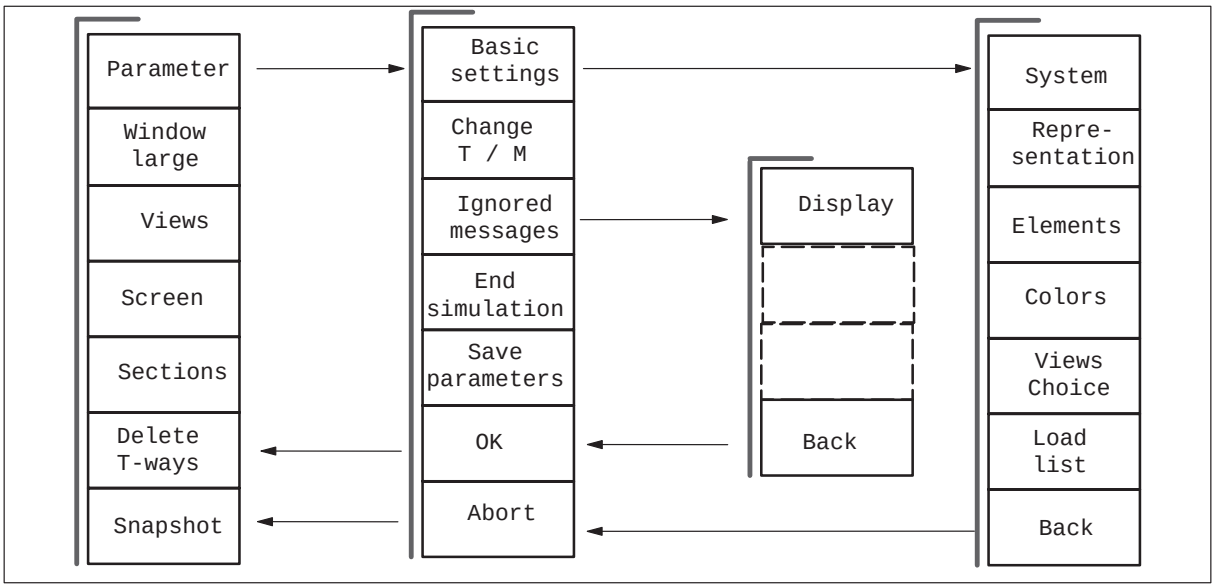


Fig. 7.3 Status: STOP/RESET – selection parameters

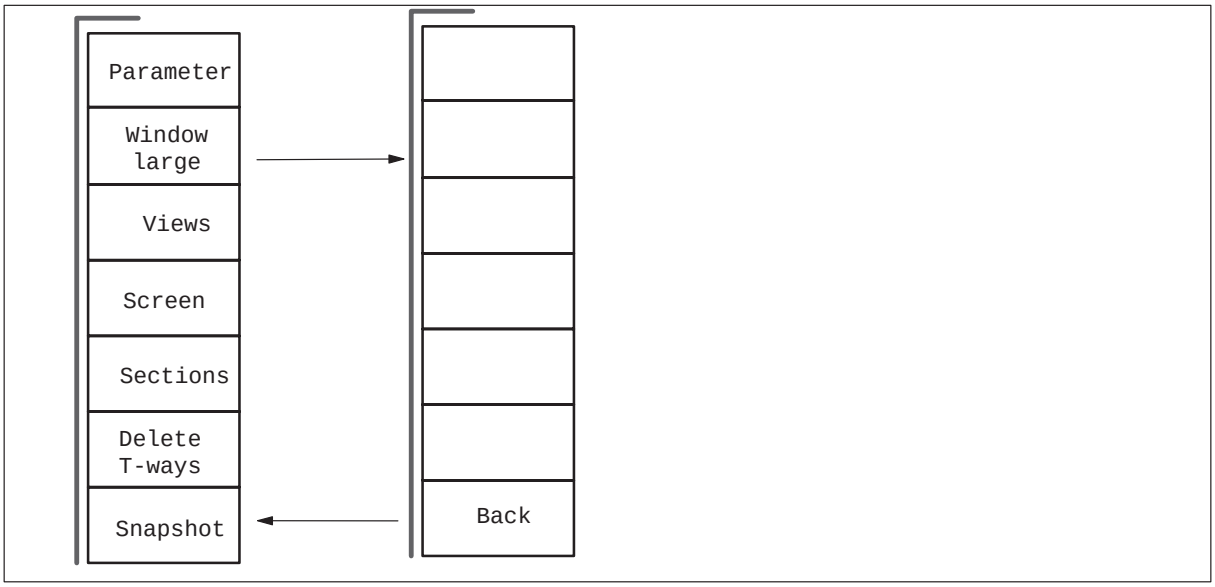


Fig. 7.4 Status: STOP/RESET – selection large

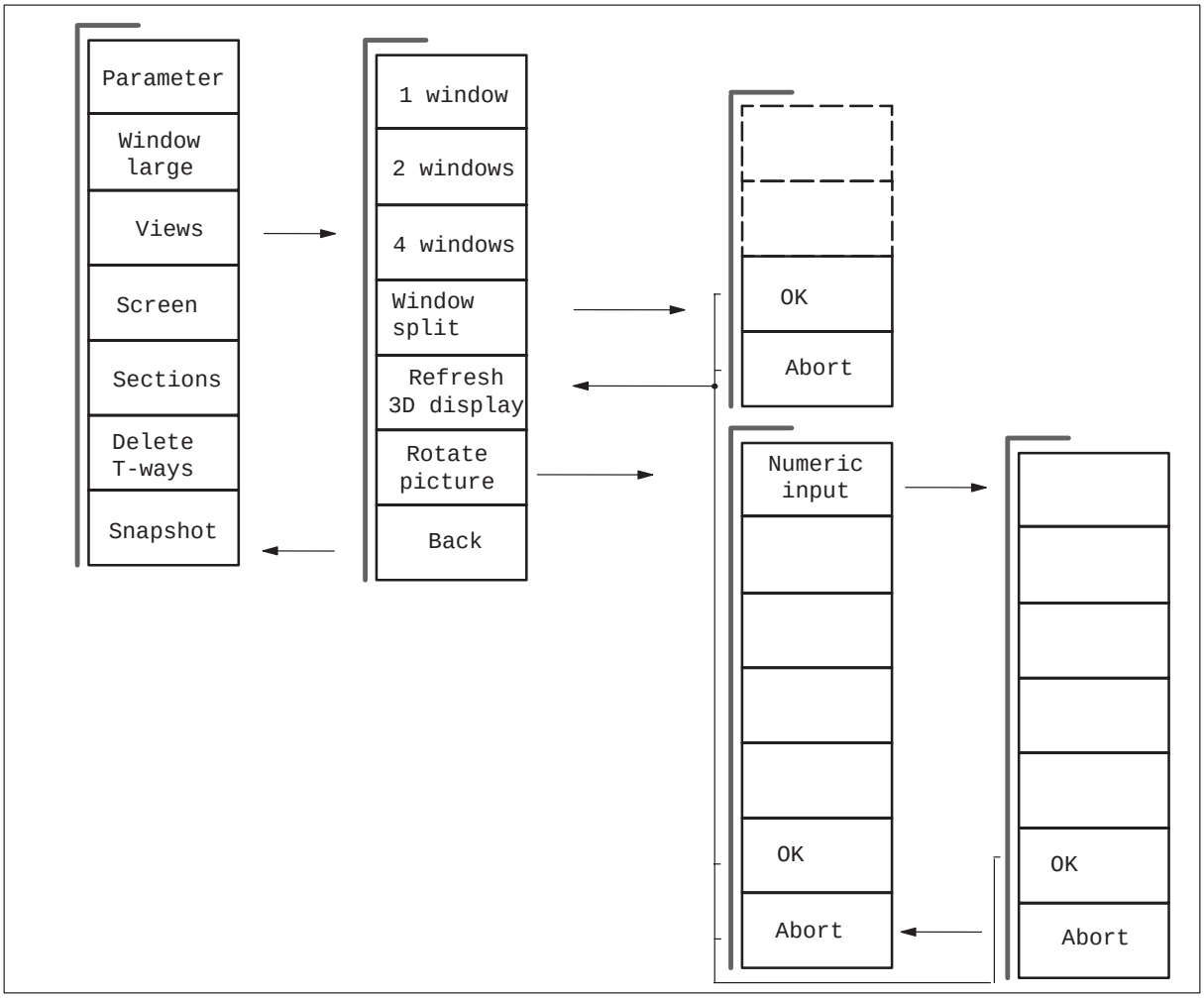


Fig. 7.5 Status: STOP/RESET – views

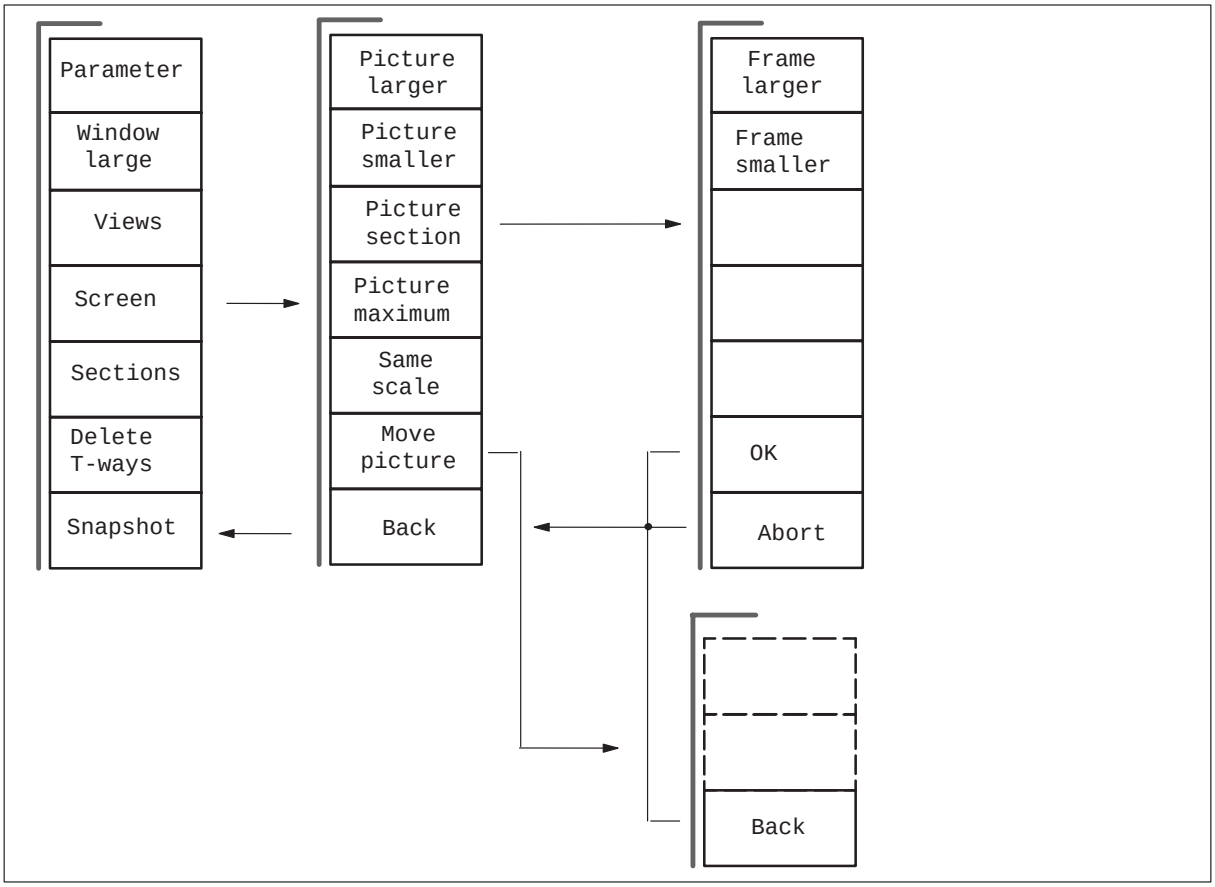


Fig. 7.6 Status: STOP/RESET – screen

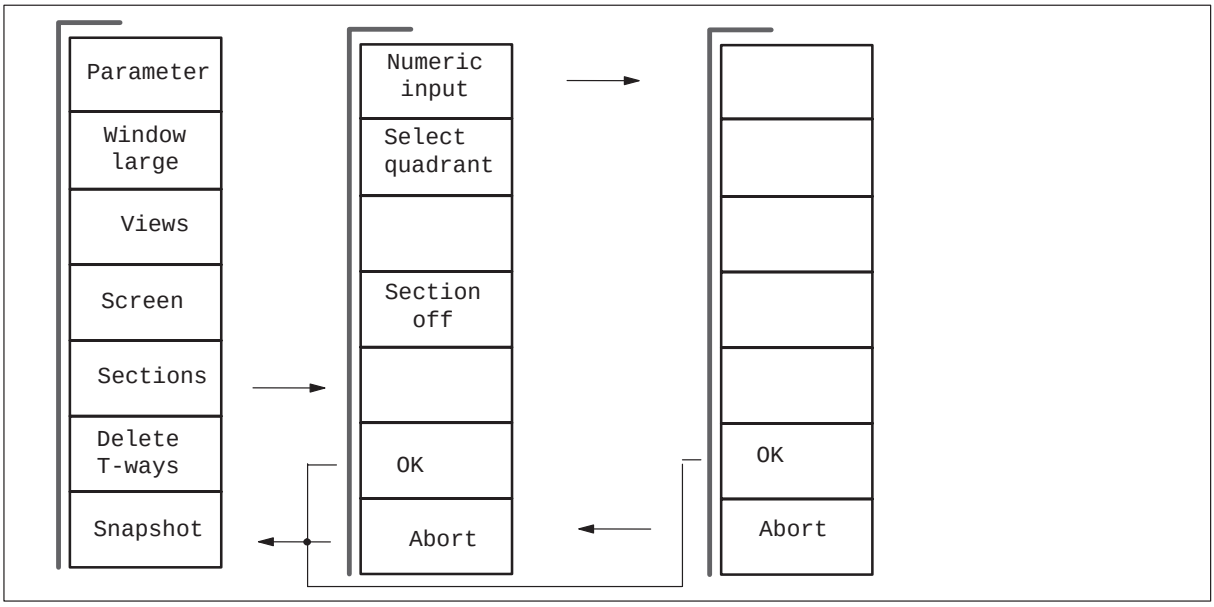


Fig. 7.7 Status: STOP/RESET – sections