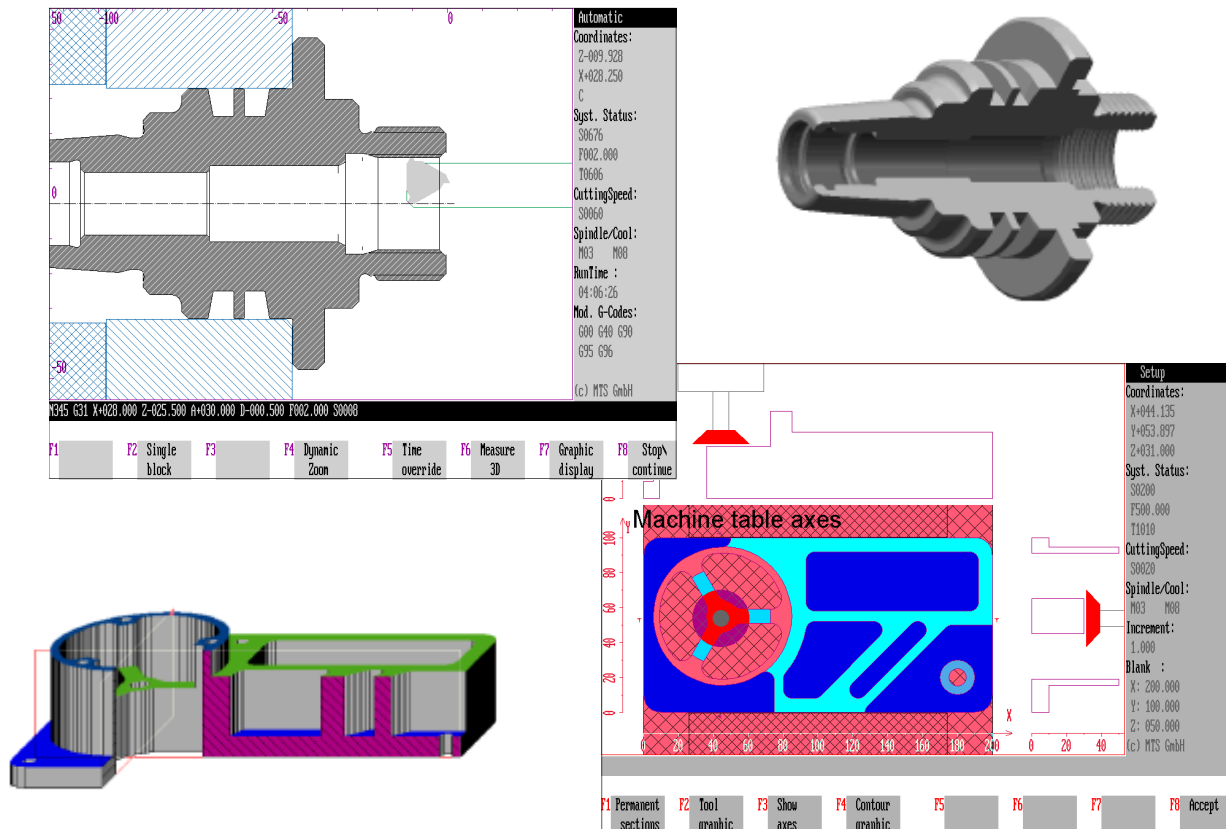


# CNC Simulator Turning & Milling Operation Manual



Version 5

CNC Simulator Turning & Milling  
Operation Manual

© MTS Mathematisch Technische Software-Entwicklung GmbH  
Kaiserin-Augusta-Allee 101 • D-10553 Berlin • Germany  
☎: +49 / 30 / 349 960 - 0 • Fax: +49 / 30 / 349 960 - 25 • eMail: mts@mts-cnc.com

All rights reserved, including photomechanical reproduction and storage on electronic media

DIN (Deutsche Industrie Norm), is the German Standard Specification as defined by the "Deutsches Institut für Normung e.V."

PAL is short for "Prüfungs- Aufgaben und Lehrmittelentwicklungsstelle" (Institute for the Development of Examination Standards and Training Aids), a division of the "IHK Mittlerer Neckar" (Chamber of Industry and Commerce of the Middle-Neckar Region)

## TABLE OF CONTENTS

<b>About this Manual</b>	<b>1- 1</b>
<b>1. System Overview</b>	<b>1- 2</b>
<b>1.1 The Workplace</b>	<b>1- 3</b>
<b>1.2 Input Options</b>	<b>1- 7</b>
<b>1.3 System Functions</b>	<b>1- 9</b>
CNC Lathe	1- 9
CNC Milling Machine	1- 11
Collision Monitoring	1- 13
Operating Modes	1- 13
Special Functions	1- 15
Data Management	1- 15
<b>2. Introduction to Operation</b>	<b>2- 1</b>
<b>2.1 Installation and Configuration</b>	<b>2- 1</b>
<b>2.2 System Start for MTS Programs</b>	<b>2- 1</b>
Starting the CNC Simulator	2- 3
Turning and Milling Menu	2- 5
<b>2.3 Screen Display</b>	<b>2- 7</b>
<b>2.4 Menu Functions</b>	<b>2- 11</b>
Shortened Menus	2- 15
Using CNC Keyboard or Tablet	2- 15
<b>2.5 Editing Menus</b>	<b>2- 17</b>
Entries in Information Column	2- 17
Entering NC Blocks	2- 17
NC Editor	2- 19
File Selection Window	2- 19
<b>2.6 Quitting CNC Simulator</b>	<b>2- 22</b>
<b>3. CNC Simulator Main Menu</b>	<b>3- 1</b>
<b>3.1 Setup Sheet and Automatic Setup</b>	<b>3- 5</b>
3.1.1 Setup Sheet Syntax Rules	3- 7
3.1.2 Setup Sheet Default Settings	3- 13
3.1.3 Create Setup Sheet	3- 15
3.1.4 Delete Setup Sheet	3- 15
3.1.5 Translate Setup Sheet	3- 17
3.1.6 Automatic Setup	3- 17
<b>3.2 Status Manager</b>	<b>3- 19</b>
Load Status	3- 20
Save Status	3- 20
Delete Status	3- 20

<b>4. Setup Mode</b>	<b>4- 1</b>
Setup Mode Menu	4- 3
<b>4.1 Blank / Workpiece</b>	<b>4- 9</b>
4.1.1 CNC Turning: Blank Changing	4- 11
Specifying Blank Dimensions	4- 13
4.1.2 CNC Milling: Blank Changing	4- 15
Specifying Blank Dimensions	4- 15
Perspective Views	4- 17
Change Viewing Point	4- 17
4.1.3 Workpiece Manager	4- 19
Inserting Existing Workpiece	4- 19
Saving/Deleting Preproduced Workpieces	4- 21
<b>4.2 Clamping Fixtures</b>	<b>4- 23</b>
4.2.1 CNC Turning: Clamping Fixtures	4- 25
Selecting Clamping Fixtures	4- 25
Changing Clamping Element	4- 27
Defining Clamping Method	4- 31
Clamping with Jaw Chuck	4- 31
Positioning Tailstock and Sleeve	4- 33
Reclamping Workpiece	4- 35
Tailstock	4- 35
4.2.2 CNC Turning: Clamping Fixture Manager	4- 37
Management Menu	4- 39
Define Clamping Element	4- 41
Input Parameters	4- 43
Copy Data Record	4- 47
Modify Clamping Element	4- 49
Delete Clamping Element	4- 51
Select Clamping Fixture	4- 53
4.2.3 CNC Milling: Clamping Fixtures	4- 55
Select Clamping System	4- 55
Changing Vice	4- 57
Rotating Vice	4- 61
Magnetic Plate Clamping	4- 61
Modular Clamping	4- 63
Define Clamping	4- 67
Reclamping	4- 69
4.2.4 CNC Milling: Clamping Fixture Manager	4- 71
Management Menu	4- 73
Define Clamping Element	4- 75
Copy Data	4- 75
Vice Input Parameters	4- 77
Modify Clamping Element	4- 79
Delete Clamping Element	4- 81
Select Clamping Elements	4- 83
<b>4.3 Tooling and Tool Change</b>	<b>4- 85</b>
4.3.1 Tool Change	4- 87
4.3.2 Turret/Magazine Configuration	4- 89
Information	4- 91
Valid Compensation Values	4- 91
Tool Engagement Times	4- 93
4.3.3 CNC Turning: Turret Tooling	4- 95
4.3.4 CNC Milling: Magazine Tooling	4- 99
4.3.5 Compensation Value Register	4-105

<b>4.4</b>	<b>Zero and Reference Points</b>	<b>4-111</b>
4.4.1	Reference Point	4-113
4.4.2	Workpiece Zeropoint	4-115
<b>4.5</b>	<b>Machine and Technology Functions</b>	<b>4-119</b>
	Speed and Feedrate	4-121
4.5.1	Spindle and Coolant	4-121
4.5.3	Increment Values	4-123
<b>4.6</b>	<b>Override/Times</b>	<b>4-125</b>
<b>4.7</b>	<b>Machining with Manual Control</b>	<b>4-127</b>
4.7.1	CNC Turning	4-127
4.7.2	CNC Milling	4-128
<b>5.</b>	<b>Tool Management</b>	<b>5- 1</b>
<b>5.1</b>	<b>CNC Turning: Define/Delete Tools</b>	<b>5- 3</b>
5.1.1	Creating New Tools	5- 5
	Data Entry: Tool with Reversible Tip or Drill	5- 5
	Copying Existing Tool Data	5- 9
	Further Steps in Tool Definition	5- 11
	Cutting Edge Point Definition	5- 13
	Defining Toolholder	5- 17
	Defining Tool Adapter	5- 19
	Zoom Function	5- 21
5.1.2	Modifying Tools	5- 23
5.1.3	Deleting Tools	5- 25
<b>5.2</b>	<b>CNC Milling: Define/Delete Tools</b>	<b>5- 27</b>
5.2.1	Creating New Tools	5- 29
	Data Entry: Tool	5- 29
	Copying Existing Tool Data	5- 33
	Available Tools: Search & List	5- 35
5.2.2	Modifying Tools	5- 37
5.2.3	Deleting Tools	5- 39
5.2.4	Creating a New Tool Adapter	5- 41
	Data Entry: Tool Adapter	5- 41
	Copying Existing Tool Adapter Data	5- 43
	Available Tool Adapters: Search & List	5- 45
5.2.5	Modifying a Tool Adapter	5- 47
5.2.6	Deleting a Tool Adapter	5- 47
<b>6.</b>	<b>Automatic Mode</b>	<b>6- 1</b>
	Automatic Mode Menu Structure	6- 3
<b>6.1</b>	<b>Starting Machining Simulation</b>	<b>6- 9</b>
	Selecting NC Program	6- 9
	Selecting Simulation Mode	6- 11
	Complete Automatic Run	6- 11
	Single NC Block Mode	6- 13
	Collision Monitoring	6- 13
<b>6.2</b>	<b>Selecting Program Range for Simulation</b>	<b>6- 15</b>

<b>6.3</b>	<b>Changing Simulation Mode</b>	<b>6- 17</b>
	Switching from Automatic Run to Single Block Mode	6- 17
	Switching from Single Block to Interactive Mode	6- 19
<b>6.4</b>	<b>Tool Path Indication</b>	<b>6- 21</b>
<b>6.5</b>	<b>Override/Times</b>	<b>6- 23</b>
<b>6.6</b>	<b>Measuring, 3D View</b>	<b>6- 25</b>
<b>6.7</b>	<b>Modifying Graphic Display, and Zoom Function</b>	<b>6- 27</b>
	CNC Turning	6- 27
	CNC Milling	6- 29
<b>6.8</b>	<b>Programming Aids</b>	<b>6- 31</b>
<b>7.</b>	<b>NC Programming</b>	<b>7- 1</b>
	Introduction	7- 1
	Programming Code	7- 3
	Format Conversion	7- 5
<b>7.1</b>	<b>NC Editor</b>	<b>7- 9</b>
7.1.1	NC Program Management	7- 11
	Selecting NC Programs	7- 11
	Program Management User Dialogues	7- 13
7.1.2	NC Editor Programming Interface	7- 15
7.1.3	Editing Functions	7- 18
7.1.4	Program Functions	7- 23
	Linking NC Programs	7- 23
	Block Group Handling	7- 25
	Modify Editing Range	7- 27
	Renumbrer NC Blocks	7- 29
	Switching to WOP Interface	7- 29
	Programming Aids	7- 31
	Find Character String	7- 31
	End NC Programming	7- 33
7.1.5	Syntax Check and Range Violation	7- 33
7.1.6	Free Format Mode	7- 35
	Temporary Free Format	7- 35
<b>8.</b>	<b>Interactive Programming</b>	<b>8- 1</b>
	Screen Layout	8- 3
<b>8.1</b>	<b>Start Interactive Programming</b>	<b>8- 5</b>
	Load Existing NC Program	8- 5
	Create and Load New NC Program	8- 5
	Load Selected Part of NC Program	8- 7
<b>8.2</b>	<b>Programming NC Blocks</b>	<b>8- 9</b>
	Interactive Check of NC Blocks	8- 13
	Collision Monitoring	8- 13
<b>8.3</b>	<b>Switching between Programming Modes</b>	<b>8- 15</b>
	Special Functions	8- 17

<b>8.4</b>	<b>Teach-In Mode</b>	<b>8- 19</b>
8.4.1	CNC Turning: Specifying Nominal Contour	8- 21
8.4.2	Enter Technology Data	8- 23
8.4.3	Programming Workpiece Zero	8- 25
8.4.4	Machining with Manual Control	8- 27
	CNC Turning: Angle Input	8- 29
	Cancel Last Travel Command	8- 29
	Change Increment	8- 30
	Return to Interactive Programming	8- 30
<b>9.</b>	<b>Measuring</b>	<b>9- 1</b>
	Measuring Menu	9- 3
	Special Graphic Symbols and Markings	9- 5
<b>9.1</b>	<b>Element Dimensioning</b>	<b>9- 9</b>
<b>9.2</b>	<b>Point Dimensioning</b>	<b>9- 13</b>
<b>9.3</b>	<b>Defining Dimension Reference Point</b>	<b>9- 17</b>
	Incremental Dimensioning	9- 19
<b>9.4</b>	<b>Measuring Configuration</b>	<b>9- 21</b>
	Position Dimensioning Frame: "Q point"	9- 23
	Dimension Values: Places after Decimal Point	9- 25
<b>9.5</b>	<b>Surface Roughness</b>	<b>9- 27</b>
	Menu: Peak-to-Valley Height	9- 29
	Peak-to-Valley Height Definition: Measuring Distance	9- 31
9.5.1	Modifying Screen Display	9- 33
<b>10.</b>	<b>Modification of Graphic Display</b>	<b>10- 1</b>
<b>10.1</b>	<b>3D Display</b>	<b>10- 3</b>
10.1.1	NC Turning: 3D Display	10- 5
	Select Viewing Angle	10- 5
	Select Display Mode	10- 5
	Select Section	10- 7
	Menu: 3D Display	10- 9
10.1.2	NC Milling: Sectional and 3D Display	10-11
	Select Sectional Position	10-11
	Select Display Mode	10-13
	Select Viewing Direction	10-13
	Menu: 3D View	10-15
	Defining Machining Depths: Z Section Dotted	10-15
<b>10.2</b>	<b>Changing Graphic Display</b>	<b>10-17</b>
10.2.1	NC Turning: Changing Display	10-17
10.2.2	NC Milling: Changing Display	10-19
10.2.2	Displaying NC Lines	10-23
<b>10.3</b>	<b>Zooming</b>	<b>10-25</b>
	Menu: Zooming	10-27
	Defining Zoom Detail	10-29
<b>10.4</b>	<b>Color Settings</b>	<b>10-31</b>
10.4.1	Color Setting for CNC Simulation	10-31
10.4.2	Color Setting for WOP, Measuring, Depth of Roughness/ Peak-to-Valley Height Definition	10-39

<b>11. Miscellaneous Functions</b>	<b>11- 0</b>
<b>11.1 Programming Aids</b>	<b>11- 1</b>
Names/Designations and Group Assignments	11- 3
CNC Turning Programming Aids	11- 4
CNC Milling Programming Aids	11- 6
Using Help Function	11- 9
NC Programming with Help Functions	11-11
<b>11.2 Printing Graphics</b>	<b>11-13</b>
<b>12. Workshop-Oriented Programming</b>	<b>12- 1</b>
Introduction	12- 1
Starting WOP Interface	12- 5
Available User Options	12- 7
<b>12.1 Menu Levels</b>	<b>12- 9</b>
Main Menu	12- 9
Element Menu 1	12-11
Element Menu 2	12-13
Input Menu for Geometry Definition	12-15
<b>12.2 Programming Contour Elements</b>	<b>12-17</b>
Editing Functions in Input Menus	12-17
Input Parameters for "Linear Path"	12-18
Input Parameters for "Circular Arc"	12-19
Selecting Alternatives	12-23
<b>12.3 Contour Modification</b>	<b>12-25</b>
Corrections during Parameter Input	12-25
Delete Element	12-27
Reset	12-29
Accept Background Elements	12-31
Copy Existing NC Blocks	12-33
<b>12.4 Programming Based on Non-Standard Dimensioning</b>	<b>12-35</b>
Open Contours and Multiple-Point Definitions	12-37
<b>Appendix 1: CNC Symbols and Miscellaneous Icons</b>	<b>A- 0</b>
<b>Appendix 2: List of Error Messages</b>	<b>A- 9</b>
<b>Appendix 3: Register</b>	<b>A- 37</b>



## About this Manual

This "**CNC Simulator Operating Manual**" is part of the MTS software documentation package which explains the functions of the CNC Simulators for turning and milling. In addition to the operating manual, this documentation package includes the programming instructions for CNC turning and CNC milling, the configuration instructions and the manuals for the postprocessors, the freely definable programming keys and much more.

One important change since version 4.1 concerns the structure of this operating manual. The programming instructions are discussed separately as turning and milling sections, however, we have **combined** the software operations to avoid unnecessary repetition. To make it easier for you and to reduce your learning period, the two CNC Simulators have identical screen arrangements, operating concepts and functional software structure. For this reason it was reasonable to combine the descriptions of the two technologies as well. This operating manual generally

applies to both turning and milling. In cases where only one of the two technologies is meant, it is indicated by graphics and text notes.

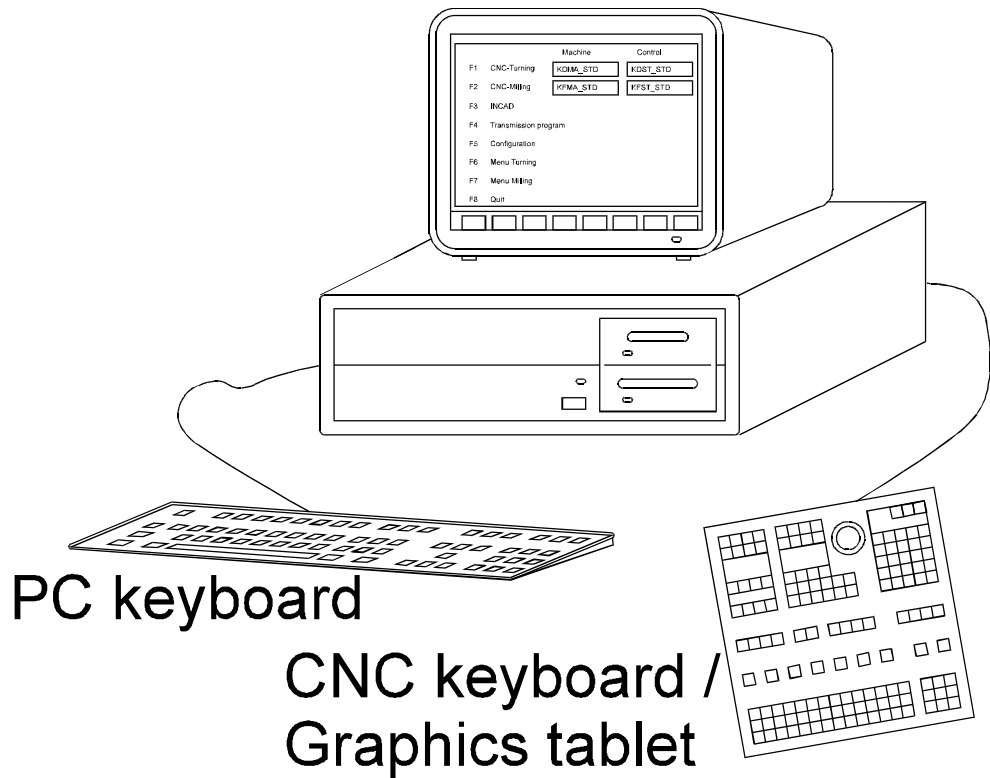
The operating manual provides an extensive introduction to the software and serves both as a quick-start guide and a reference. In individual descriptions and chapters each step is presented as an independent unit of a larger context, however without losing its contribution to the whole. For the sake of clarity, the descriptions are further divided into sections: introduction, call procedure, processing/editing and notes. This makes it easier for you to work through whole chapters or just parts of a chapter. The index is a quick and easy reference for arising questions.

The systematic structure of this manual is supported by graphics. The key functions are depicted as icons in the left column in addition to the text descriptions. Every menu and function choice is also represented by a procedural schematic. The margin notes such as "Selection", "Procedure" or "Note" provide further orientation and clarity. Finally, we have provided a CNC Simulator screen shot for all work situations to be able to compare the text and the display directly. The graphics are usually preceded by a graphical schematic representing the menu structure.

As the number of features grows, it is only natural that the software descriptions become more extensive. The multitude of editing options might seem confusing at first sight. You need some time to get familiar with all the operations of your new hi-fi system as well, don't you. You will soon have a good idea on how to proceed with your CNC Simulator. And if in doubt, consult this manual about how to perform certain functions.

This about your manual! We would like to thank you for your confidence, and hope that this introduction to our CNC Simulators is of practical value. At the same time we would appreciate any comments you might have. Please send in your comments and proposals to MTS GmbH.

Figure 1.1-1:  
The minimum hardware required for the CNC Simulator workplace is a central processing unit, a monitor and a PC keyboard. A tablet with a CNC template can be used in addition.



#### Available Video-Chips

- ATI 18800/28800 ver 1
- ATI 18800/28800 ver 2+
- GENOA GVGA
- GENOA 5000series
- TSENG ET 3000
- TSENG ET 4000
- TRIDENT 8800
- TRIDENT 8900
- VESA Standard

## 1. System Overview

In the first two chapters you find a general overview of the system configuration with an introduction to the modified screen displays and to the new operating concept. If you are already familiar with the CNC Simulator, you can skip this introduction.

### 1.1 The Workplace

#### Hardware

The minimum hardware requirement for a single CNC Simulator workplace is:

- a personal computer with a hard disk and diskette drive,
- a monitor,
- a PC keyboard.

This can be supplemented by the following equipment:

- a CNC keyboard,
- a tablet,
- a printer, and other peripherals.

How to set up, connect and run this equipment is described in the manufacturer's documentation.

The efficiency of your PC depends mainly on its processor. Due to the rapid technical progress a wide range of computers are available. We therefore offer the CNC Simulator in one 32-bit version (386 or 486) with 4 MB of RAM.

#### Graphics adapter

To use the CNC Simulator you will need a graphics adapter card equipped with one of the following chipsets:

- TSENG ET 3000 or TSENG ET 4000
- ATI 18800 or ATI 28800
- TRIDENT 8800 or TRIDENT 8900

Before you start the CNC Simulator, check the documentation of the graphics adapter to see which chipset the card uses.

If your graphics adapter has a different chipset and problems occur when starting the CNC Simulator, you may be able to work out the problem as follows; so do not replace the graphics adapter yet:

1. You change the applicable settings in the configuration, so as to suit your video card (see Configuration Instructions)
2. At the DOS prompt, in the main directory containing the CNC Simulator, you rename the files "VERTEIL.KN\$" and "KONFIG.KN\$" to "VERTEIL.KNF" and "KONFIG.KNF" and start the CNC Simulator again. The MTS programs can be run in VGA mode in general.

#### Graphics Standard

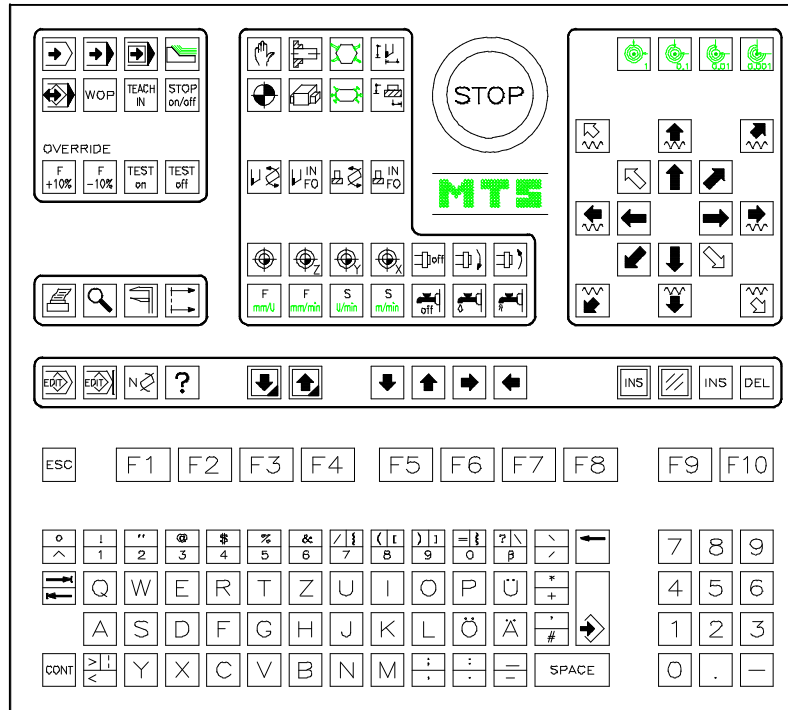
The following video modes are supported:

- YVGA (1024 x 768 pixels)
- SVGA (800 x 600 pixels)
- VGA (640 x 480 pixels)

#### Note

The graphics mode is selected in the configuration procedure (see configuration instructions).

Figure 1.1-2:  
 Template for the tablet;  
 as an alternative to the PC keyboard, the CNC Simulator can also be operated from a CNC keyboard or tablet.  
 In this case the tablet is equipped with a CNC template.



**Ports**

Transition points between the devices of a system for exchanging data or signals are called "interfaces". In other words, these are the connection points for peripheral devices such as printers, monitors and keyboards - commonly called "ports". The number and type of ports depends on the peripherals you want to use. Please refer to the manufacturer's notes and the MTS Configuration Instructions.

**Operating System**

Version 5 of the CNC Simulator was designed to run on personal computers using the MS-DOS **operating system** (version 3.3 or higher).

The CNC Simulator can also be installed in a Novell **network**.

Figure 1.2-1:  
CNC Turning,  
Main menu;  
Function keys with text  
notes for the process-  
ing options available.

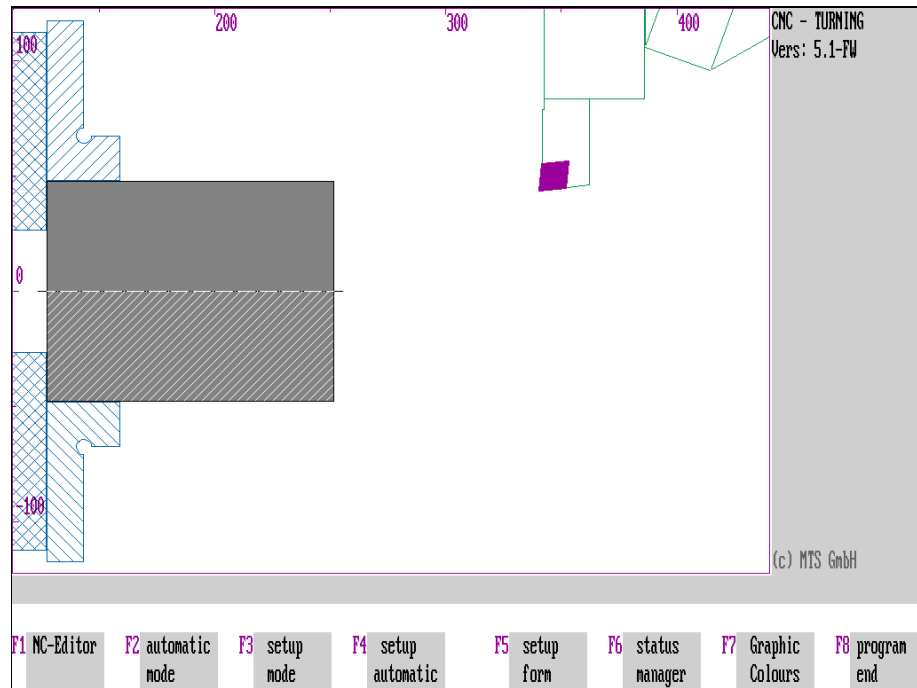
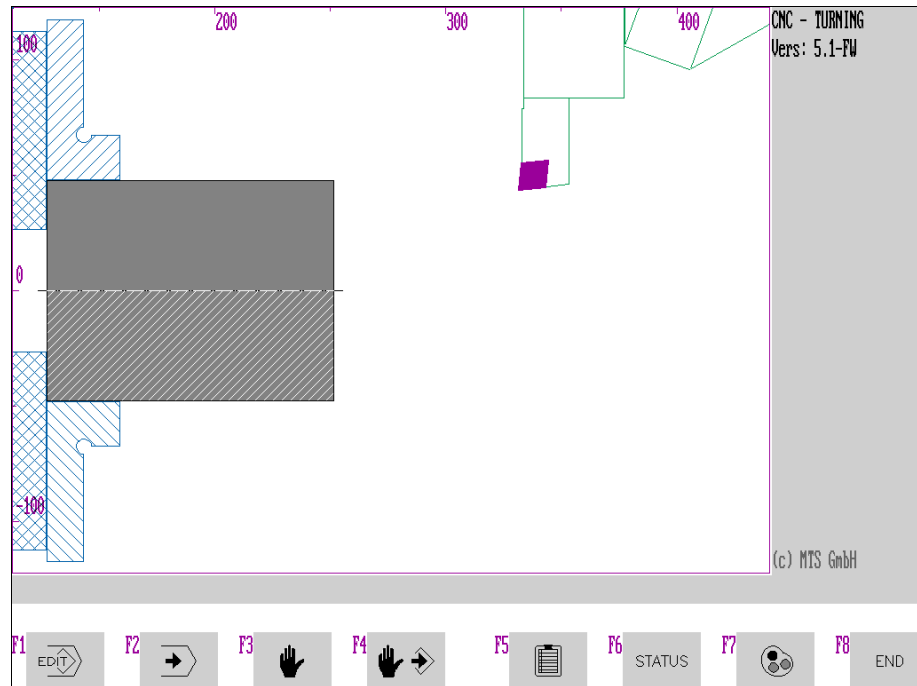


Figure 1.2-2:  
CNC Turning,  
Main Menu;  
as an alternative to  
text labels on the  
function keys, CNC  
symbols and other  
icons can be dis-  
played.



## 1.2 Input Options

The CNC Simulator can be used with various different input media. These can, of course, also be used parallel to each other (see [Configuration Instructions](#)).

### PC Keyboard

A PC keyboard is basically all you needed to use the CNC Simulator. In version 5 of the CNC Simulator you select all program functions with the **function keys** and enter machine commands and NC program blocks as sequences of digits and letters.

### CNC Symbols

The function keys displayed on the screen are usually labelled with a short text indicating the subsequent editing steps. To imitate with the CNC Simulator the handling of an NC machine tool, **CNC symbols** can be displayed instead of the text labels. This option is recommended, in particular, for CNC training; the CNC symbols being part of the curriculum.



There are some special menus, for example the tool and clamping fixture management menus, in which the function keys remain labelled with text. The appendix provides a summary of the CNC symbols and other icons.

### CNC Keyboard

Instead of the PC keyboard a CNC keyboard can be used. It is a non-proprietary design with 7 key blocks configured for NC purposes. The key blocks are grouped as follows:

- Editor / Programming Mode,
- Automatic Mode,
- Setup Mode / Manual Mode,
- Character Inputs,
- Digit Inputs,
- Multipurpose keys: Input / Resume / Cancel,
- Travel keys.

Similar to the CNC template for the tablet, standardized symbols for operating an NC machine tool are assigned to the upper three key blocks.

### Tablet

If the INCAD CAD/CAM system and the CNC Simulator are used at the same time, the PC needs to be equipped with a mouse or a tablet. A **template** can be placed over the tablet (see [figure 1.1-2](#)) to provide a CNC **keyboard** configuration.

The template has a total of 173 input keys including a reduced typewriter set and the keys for NC editing arranged in functional groups as follows:

- Automatic Mode,
- Setup Mode,
- Programming Mode Functions,
- Supplementary Functions,
- Travel Keys.

Even when using the tablet you retain the advantages of the dialogue-oriented user guidance of the CNC Simulator as well as those of operation by CNC symbols.

The CNC keyboard or tablet can easily be connected with the serial port of the personal computer.

Figure 1.3-1:  
CNC Turning;  
Schematic of the ma-  
chine configuration,  
postaxial machining.

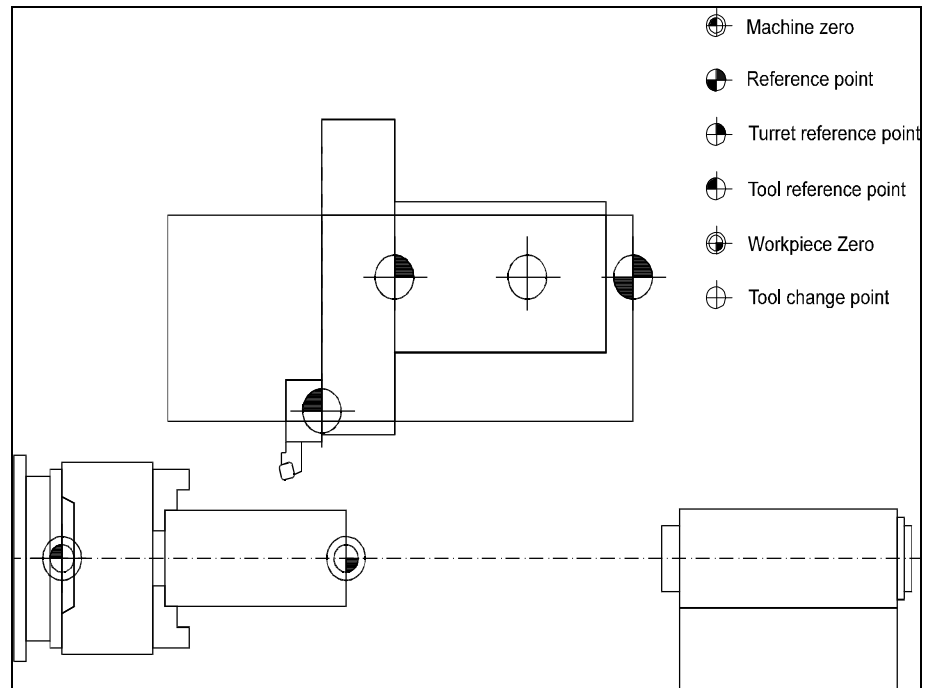
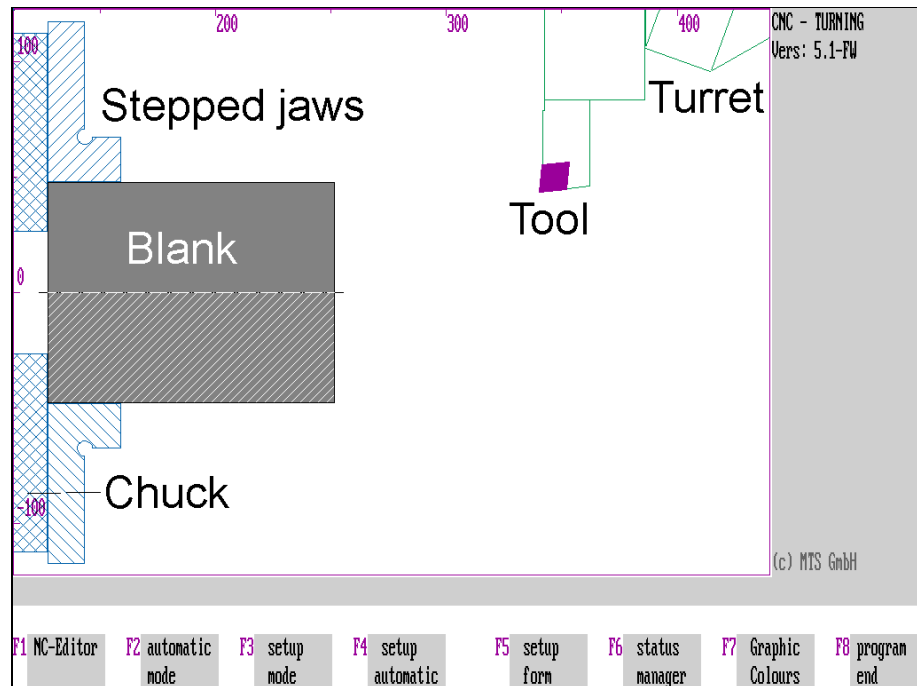


Figure 1.3-2:  
CNC Turning,  
Main Menu;  
The machine ele-  
ments: spindle, chuck,  
stepped jaws or cen-  
ters, turret, tool, tail-  
stock, sleeve etc. are  
also represented in the  
CNC simulation.





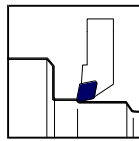
## 1.3 System Functions

Before we deal with the handling of the individual program sections of the CNC Simulator, we will first introduce the system as a whole. If you are already familiar with the CNC Simulator, you can skip this chapter.

The simulation of the machining process with CNC Simulators closely follows actual work procedures on a machine tool and includes all the functional steps of the machine tool and the control system operation. For this reason the CNC Turning and Milling simulators can be adapted to all common machine tool types and their equipment:

### CNC Lathe

#### Machine Type



The CNC Turning Simulator is a single-spindle machine with 2-axis contouring control.

It can also be equipped with driven tools.

The spindle, chuck, clamping fixtures, turret, tools and tailstock can be specified according to the equipment on the actual machine tool. The user can also select the travel range, the maximum and minimum speed and feedrate, the settings for preaxial and postaxial machining and many other things.

#### Clamping Fixtures

The CNC Simulator allows you to machine with the workpart clamped between centers or in a stepped-jaw chuck. Various (definable) chucks, stepped jaws, lathe centers and face drivers are available for this purpose.

It is also possible to select a configurable tailstock which can be equipped with a choice of lathe centers.

#### Tool Carrier

The CNC Simulator can be equipped with various turret heads, subgrouped in three different turret types:

- Hexagon Turret,
- Disk Turret and
- Crown Turret.

Each turret has up to 16 tool positions.

#### Tools

New in version 5 is a separate tool management system for specifying toolholders, cutting edges or reversible carbide tips [to DIN]. The following lathe tool types are available in the standard version:

- side and external cutting tools,
- internal cutting tools,
- copying tools,
- groove recessing tools,
- threading tools,
- twist, center and reversible tip drills.

#### Control

The MTS programming code is manufacturer-neutral and complies with DIN 66025. It contains all common extensions such as segment contours, cycles, tool nose compensation, subroutine and parameter programming.

Figure 1.3-3:  
CNC Milling;  
Schematic of the ma-  
chine configuration.

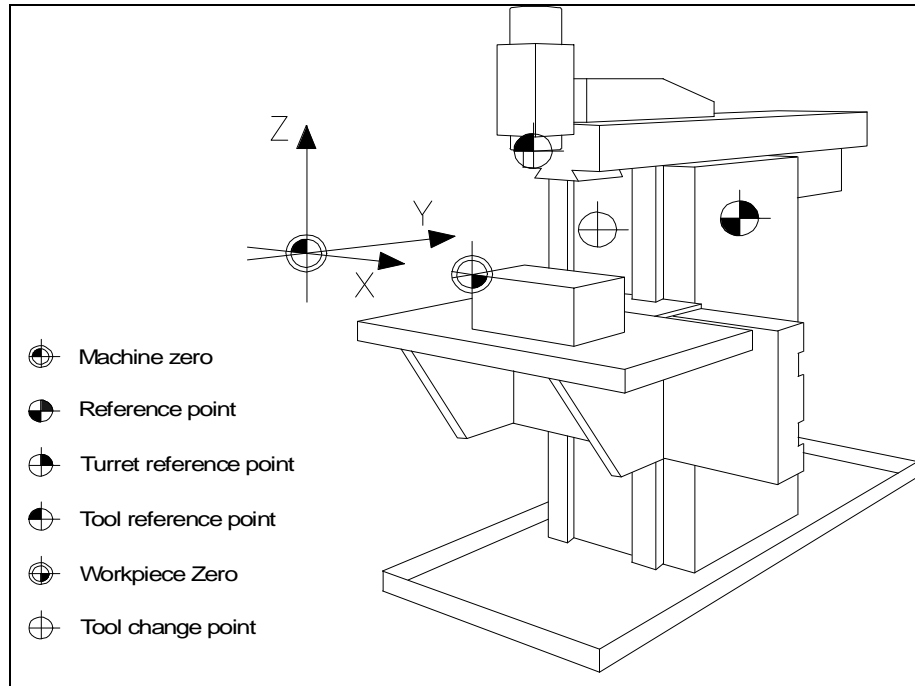
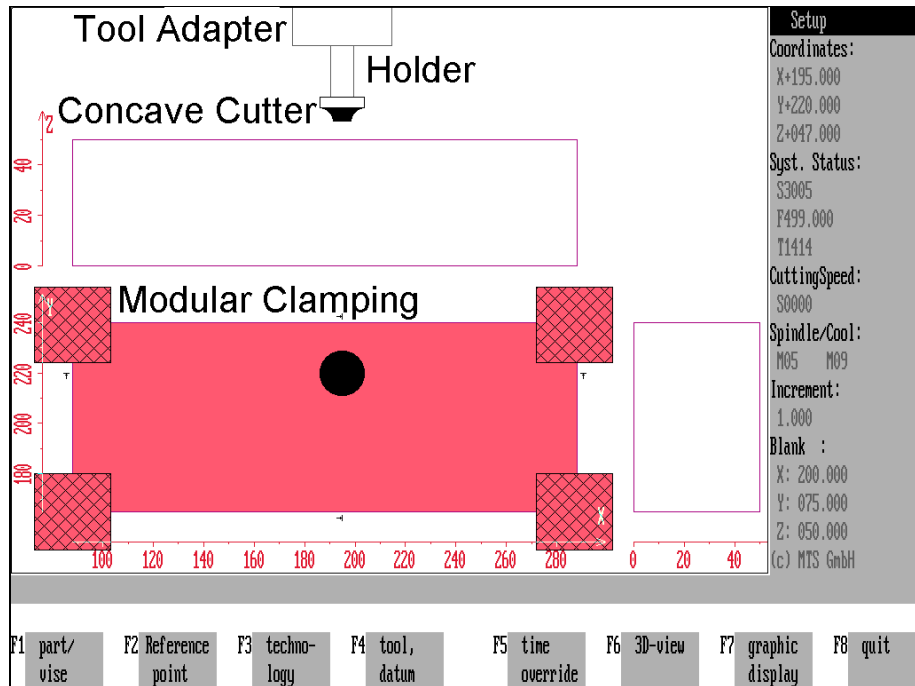
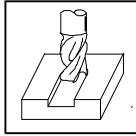


Figure 1.3-4:  
CNC Milling,  
Setup Mode;  
As with turning, the  
CNC Milling Simulator  
provides the following  
machine elements:  
clamping fixtures,  
spindle, magazine, tool  
adapter, tools and  
many more.



## CNC Milling Machine

### Machine Type



The CNC Milling Simulator simulates a 3-axis milling machine with vertical or horizontal spindle position.

In the CNC simulation no distinction is made between knee type and plano-milling machines or machining centres; instead, all positioning and feed movements appear to be made by the tool carrier.

To simulate different machine types, however, the machining planes can be selected from G17, G18 and G19:

Plane Selection	long.	lat.	vert.
<b>G17</b>	X	Y	Z
<b>G18</b>	X	Z	-Y
<b>G19</b>	Z	Y	-X

### Chuck

The workpart can be clamped by using:

- jaws,
- magnetic plate,
- modular clamping.

### Magazine

The magazine holds up to 99 tool positions (pockets) in which the tools are inserted from the tool manager.

### Tools

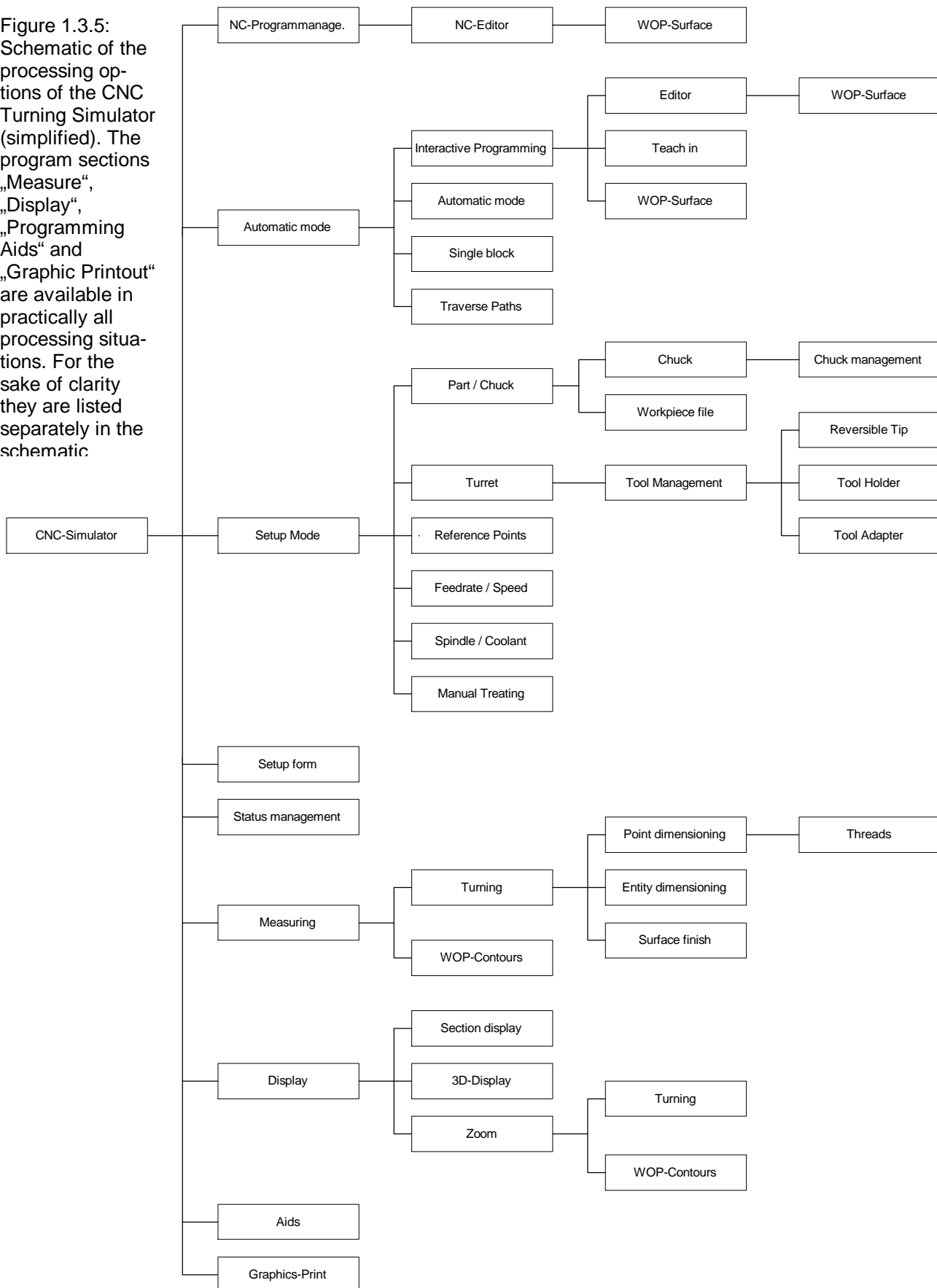
The Tool Manager has been fully redesigned for version 5, so that tool adapters and tools can be specified in compliance with DIN. The following tool types are available in the standard version:

- (shank-type) end mills,
- shell end mills, boring slot cutters, face end mills,
- (disc) side cutters,
- T-slot cutters,
- angular milling cutters (type A and B), concave milling cutters, radius -milling cutters,
- reamers,
- countersinks,
- drills, reversible carbide tip drills, step drills and taps.

### Control

The MTS programming code is manufacturer-neutral and complies with DIN 66025. It contains all common extensions such as segment contours, cycles, tool cutter radius compensation, subroutine and parameter programming.

Figure 1.3.5: Schematic of the processing options of the CNC Turning Simulator (simplified). The program sections „Measure“, „Display“, „Programming Aids“ and „Graphic Printout“ are available in practically all processing situations. For the sake of clarity they are listed separately in the schematic.



## Collision Monitoring

Reality-oriented simulation of machining processes is based on the fact that the CNC Simulators function like the actual machine tools in the workshop :  
During workpart machining, **collision monitoring** is performed and the results are displayed as error messages. The entire machine tool space with workpart, fixtures, tool system etc. is taken into account. During machining the internally followed mathematical model computes the **actually resulting workpart contour** using the programmed tool paths during program execution, with a tolerance range of less than 1  $\mu\text{m}$ . As the simulation can be performed for different tool qualities and materials etc., the error and collision monitoring function does not check the programmed feedrate or speed nor does it use coolant.

## Operating Modes

The following three modes can be used for processing:

### Setup Mode

**Setup Mode** follows the corresponding steps on the machine tool. In this mode, all the necessary preparatory operations are performed, such as mounting the blank, choosing the clamping fixture, choosing the tool, entering the tool offset values, specifying the machine zero, referencing the machine etc. (see Figure). Of course, Setup Mode also allows manual machining just as the workshop machine tool does.

### Automatic Mode

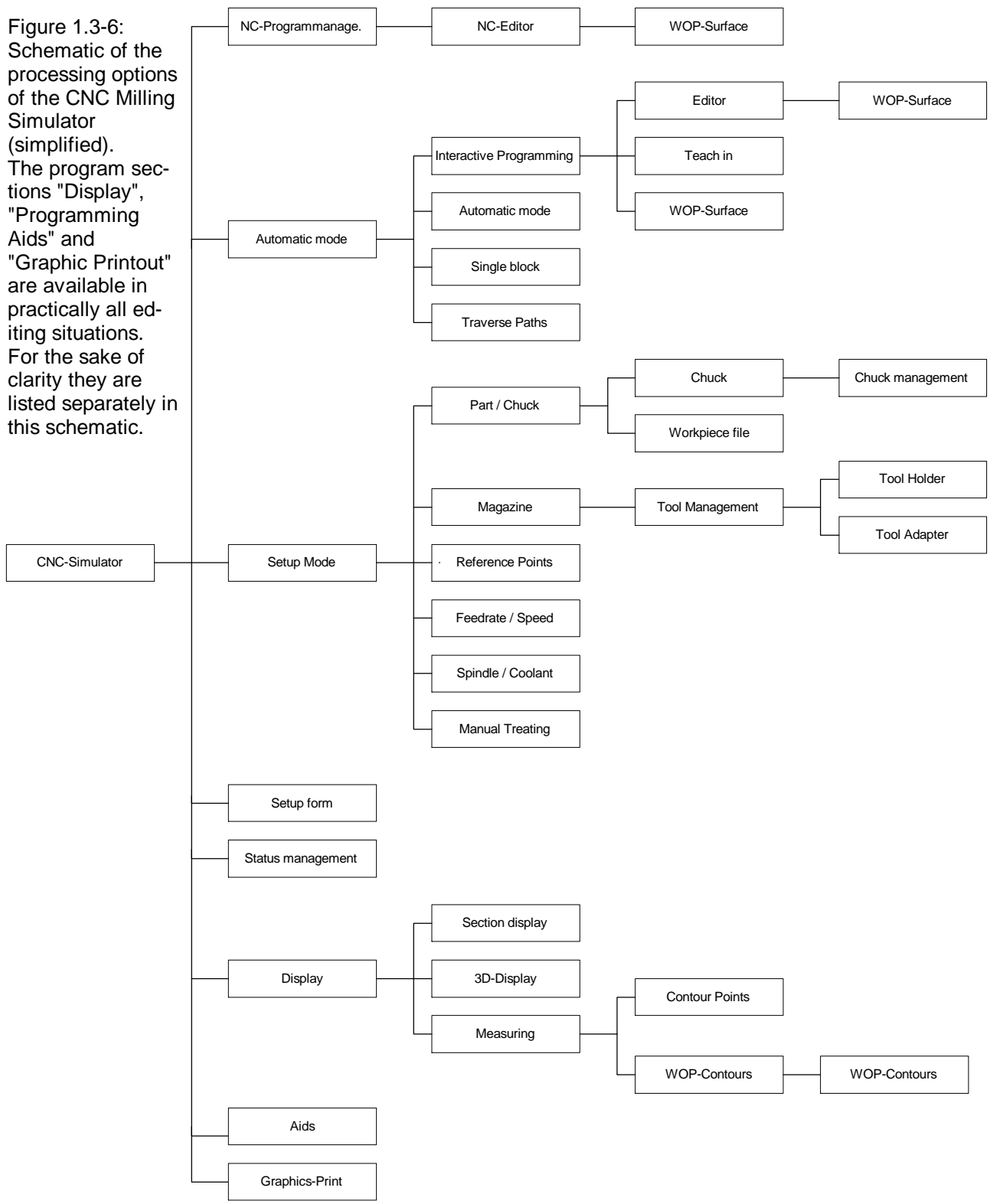
In **Automatic Mode**, NC programs are executed and tested in real-time simulation. The machining of the workpart is displayed dynamically in real-time with continuously actualized coordinates. At the same time the current NC blocks are displayed and the machining and feed times are computed. Collision monitoring ensures that program errors can be quickly recognized and corrected.

### Programming

**Programming Mode** provides four ways of generating an NC program:

- An **Editor** is available for direct input of NC blocks. It has a special programming interface for NC blocks and checks the syntax (the formal structure of the NC block) as the block is being entered.
- **Interactive Programming** is a feature in which the Automatic Mode and Editor complement each other to provide the simplest and most efficient way to get started with NC programming. The simulation follows step by step the creation of an NC program, including collision monitoring.
- A special form of interactive programming is the **Teach-In** function. As in Setup Mode, the workpart is machined manually and the corresponding simple travel commands complying with DIN 66025 are generated and automatically inserted in the NC program.
- To make it easy to program even complex contours, the editor and the interactive programming function have a dialogue-driven **WOP Interface**. The inputs are supported by a user guidance system with explanatory graphics

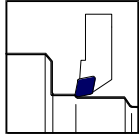
Figure 1.3-6: Schematic of the processing options of the CNC Milling Simulator (simplified). The program sections "Display", "Programming Aids" and "Graphic Printout" are available in practically all editing situations. For the sake of clarity they are listed separately in this schematic.



- (-) The dialogue provides only those input options the programming code would permit in the given situation. The contours created with the WOP Interface can also be measured and zoomed.

Measure/Roughness

CNC Turning



The CNC Turning Simulator provides "**Measure**" and "**Roughness Depth Gauging**" ("Surface Peak-to-Valley Height") functions with which the **workpart quality** can be tested before production. These functions can be activated any time, even during workpart machining or NC programming.

A further new feature in version 5 is the **recording and display of thread geometry**. This function lets you test thread pitch, depth, root and nominal diameter and the right choice of tool.

An additional aid is the **Zoom** function, with which a section of the lathe workpart or machining process can be enlarged or reduced any time.

## Special Functions

Finally, the CNC Simulator incorporates some special functions which effectively support processing and NC programming:

3D View

A 3-dimensional view of the workpart can be displayed during turning or milling. The workpart can also be "cut open" to reveal internal machining.

Programming Aids

A help function is available in all editing situations. It is supported by illustrative graphics and provides basic information on NC programming while explaining individual commands in detail. When selected from a programming mode it allows you to edit the current NC program directly. In this manner, almost the entire programming instruction manual can be consulted during editing.

Printout

To document the current editing progress, NC programs and screen displays can be **printed**. These printouts give you a means of checking and filing the results of your work.

## Data Management

The internal **data management** functions provide a convenient means of documenting and backing up all work results without previous knowledge of the operating system. These functions include:

- NC Program Manager;
- Tool Manager;
- Clamping Fixture Manager;
- Saving created workparts;
- Saving current editing progress;
- Generating various setup sheets;
- Managing configuration files.





## 2. Introduction to Operation

Before we explain the functional operations such as setting up the machine tool, creating and testing the NC program in Automatic Mode etc., we first want to discuss the screen display and menu operations in general. After that we can start to work.

### 2.1 Installation and Configuration

Before starting to work with the CNC Simulator you need to install the software on your PC in a certain directory. You then need to modify the configuration to adjust the default settings of the machine tool, CNC control system and hardware to meet your specific working environment. This initial work prepares the CNC Simulator ready for operation.

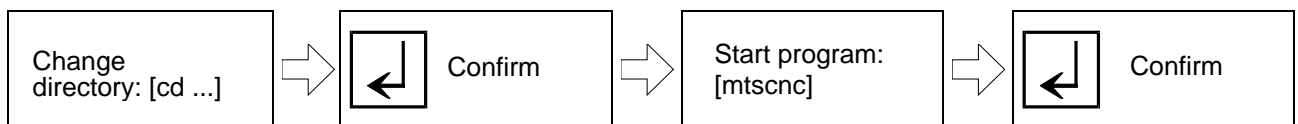


The installation and configuration procedures are supported by special utility programs. Separate instructions are available for these programs, and need to be studied before you proceed here.

### 2.2 System Start for MTS Programs

When you switch on your PC, the MS-DOS operating system prompt indicates the current drive. To run the MTS software, first change to the drive and directory where the MTS programs are stored. Then run the CNC Simulator by entering the command "MTSCNC":

#### Input:



#### First Example:

The current directory is given as "C:\>\_" and the CNC Simulator is in the directory "C:\MTSCNC" as proposed by the installation program:

First enter the name of the directory and confirm by pressing ; the prompt changes to indicate the new directory:

```
C:\CD MTSCNC →  → C:\MTSCNC>_
```

Now enter the command "MTSCNC" and again confirm with ; the MTS program will now load:

```
C:\MTSCNC\MTSCNC →  → [MTS program]
```

Figure 2.2-1:  
Example of the DOS  
commands for starting  
the CNC Simulator.

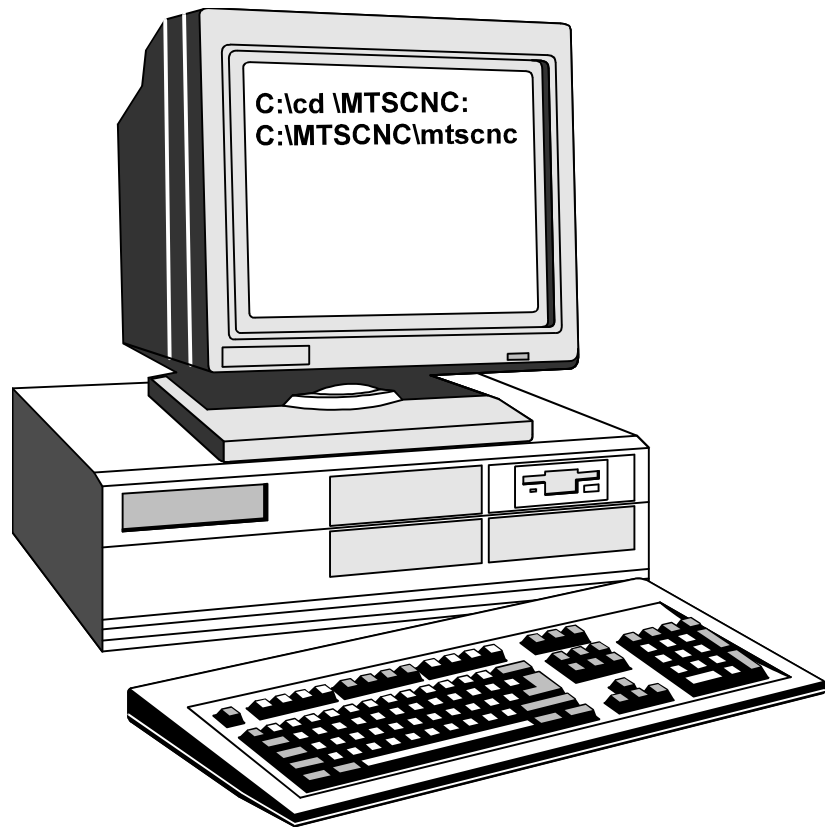


Figure 2.2-2:  
Menu of MTS pro-  
grams;  
The individual pro-  
grams are started us-  
ing the function keys:  
e.g. CNC Turning by  
pressing the "F1" key  
etc.

(C) Mathematisch Technische Software-Entwicklung GmbH			
	Machine	Control	
F1 CNC-Turning	MTS-Drehmaschine	MTS-CNCT	
F2 CNC-Milling	MTS-Fräsmaschine	MTS-CNCM	
F3 INCAD			
F4 Transmission program			
F5 Configuration			
F6 Menu Turning			
F7 Menu Milling			
F8 END			


  

Selection Machine (F6), Control (F7)							
1 Turning	2 Milling	3	4	5 Configu- ration	6 Menu Turning	7 Menu Milling	8 END

**Second Example:**


The current directory is "C:\SYS>\_" and the CNC Simulator is in the directory "D:\MTS386":

First change to hard disk partition "D:":

C:\SYS>D: →  → D:\>\_

Then enter the subdirectory ".\MTS386":

D:\>CD MTS386 →  → D:\MTS386>\_

Now, as above, enter the command "MTSCNC" and confirm by pressing ; the MTS program will now load:

D:\MTS386>MTSCNC →  → [MTS program]

**Cross Reference**

If you are not familiar with the commands of the operating system, please consult your MS-DOS manual. The appendix of the Installation Instructions also includes a brief introduction to the operating system.



The launching procedure described above can be simplified using an appropriate **batch file**.

**Starting CNC Simulator**

Once you have launched the program, the menu ([see figure](#)) appears with the choice of the MTS programs available in your system. The highlighted rectangular boxes under "Machine" and "Control" indicate the currently selected configuration files. Select the desired program by pressing a function key:

**F1**

or

**F2**

**Turning/Milling:** the **F1** and **F2** function keys launch the CNC Turning and Milling Simulators, respectively (if they are installed).

**F3**

**[INCAD]:** the **F3** function key launches the INCAD NC/CAD system if it is installed.

**F4**

**[Transfer program]:** **F4** starts the Transfer program if it is installed.

**F5**

**Configuration:** the **F5** key selects the Configuration program to specify default settings.

**F6**

or

**F7**

**Turning/Milling Menu:** these menus let you select the current files for machine and control configuration. They also allow you to start the postprocessors, the Format Conversion Program and the PAL Marking Program.

**F8**

The **F8** key returns you to MS-DOS.



**Try out these operations and run one of the CNC Simulators !**

Figure 2.2-3:  
CNC Turning,  
Menu of utilities and  
machine control and  
configuration options.

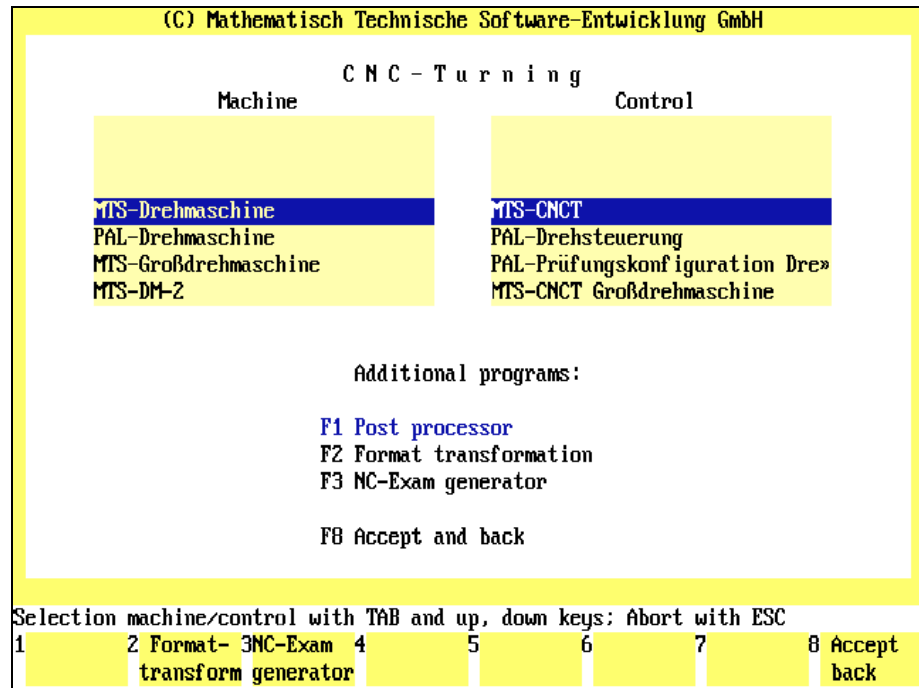
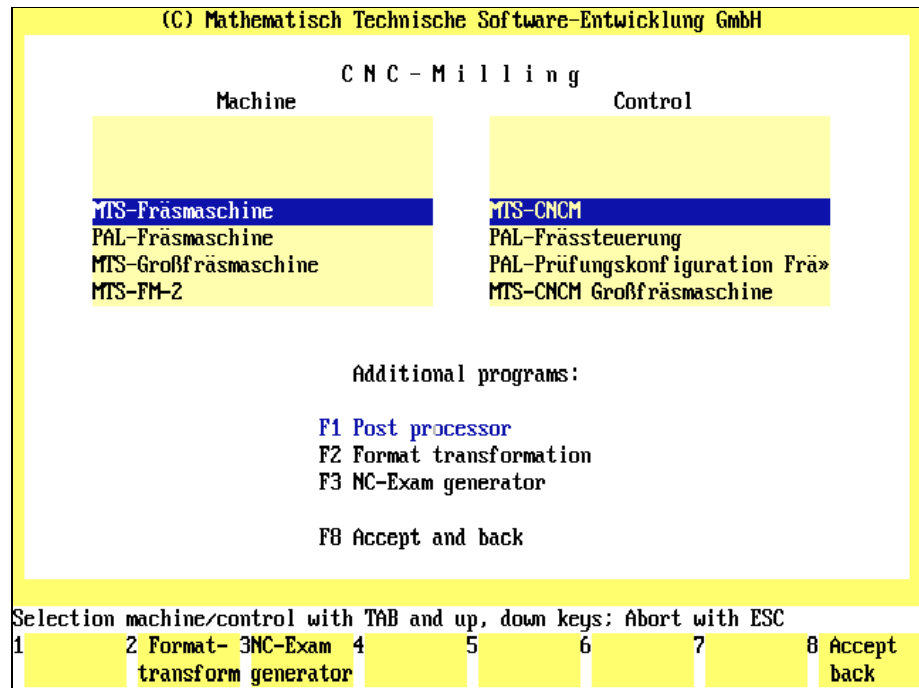


Figure 2.2-4:  
CNC Milling,  
Menu of utilities and  
machine control and  
configuration options.




## Turning and Milling Menu



From this menu you can run certain utilities and select the files for the current machine and control configuration. Only those utilities installed on your system are displayed on the function keys.



or



The  key activates either the list of machine configuration files or the list of control configuration files.

Use the  or  cursor keys to mark the desired configuration file you want to select as the current file.

Press the  key to confirm your selections and to return to the starting point.

Press the  key to cancel the selections and to return to the starting point.

Besides the functions described above, this menu also allows you to run the following utilities:



**[Postprocessor]:** the postprocessor translates an NC program created with the MTS programming code into a DIN version of a specific control system and transfers it to the CNC control system (see also the Postprocessor Manual).

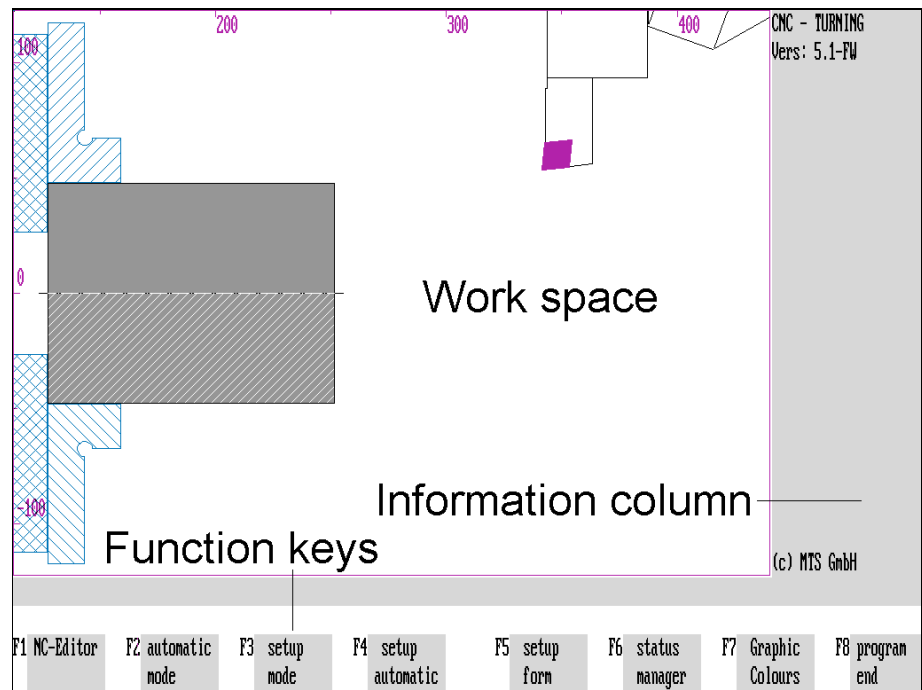


**Format Conversion:** to edit or test an NC program that was created with an older version of the CNC Simulator or with the INCAD NC/CAD system or another system, the data format has to be adapted to the new MTS standard. These programs cannot be loaded into the NC Editor or CNC Simulator until they have been converted ([see also chapter 8](#)).



**[PAL Marking Program]:** to modify an existing NC program in compliance with the PAL guidelines for CNC training and create the so-called "gap texts", run the PAL Marking program ([see also the PAL turning/milling manual](#)).

Figure 2.3-1:  
CNC Turning,  
Main Menu



## 2.3 Screen Display

When you launch a CNC Simulator, its **Main Menu** appears on the screen. It is the top menu level, from which all other processing functions can be chosen.

In the new version the screen display of the Turning Simulator has been converted to **vector graphics**. You can soon notice the difference between this and the CNC Simulator 4.1: text and picture build-up on the screen is quick and accurate.

### Screen Layout

After starting your CNC Simulator you can see the second improvement: the **screen layout**. The screen representation of CNC machining is in general divided into three areas, with workpart machining displayed graphically and dynamically in the work space window and the necessary text information in the "Information Column" beside it. It contains the information you need in your current work situation. The display representation of each program section is called its **interface**. If the screen also contains information (in the form of text or symbols) on subsequent machining steps (option choices) it is called **menu**.



One of our principles was to try to standardize the displays and menu structures for CNC turning and CNC milling technologies so that each operation essentially follows the same scheme. Therefore, if you are familiar with one CNC Simulator, you are automatically able to use the other.

The main menu illustrates the above-mentioned changes in the screen layout:

### Screen Windows

1. The **left screen window** shows a section of the work space with the blank or workpart, the clamping fixture and, depending on its position, the tool. This applies to the Main Menu and to Setup and Automatic Mode. In other operating functions this screen window always contains a graphical representation of the current work situation.

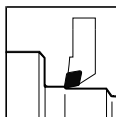
### Information Column

2. The **right column** contains text information on certain machining situations. In the Main Menu none of the modes is active, therefore, no information is shown. In Setup Mode and Automatic Mode this column contains information on the current machine and system status. During measuring, for instance, it contains dimensions etc.

### Menu Functions

3. The numbered boxes at the bottom of the screen indicate the program functions that can be selected with the **function keys** during processing. The two lines above the function keys are reserved for the **program dialogue**. After starting an NC program the current NC blocks are shown on the upper line. The bottom line contains error messages.

### CNC Turning



The CNC Turning Simulator simulates a single-slide machine tool. The **work space** shows a section of the Z/X axis, including the turret, tool, fixture and possibly the tailstock and the blank or workpart.

Figure 2.3-2:  
CNC Milling,  
Main Menu

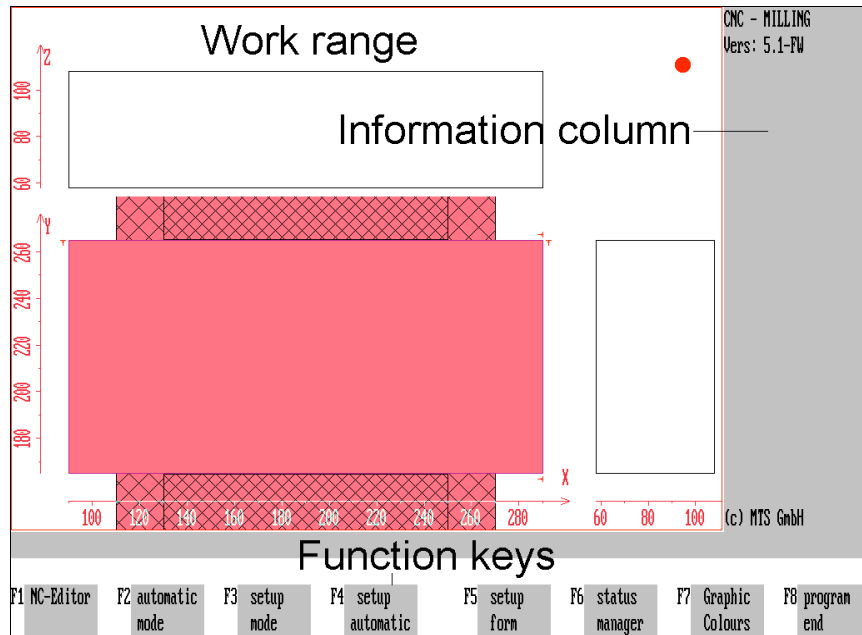


Figure 2.3-3:  
CNC Turning,  
Menu: Graphical Display;  
A half-section view of  
the workpart has been  
selected.

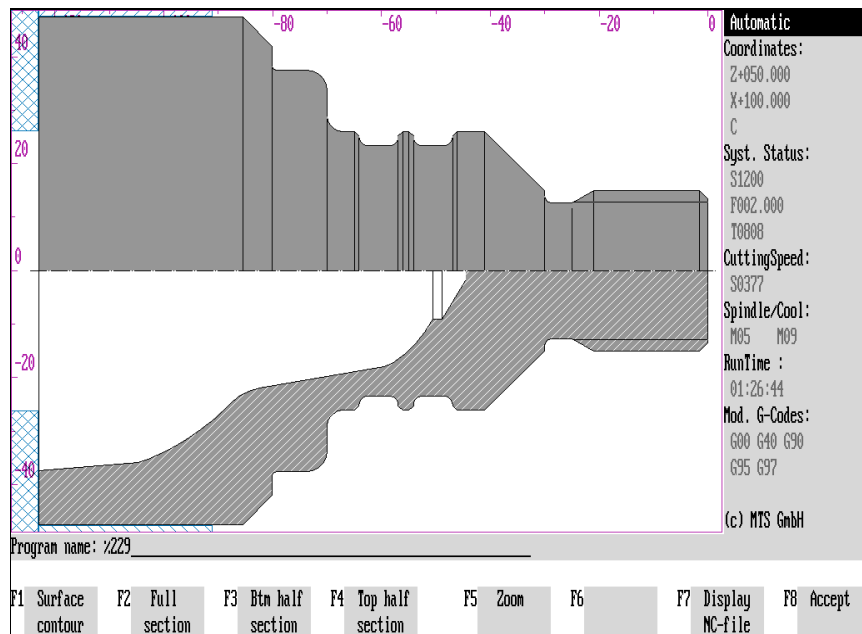
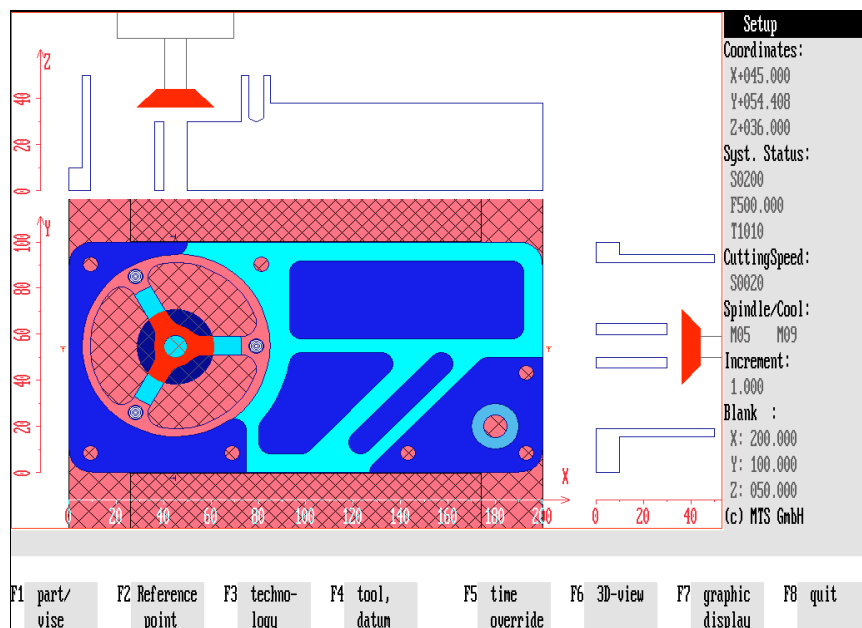
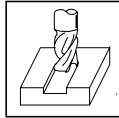


Figure 2.3-4:  
CNC Milling,  
Menu: Graphical Display;  
In addition to the top  
view, side views and a  
system of coordinates  
can also be displayed.





**CNC Milling**

The CNC milling Simulator simulates a 3D milling machine with vertical or horizontal spindle position. The **work space** shows a top view of the workpart; side views and an axis cross can also be displayed.

To display the machining processes of milled workparts with extreme size differences as well, the display size adjusts itself automatically

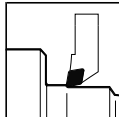
## Modifying the Graphical Display

Besides the display types described above, the CNC Turning and Milling Simulators provide various ways of modifying the screen display to improve the representation of machining processes.

### Section Display

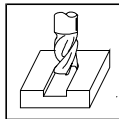
In the CNC Turning Simulator the changes often involve various **views of the workpart**, while in the CNC Milling Simulator the top view is complemented by optional **dynamic graphical section views** according to the working position of the tool in X/Z and Y/Z direction, and of the **axes**.

### Zooming

**CNC Turning**

The Turning Simulator also provides continuous in and out zooming of the screen section (up to the full work space of the machine tool during machining).

This zoom function is also available during measuring and Workshop-Oriented Programming (WOP).

**CNC Milling**

Unlike the Turning Simulator, **only WOP-programmed contours can be zoomed during milling**, and the view of the entire machine tool work space is available only as a special function in setup mode when choosing the clamping fixture

### Color Display

In both CNC Simulators you can confirm the **colors** of your choice for the screen elements (within the limits of your graphics adapter).

**Cross Reference**

The zoom function and the screen display modification options are discussed in detail in chapters [9](#) and [10](#).

Figure 2.4-1:  
CNC Turning,  
Menu sequence to define a cylinder as a blank:

from Setup Mode, first select the "Workpart /Fixture" function with [F1]. In the subsequent menu choose "New Workpart", again with the [F1] key, then select the "Cylinder" function.

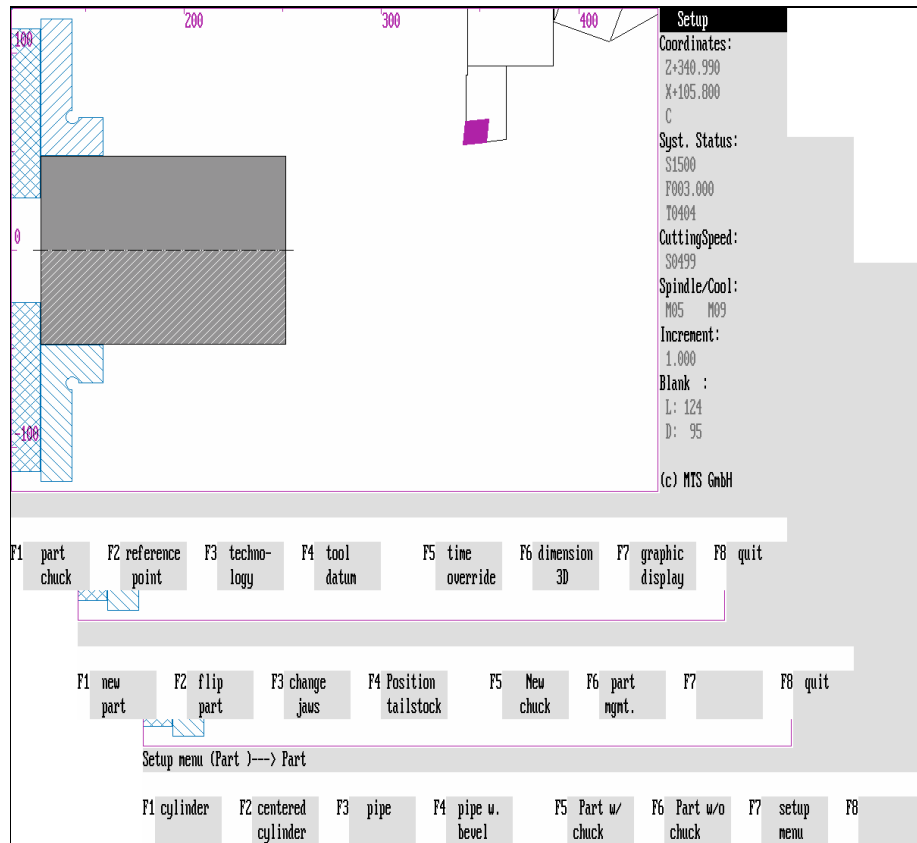
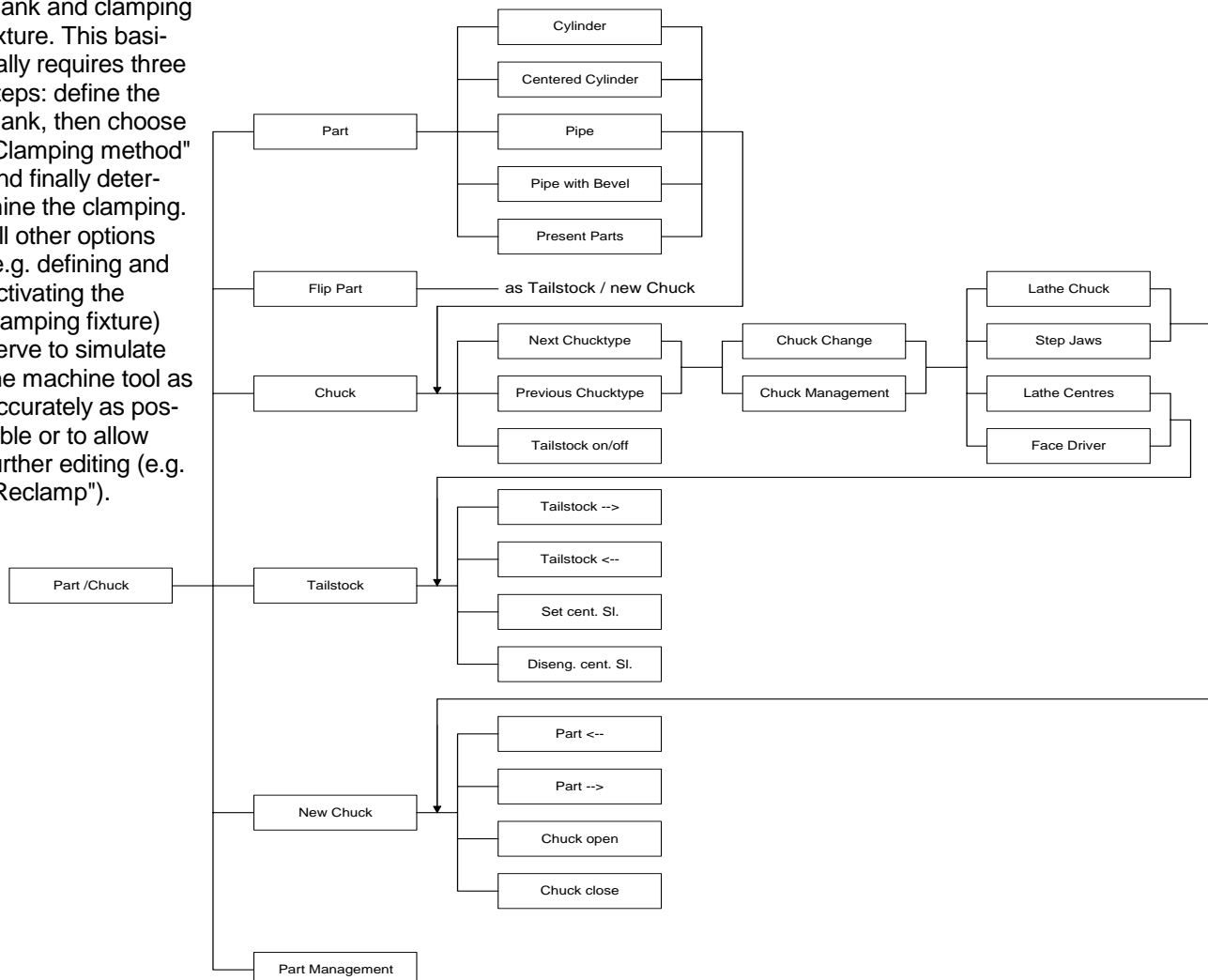


Figure 2.4-2:  
CNC Turning,  
Schematic of the sequence of steps for defining the blank and clamping fixture. This basically requires three steps: define the blank, then choose "Clamping method" and finally determine the clamping. All other options (e.g. defining and activating the clamping fixture) serve to simulate the machine tool as accurately as possible or to allow further editing (e.g. "Reclamp").



## 2.4 Menu Functions

When starting the system you have already taken the first steps in program execution. When using the CNC Simulator, a general distinction is made between **program control** inputs and those for certain values or texts, called **editing inputs**.

### WOP Operating Concept

All program control options of the CNC Simulator are given as context-sensitive **menu items**, with important procedural steps supported by **program dialogues**. This new menu concept of the CNC Simulator is based on the **WOP operating concept** ("Workshop-Oriented Programming" - cf. Chapter 12). It enables you to carry out the operational steps without further assistance.

The only disadvantage of the WOP operating concept is, however, the fact that with increasing functionality the number of submenus correspondingly grows. But as the available options are shown in each work situation, the function keys at the bottom of the screen give you guidance all the time.

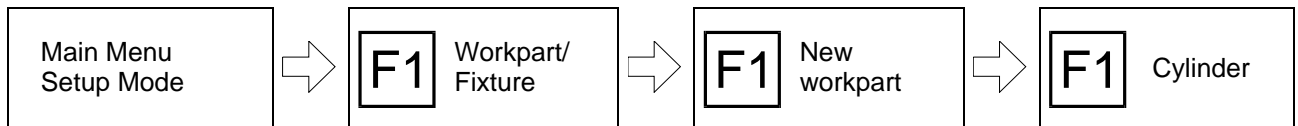
On the other hand you can benefit from this operating concept because you are able to make use of the operating functions without prior knowledge; as the work sequences are structured functionally most of the menus are self-explanatory (e.g. blank definition is followed by the selection of clamping fixtures and definition of the clamping method - see schematic).

### Example

The figure illustrates an example of an operation sequence:

From the Main Menu, select Setup Mode by pressing the **F3** function key. To use a cylinder as a blank, continue by pressing the **F1** function key three times, then enter the blank dimensions. Such work sequences are illustrated with the following scheme in this manual:

### Presentation of Program Control/Selection:



### CNC Symbols

It is important to know the CNC symbols when using a machine tool. We therefore recommend that during CNC training the CNC Simulator is configured to show the CNC symbols on the function keys instead of the text labels. The hardware setting "Show Icons: yes" (under "MTS SoftKonfg", see Configuration Instructions) displays the available work steps with **CNC symbols**: pressing the associated **function key** then activates the processing function. The numbers to the left of the square boxes refer to the function keys.

An example: from the main menu, to choose Setup Mode you press the CNC key



Now, try out the menu functions and operating steps! Don't worry about something going wrong - unlike on the machine tool, nothing can be damaged in the Simulator. Before you start programming make sure not to overwrite or delete an existing NC program.

Figure 2.2-3:  
CNC Milling,  
Menu sequence for  
testing an NC program:  
starting in Automatic  
Mode, first enter the  
program name and  
then choose the de-  
sired simulation type.

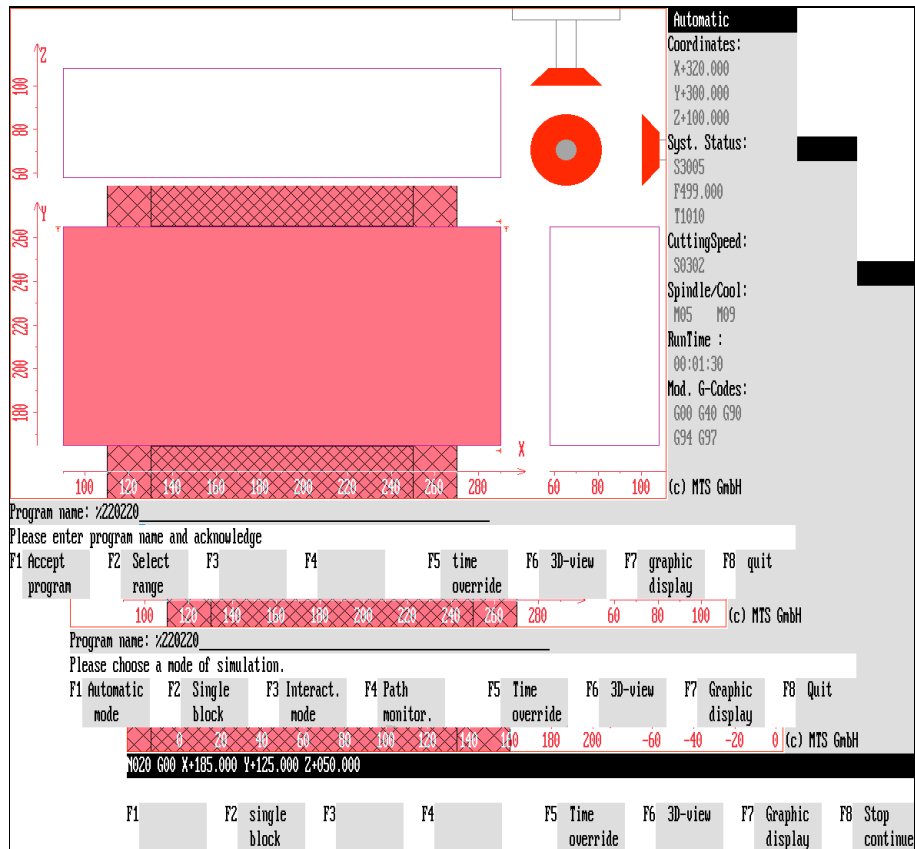
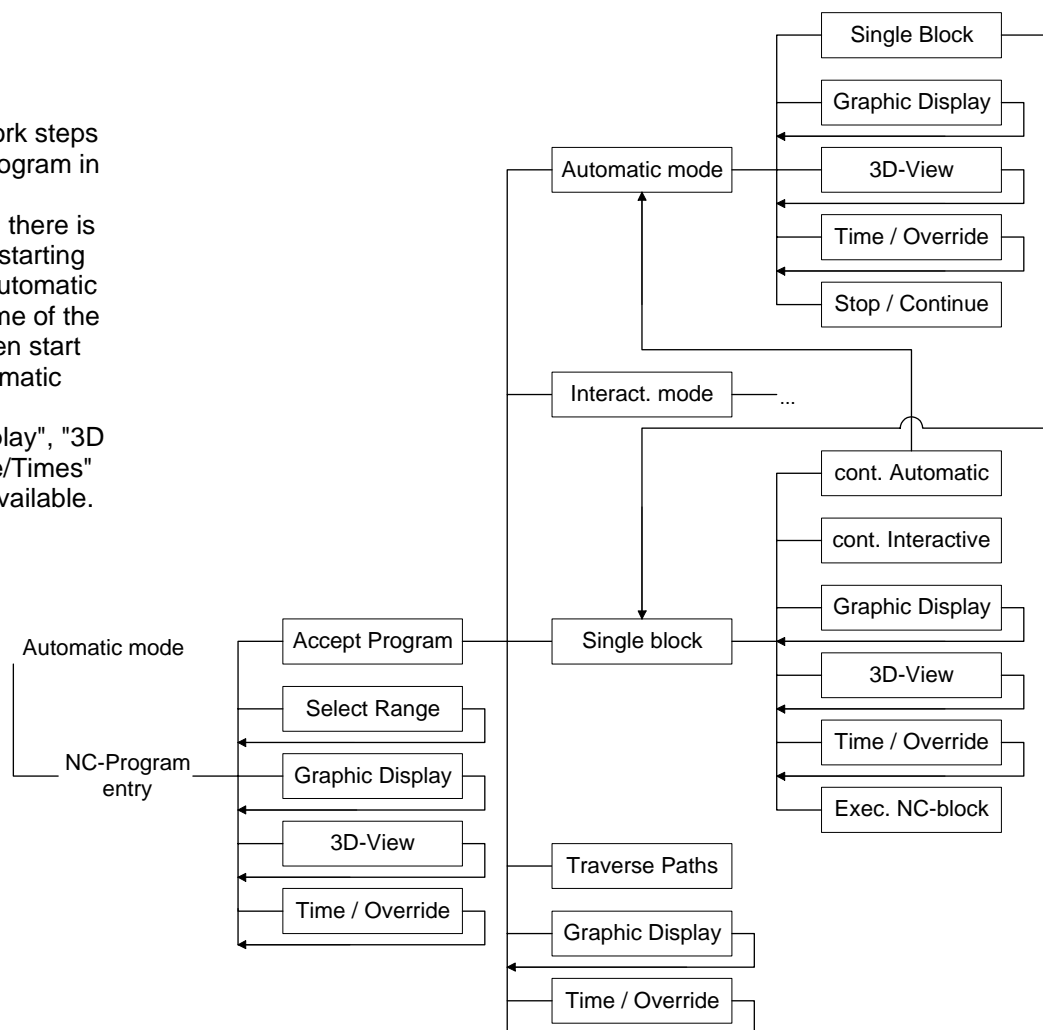


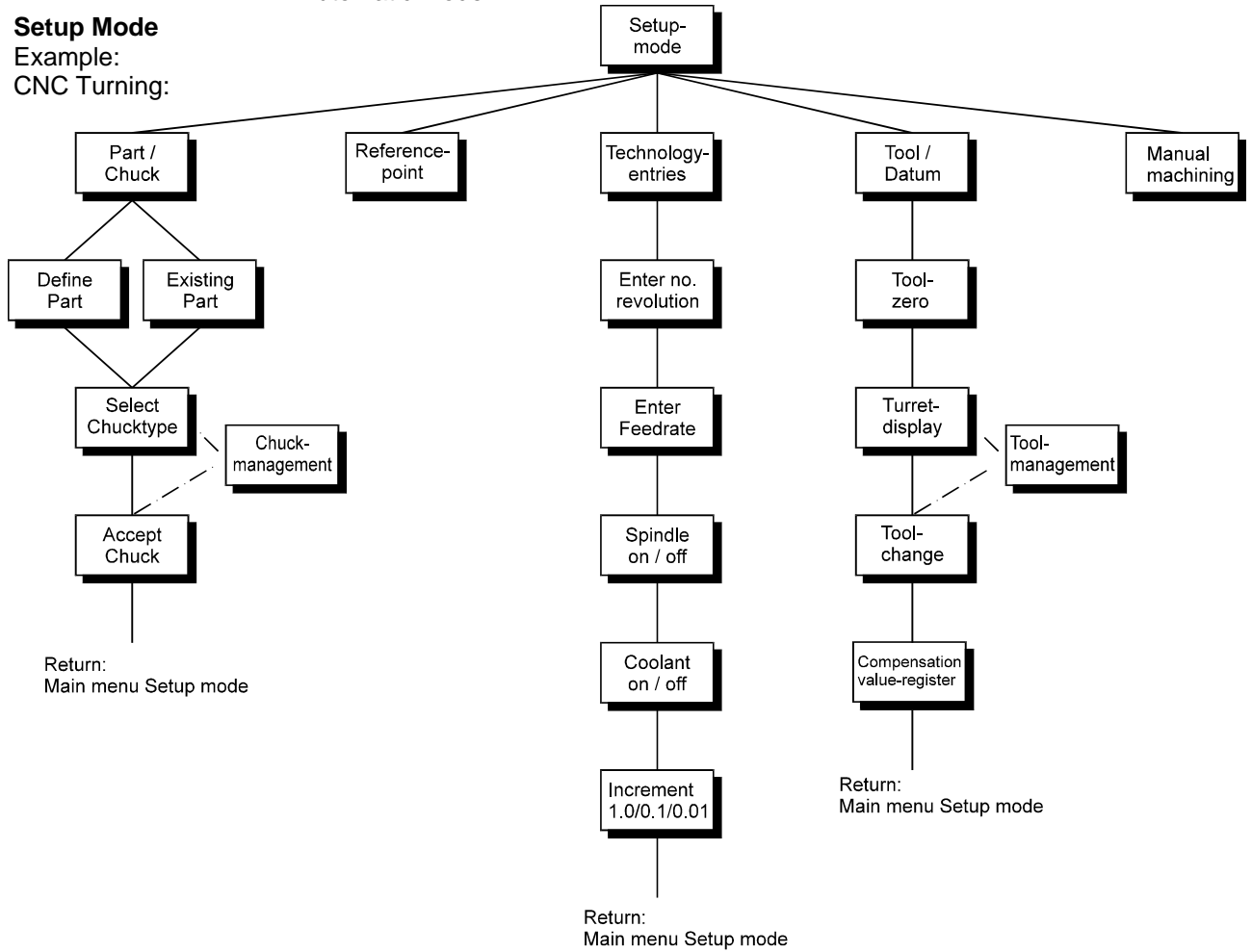
Figure 2.2-4:  
CNC Milling,  
Schematic of the work steps  
for testing an NC program  
in Automatic Mode.  
Like in Setup Mode, there is  
a simple method of starting  
a program: select Automatic  
Mode, enter the name of the  
NC program and then start  
the continuous automatic  
run.  
The "Graphical Display", "3D  
View" and "Override/Times"  
functions are also available.



The below two flow-charts show the most important work sequences in Setup and Automatic Mode:

**Setup Mode**

Example: CNC Turning:



**Automatic Mode**

Example: CNC Turning/CNC Milling:  
 The "Override/Times", "3D view" and "Graphical Display" functions are indicated as "Additional Functions" in the graphic

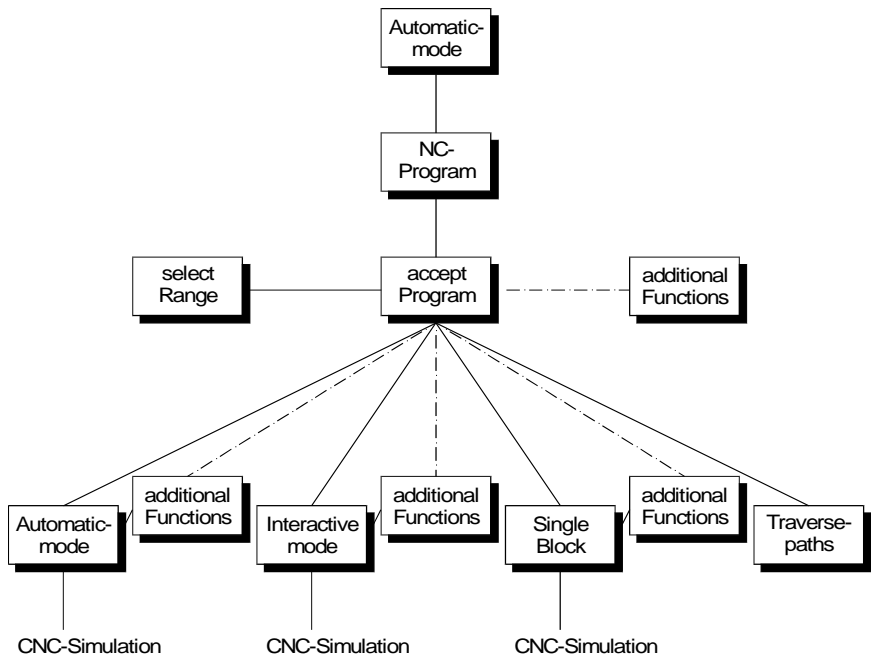
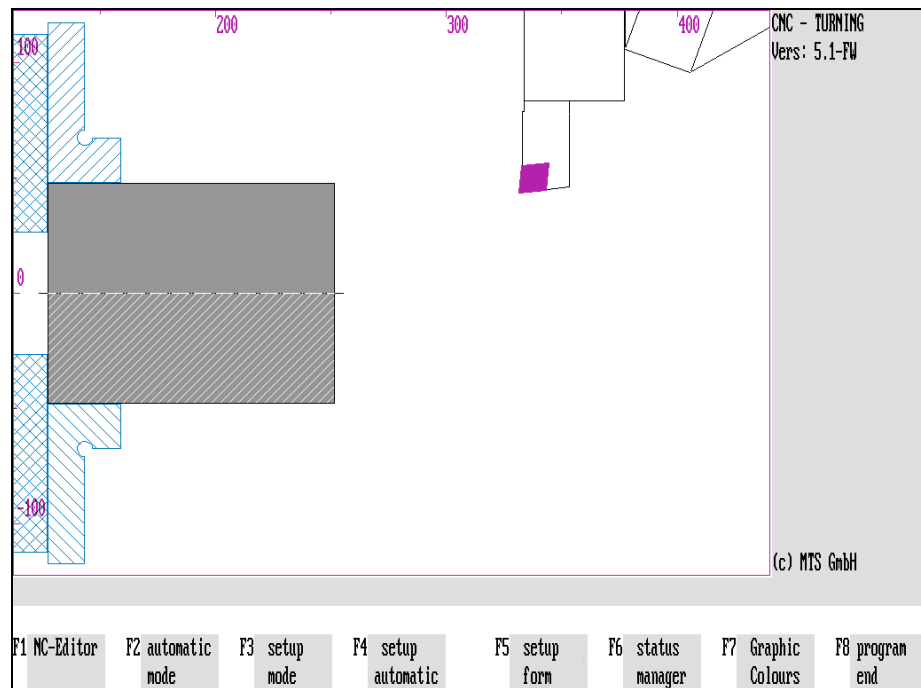


Figure 2.4-5: CNC Turning, Main menu; restricted processing options: in the configuration file you can specify whether certain operations are to be excluded. In this example, "Setup Mode" has been disabled, and can no longer be activated. Instead, "Automatic Setup" is available restoring the configurable start-up state.





Starting from a fixed starting point now try and see, which menus follow in sequence and how the editing steps are "interlocked" in consequence of the menu sequence. If in doubt, return to your starting point by pressing **[ESC]** or **[F8]**. Finally, you can always consult this manual to find the answer to your eventual questions.

## Shortened Menu Sequences

For training purposes etc., certain menu sequences can be deactivated in the configuration to reduce operation times or limit certain activities (e.g. tool and fixture management).

### Note

**In this case, the deactivated processing functions are no longer displayed and their function keys have no labels. Please check the configuration of your system** (see Configuration Instructions).



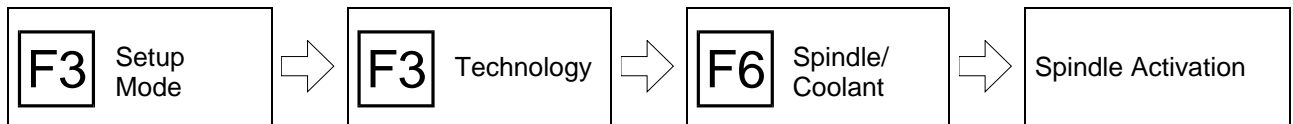
This manual documents, of course, the full scope of processing options.

## Using the CNC Keyboard or Tablet

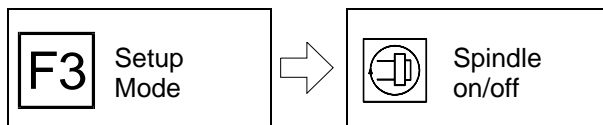
**Unlike the PC keyboard, the CNC keyboard or tablet allows you to select many functions directly. The intermediate menu steps can so be skipped!**

An example: spindle activation can be selected through the menus or directly with the corresponding key:

### Menu Selection:



### CNC Keyboard Selection:



**Always choose the selection method that appears the simplest !**

Figure 2.5-1:  
CNC Turning, Setup  
Mode;  
Entering blank dimen-  
sions.

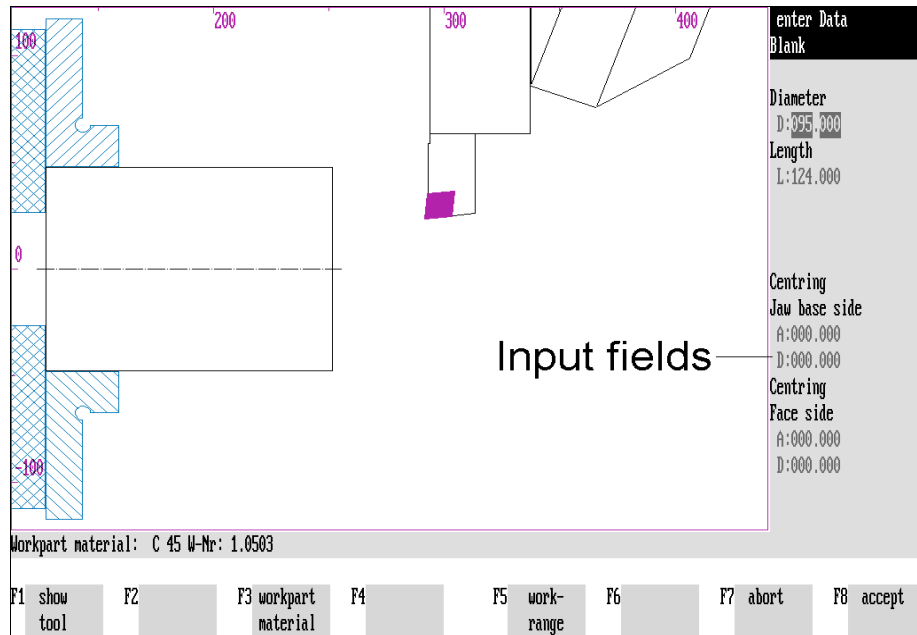


Figure 2.5-2:  
CNC Turning,  
WOP Interface;  
the address values are  
entered in the input  
fields in the right col-  
umn.

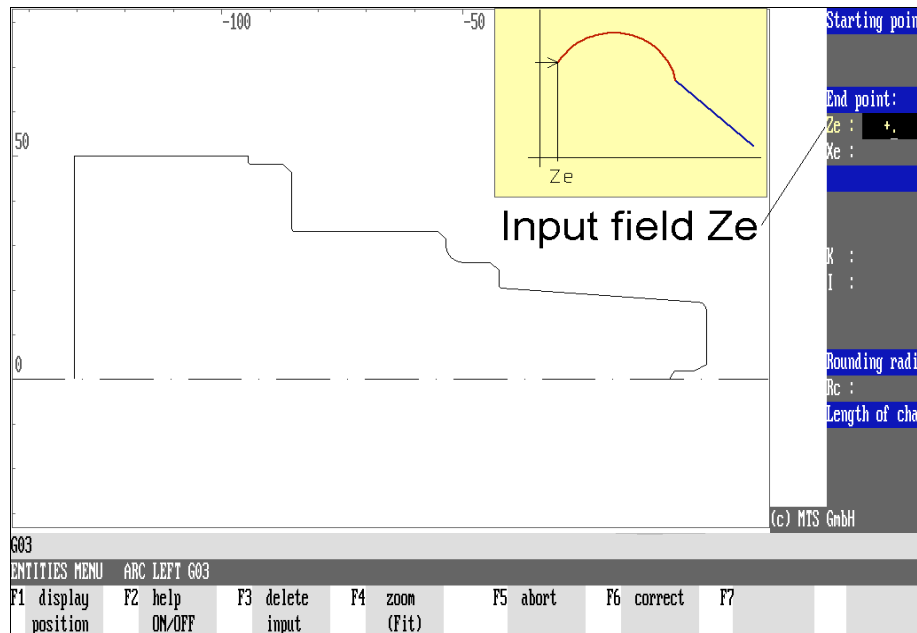
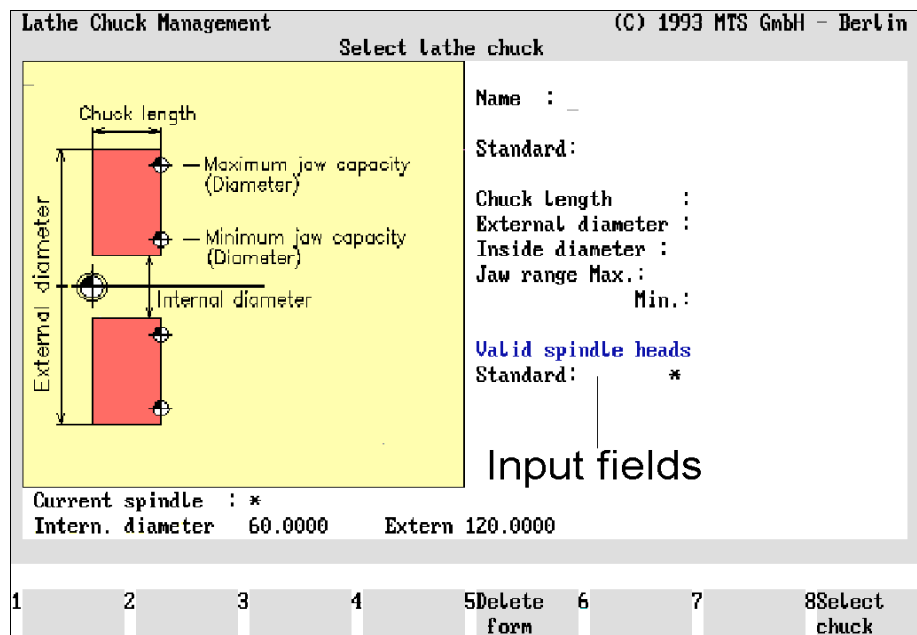


Figure 2.5-3:  
CNC Turning,  
Clamping Fixture Man-  
ager;  
Defining a lathe chuck.





## 2.5 Editing Menus

Beside the above-mentioned screen layout and the menu operations, the various modes of the CNC Simulator include editing menus requiring text or values to be entered. The input routine is illustrated in the following with two examples:

### Entries in the Information Column

In Setup Mode, the WOP interface and other menus, the input fields for values or other data are located in the information column in certain editing situations. There are various ways of activating an input field:



Press the  key to move from one input field into another.

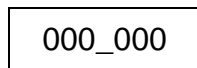


or



Alternatively, you can often select an input field using the  and  cursor keys.

An input field is active when highlighted in color; the cursor is initially placed in the middle of the input field or input entry:



There are two ways of entering the input:

**[Input]**

Either enter the desired value by typing the number digit by digit, with decimal points entered as <.> or <,>. The previous values are automatically deleted when you enter the new value.



or





Or, to change only a single digit, move the cursor to the right position and overwrite the existing entry.



or





The normal editing functions such as inserting characters: , deleting characters:  apply here too.



or




**Accept/End Input:** after having entered all required values in the input fields, conclude your input by pressing  or .



The procedure described above applies to all editing situations. Any extended functions or differences are described in connection of these specific editing functions.

### Entering NC Blocks

NC blocks are entered the same way the values described above. Even more simple: you only need to enter an address letter, and the address letter and fields are automatically displayed so that you can enter values. Now enter the desired value and then program the next NC word.

To delete an NC word, move the cursor to the address letter and press  to delete the entire NC word.



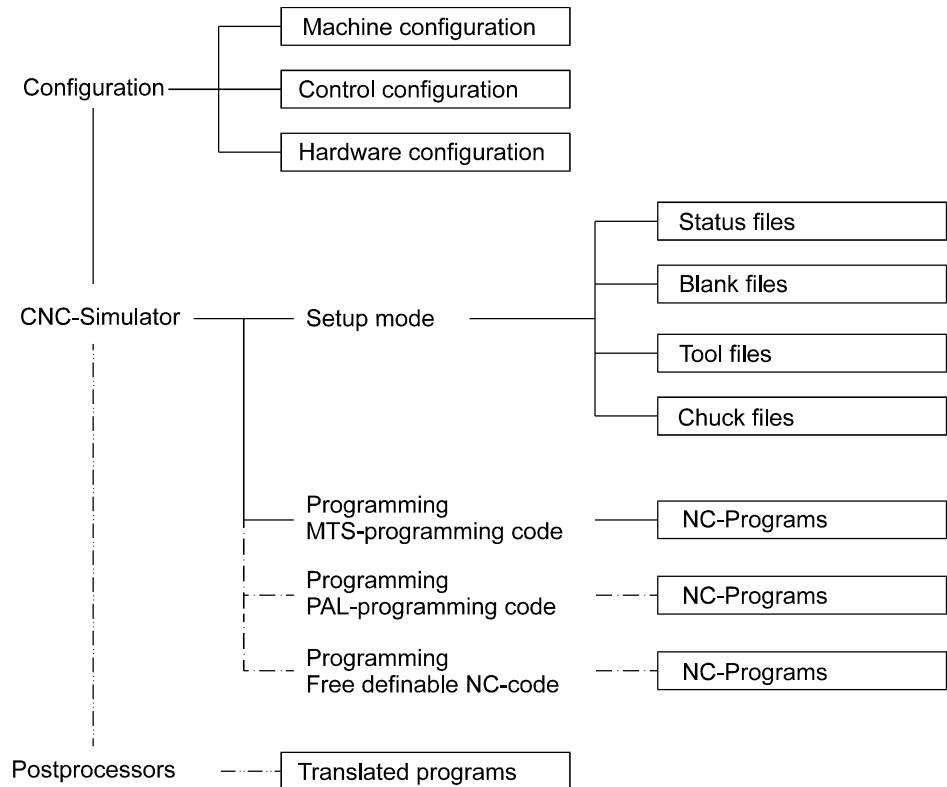
**Have a try and practice entering values and programming NC blocks !**

Figure 2.5-4:  
NC Editor,  
Program example.

```

M T S - GmbH Berlin
Program name : %229.DNC          Syntax : correct
-----
N130 G42
N135 /02 G22 U000229
N140 G40
N145 G00 X+150.000 Z+050.000
N150 (
N155 ( GEWINDEZYKLUS
N160 G97 S1200 T1616 M03
N165 X+035.000 Z+005.000
N170 G31 X+030.000 Z-026.000 A+030.000 D-002.147 F003.500 S0005
N175 G00 X+150.000 Z+100.000
N180 T0800 M04
-----
N185 X+052.100 Z-043.600 F002.000
-----
N190 G57 X+000.000 Z+000.000
N195 G83 X+052.100 Z-063.600 K+010.000
N200 G79 X+047.000 Z-064.000 I+001.000 K+001.000 A+001.000 W+001.000 P110 Q110
N205 G80
N210 (
N215 G00 X+100.000 Z+050.000 M30
1Link 2 group- 3 editing 4renumber 5 WOP 6 help 7 search 8 save/
Programs operation range
    
```

Figure 2.5-5:  
Schematic overview of  
the files that can be  
created or edited while  
working with the CNC  
Simulator.



## NC Editor

NC programs are edited directly using the editor's programming interface. Convenient functions are provided to support block input; you can check the syntax of address combinations and specified value ranges etc.

To improve readability, the screen layout of the editor is slightly different from the Setup and Automatic Mode interfaces described above (see figure):

The **header** contains the program name and the error messages.

The **NC program** is displayed below the header. If the NC program is long, the NC blocks can be scrolled up or down to display the entire program.

The **editing area** (input area) is marked by double lines.

As usual, the numbered boxes at the bottom of the screen indicate the program functions that can be selected using the **function keys** during editing.

### Note

When processing a program with the editor, keep in mind the difference between **editing functions** such as entering values, inserting, deleting etc., and **program functions** such as changing the block number, launching programming utilities, search functions etc.

## File Selection Window

As you work with the CNC Simulator there are various ways to save the results of your work in a file and load them again. These include:

- the **configuration files**, for saving default settings,
- the files of the **NC programs** or Setup Sheets,
- the **workpart files**, for saving created workpart geometries,
- the **status files**, for saving a certain stage of editing and simulation progress.

Individual program sections offer simple management functions, such as „Load“, „Delete“ and „Save“ to access above files without interrupting the simulation. No special knowledge of the operating system is required for their use.

### File Selection

In the "**File Selection Window**" you can select or name files. This window lists all the files stored in the currently defined path. The selected editing function is displayed at the top of the list of the NC programs.

The file selection window has the same structure in all file management functions of the CNC Simulator, and also works in the same way, with only a few slight differences:

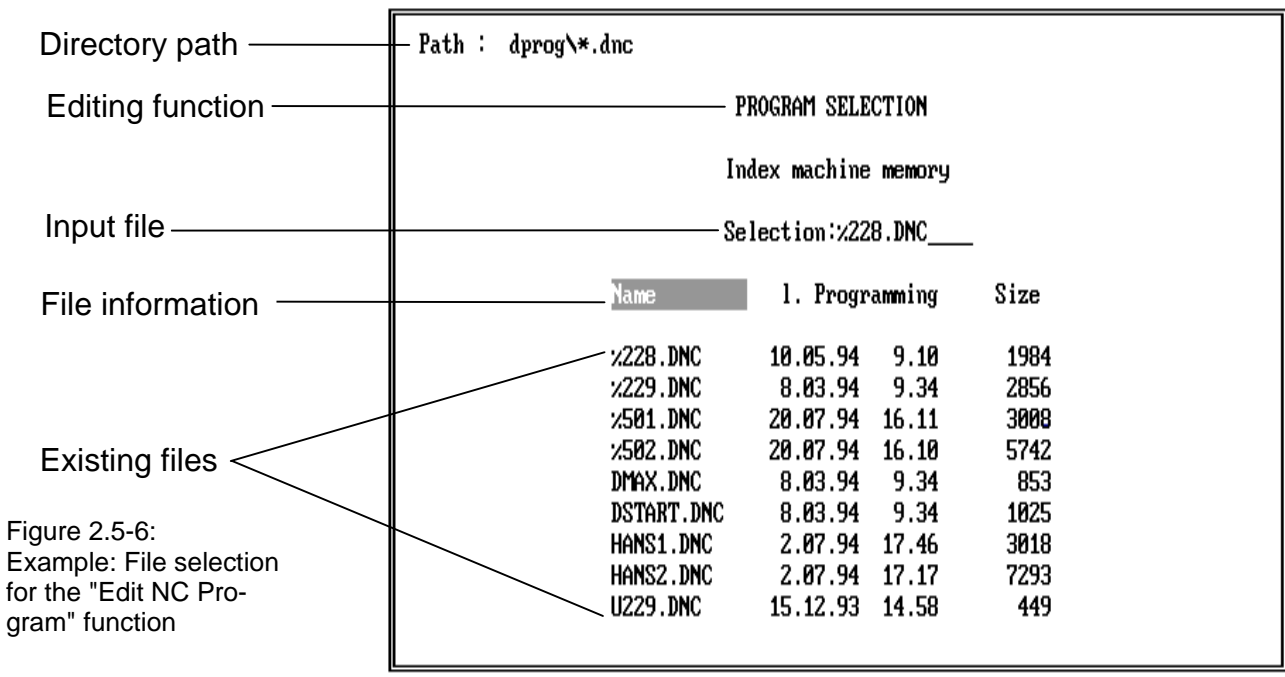


Figure 2.5-6:  
Example: File selection  
for the "Edit NC Pro-  
gram" function

Figure 2.5-7:  
The default path  
settings.

File	CNC Turning	CNC Milling
NC program (MTS)	.DPROG*.DNC	.FPROG*.FNC
NC program (PAL)	.DPAL*.PAL	.FPAL*.FPAL
Tool	.DWZ*.DVW	.FWZ*.DVW
Workpart	.DREHWS*.DWS	.FRAESWS*.FWS
Status	.*.DST	.*.FST
Configuration	KNF*.KNF	KNF*.KNF

## Procedure



[Enter Directory]

To change **directory**, press the key. This requires that access to the path has not been disabled in the configuration.

The "Path" input field is then highlighted and you can enter the desired directory address. You can also use wildcards "\*" and "?" to define groups of files.

The various files usually have a special **file name extension** (see below). In the displayed file list the files are listed according to the directory name and file name extension.

An example: the files for the turning programs have a default extension of ".DNC", so using the conventions of MS-DOS they are grouped under "\*.DNC". Assuming that the path for the subdirectory is ".DPROG", the correct path statement is : .DPROG\\*.DNC



Confirm the directory name with ; the cursor returns to the "selection field" and the file list displays the programs according to your path definition.



and



To change the **order criterion**, press the and keys at the same time. The cursor moves to the next field and the following file list is rearranged.

[Enter the  
Program Name]

In the selection field, enter the **name** of the file you want to create or edit. To do this, the file box provides additional functions which make it easier to select from the existing program files:



or



To select a file from the list, move the cursor using the and keys. The file name marked by the cursor automatically appears in the selection field.



or



If the program directory is very large, you can also browse through the file list using the and keys.



or



When overwriting an existing name you can move the cursor using the and keys.



The key deletes the character that is currently under the cursor in the selection field.



After having entered or selected the program name, confirm your selection with or the corresponding function key.



You can press to cancel the file selection and return to the starting point.

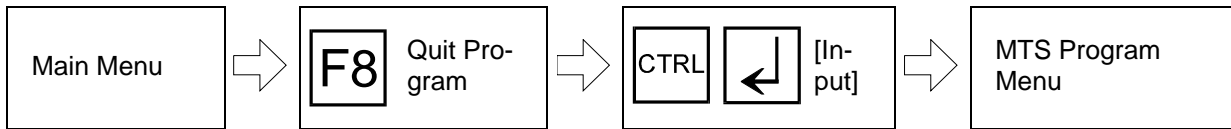
## Note

**Changes in directory** during program editing can be **disabled** with a corresponding entry in the configuration. You can also use the default file extensions or specify your own in the configuration (see figure).

## 2.6 Quitting the CNC Simulator

Close and quit the CNC Simulator from the main menu. After quitting the Simulator program, the Main Menu of the MTS programs (see above) is reloaded. You can now launch some other program or return to the operating system:

### Return:





### Procedure

To quit the CNC Simulator, you first need to conclude the work you are currently doing and return to the main menu.

**F8**

The **F8** function key starts the exit procedure. To make sure you really want to quit, you are prompted: "**CTRL and ENTER: Exit the Simulator**". This prompt remains displayed in the dialogue line for about 10 seconds.

**CTRL** and 

Use the key combination **CTRL** and  to terminate the program and return to the main MTS program menu.

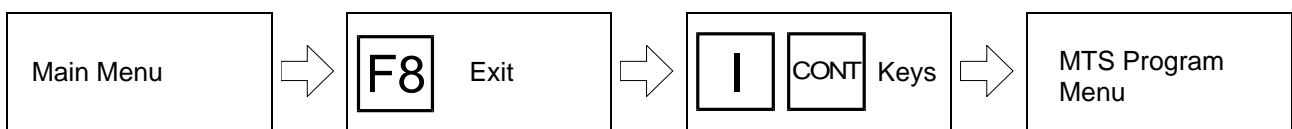
### Note

If you switch the computer off while the Simulator is running instead of concluding the program correctly, there may be "lost chains" on the hard disk. The MS-DOS "CHKDSK /F" command can be used to convert lost chains to files so that they can then be deleted.

## Using the CNC Keyboard or Tablet to Quit

When using the CNC keyboard or a tablet, the above dialogue is slightly different:

### Quitting Procedure on the CNC Keyboard/Tablet:



**F8**

The **F8** function key starts the exit procedure. To make sure you really want to quit, you are prompted: "**<I> and <CONT>: Exit the Simulator**". This prompt remains displayed in the dialogue line for about 10 seconds.

**I** and **CONT**

Use the key combination **<I>** and **<CONT>** to quit the program and return to the main MTS program menu.

### 3. CNC Simulator Main Menu

When starting the CNC Simulator, its main menu appears on the screen. It is the highest menu level, from which all other processing functions are activated.

#### Display

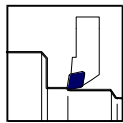
In order to standardize the user interfaces of the CNC Simulators the same display and operating principles are used in the turning and milling technologies. The screen is, in general, divided into **three areas**:

#### Screen Window

The **left screen window** in the main menu shows a section of the work space of the machine tool, with the blank or workpart, the clamping fixture and - depending on its position - the tool.

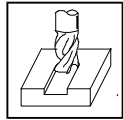
A graphic representation of the current work situation is always displayed in this window.

#### CNC Turning



The view of the **work space** shows a section in Z/X direction, depicting the turret, tool, fixture and, if visible, the tailstock and blank or workpart. This section can be zoomed or reduced continuously.

#### CNC Milling



The display of the **work space** shows a top view of the workpart. Additional sections and the crossing axes can also be displayed.

#### Information Column

The **right column** contains information about certain processing states. The fact that you are in the Main Menu means no mode has been selected yet. Therefore, no information is displayed here.

#### Menu Functions

The numbered boxes at the bottom of the screen show the program functions that can be selected using the **function keys** during processing.

Figure 3.-1:  
CNC Turning and  
Milling,  
Schematic of the Main  
Menu.

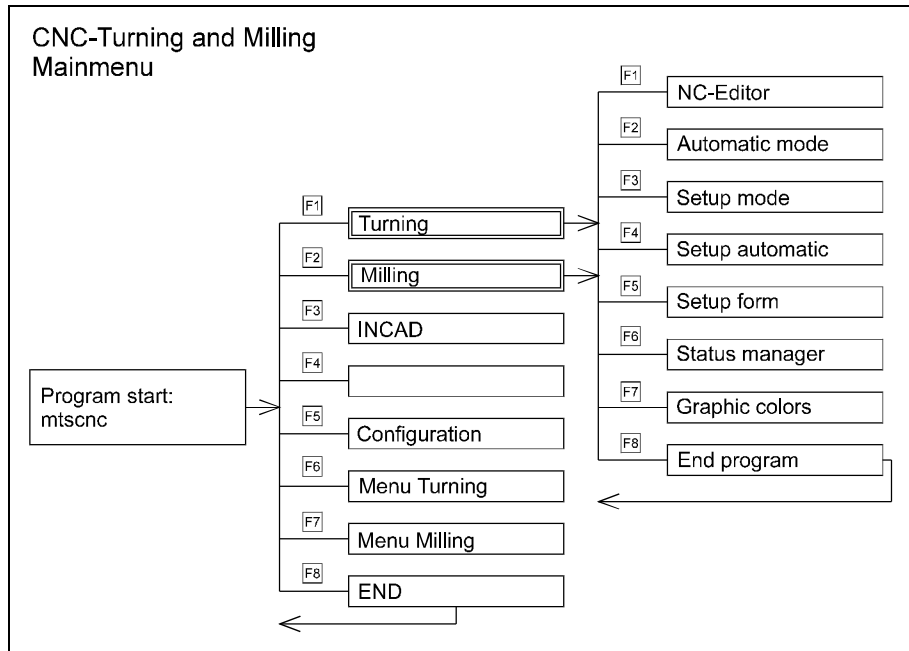


Figure 3.-2:  
CNC Turning,  
Main Menu.

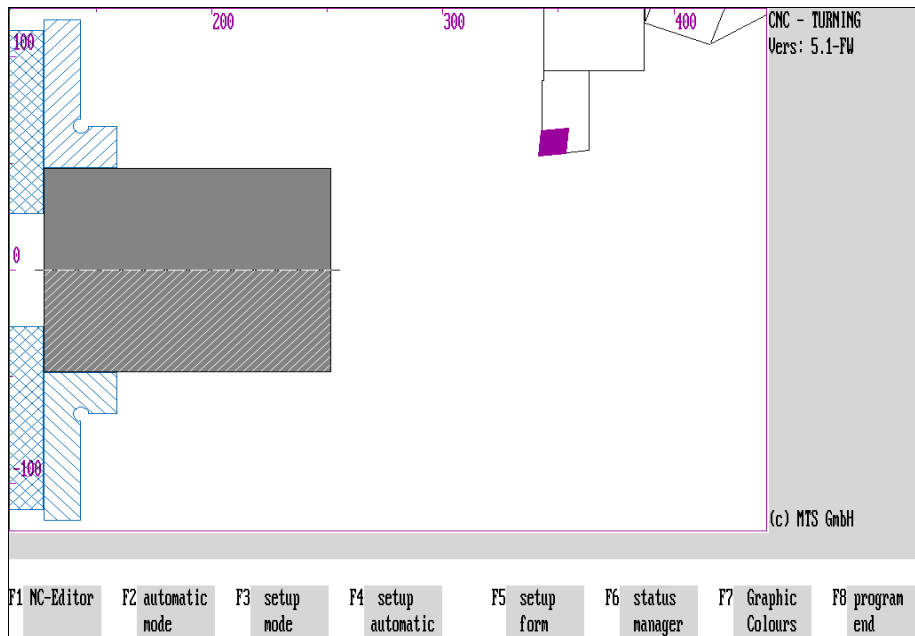
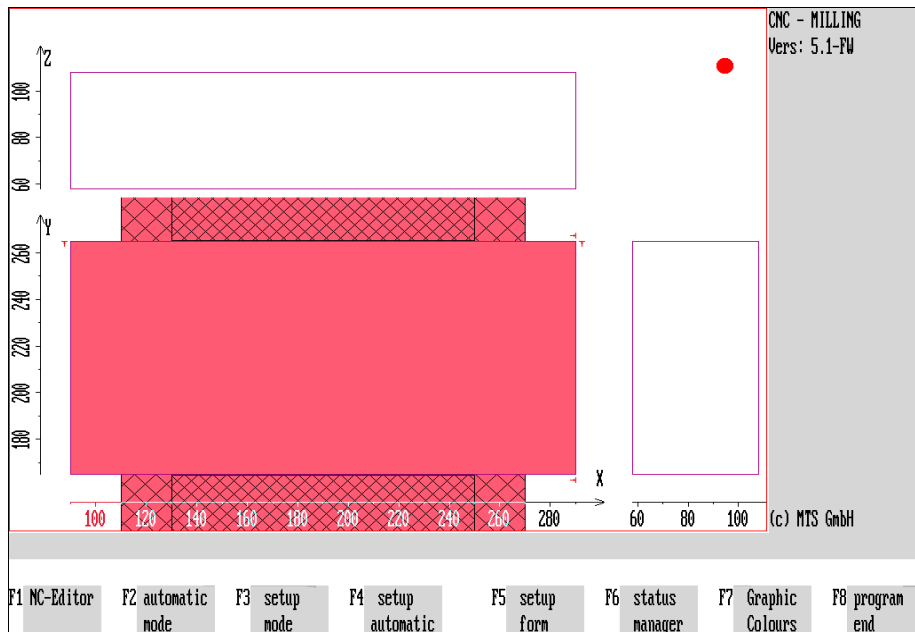


Figure 3.-3:  
CNC Milling,  
Main Menu.





The main menu offers you the following options:

F1

**NC Editor:** Press **F1** to start the NC program manager and the NC editor (see chapter 8).

F2

**Automatic Mode:** before you select Automatic Mode with **F2**, you have to go to the reference point in Setup Mode (see chapter 4).

F3

**Setup Mode:** Press **F3** to switch to Setup Mode (see chapter 4).

F4

**Automatic Setup:** Press **F4** to start the automatic load of the setup data as specified in the configuration.

F5

**Setup Sheet:** the "Setup Sheet" function has been added to version 5.0 of the CNC Simulator. It allows a specified setup to be generated as a setup sheet and to be included in an NC program. This enables you an automatic set up of the CNC Simulator during the program test.

F6

**Status Manager:** Press **F6** to run the status manager.

F7

**Graphics, Colors:** Press **F7** to run the color configuration (see chapter 10). For turning, this subfunction includes the graphics configuration for the simulation.

F8

**Exit Program:** Press **F8** to quit the CNC Simulator.

Figure 3.1-1: CNC Turning, Setup Sheet menu; if you have selected clamping between centers, for example, this setup data can be saved as a setup sheet.

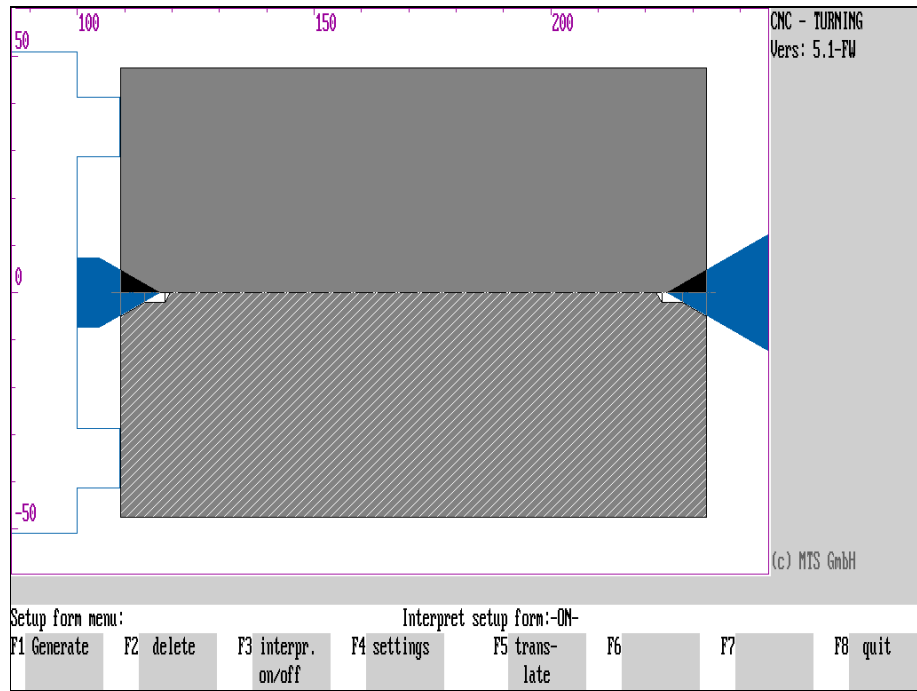
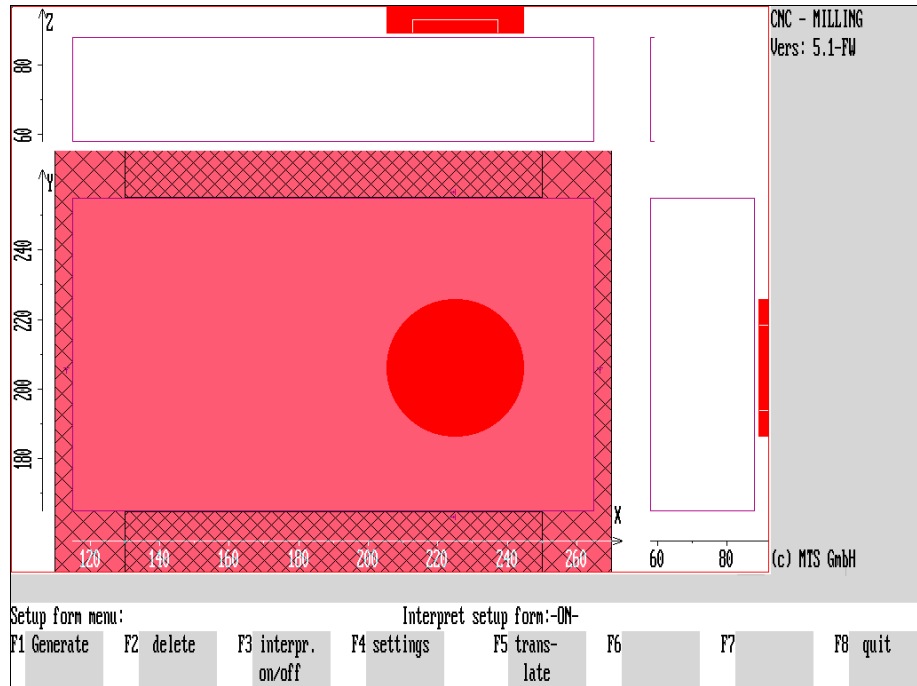


Figure 3.1-2: CNC Milling, Setup Sheet menu.



### 3.1 Setup Sheet and Automatic Setup

A Setup Sheet contains all the information needed for an automatic set up of the simulated machine tool when starting an NC program. This information includes:

- blank/workpart geometry
- clamping fixture and method
- tool in working position and turret/magazine configuration
- offset values of the tools used

A Setup Sheet can be created for every current machine tool status. It is prefixed to the NC program for which the setup status was created.

During the NC program load in Automatic Mode or for interactive programming the CNC Simulator is set up automatically with the **Setup Sheet Interpreter** - if activated - according to the stored information

To have a machine tool status loaded automatically during the CNC Simulator start, you can specify the Setup Sheet describing that status in the configuration. If you enter a Setup Sheet and a status file in the configuration, the Simulator will be set up according to the status file.

#### Selection:



**Setup Sheet:** from the main menu, activate the "Setup Sheet" function by pressing **F5**.

The following options are then available:

**F1**

**Create:** to create a Setup Sheet with the current machine status, press **F1**.

**F2**

**Delete:** to delete an existing Setup Sheet, press **F2**.

**F3**

**Interpreter on/off:** **F3** switches the Setup Sheet interpreter on and off. Its current status is shown in the dialogue line.

**F4**

**Settings:** With **F4** you can define which information is to be saved in the Setup Sheet.

**F5**

**Translate:** The information in the Setup Sheet can be saved as text or in coded form. Use **F5** to have the current state translated in accordance with the "settings".

**F8**

**Return:** **F8** takes you back to the main menu.



The Setup Sheet function saves you time because you need not set up the CNC Simulator every time you want to edit a program. At the same time this machine tool status can be listed as text in the NC programs, and can be checked or modified at any time (see Configuration Instructions). We recommend to print out the NC programs to have the data as a hardcopy as well.

#### Note

**Caution:** when creating a Setup Sheet for a certain machine tool status, it is automatically generated and inserted in the NC program without a prompt for confirmation. If the NC program already has a Setup Sheet prefixed to it, the existing information is overwritten without control query.

Figure 3.1-3:  
CNC Turning,  
example of a Setup  
Sheet (excerpt).

```

M T S - GmbH Berlin
Program name : %501.DNC          Syntax : correct
(
( M30
( MATERIAL C 45 W-Nr: 1.0503
(
( CHUCKING DEVICE
(   CHUCK SP 250-68 3
(   STEP JAW SP-GSBS 250
( TAILSTOCK POSITION Z+000.000
( CHUCKING DEPTH E32.000
(( right side of the part Z+292.000
(
( CURRENT TOOL T02

( TOOLS

(   T01 INSIDE RECESSING TOOL POSTAXI. ALT/541
(   T02 LEFT CORNER TOOL           ALT/508
(   T03 LEFT THREADING TOOL       ALT/050-1
(   T04 LEFT CORNER TOOL           ALT/506
(   T05 LEFT CORNER TOOL           ALT/517
(   T06 INTERN: THREADING TOOL POSTAX. ALT/047/1
1Link  2 group- 3 editing 4renumber 5 WOP  6 help  7 search 8 save/
Programs operation range          quit

```

Figure 3.1-4:  
CNC Milling,  
example of a Setup  
Sheet (excerpt).

```

M T S - GmbH Berlin
Program name : %520.FNC          Syntax : correct
( BLANK DIMENSIONS X+200.000 Y+100.000 Z+050.000
( PART POSITION X+090.000 Y+165.000
(( top front left corner of the part X+090.000 Y+165.000 Z+108.000
( MATERIAL ST 37-2 W-Nr: 1.0037
(
( VISE RS 110
( CHUCKED HEIGHT E+000.000
( SHIFT U+000.000
( ORIENTATION A90°
(
( CURRENT TOOL T11

( TOOLS

(   T01 DRILL                      D06
(   T02 RADIUS CUTTER              D20 DIN 844
(   T03 SLOT MILLING TOOL          D10
(   T04 SLOT MILLING TOOL          D10
(   T05 FACE END MILL              PMK-80
(   T06 T-SLOT MILL                T28 DIN 851T NUT DIN-650
1Link  2 group- 3 editing 4renumber 5 WOP  6 help  7 search 8 save/
Programs operation range          quit

```

### 3.1.1 Setup Sheet Syntax Rules

On the opposite page you see two examples of automatically generated Setup Sheets. When editing an NC program with the NC editor, you can also edit its Setup Sheet unless this option has not been included in the configuration. However, you need to follow the following syntax rules:

**Start and End Marker:** The character pair "**()**" marks the start and the end of the Setup Sheet. If one of these characters is missing, there is no guarantee for a correct program run.

**Beginning of Line** The beginning of each line is marked by the "(" character, which is normally used to mark comments.

**Delimiter** If various items are entered on one line, they need to be separated by at least one **space**.

**Keywords** Certain "keywords" are used for setup information. They indicate the element in the machine space the subsequent information refers to. The following pages give a summary of the keywords.

Exm.: ( CYLINDER D100 L135

The "(" character defines the beginning of the line and the following word "**CYLINDER**" is the keyword defining the blank.

**Parameters** The keywords are followed by the dimensions, or the object or file names of the elements.

Exm.: ( T02 RADIUS CUTTER D20 DIN 844

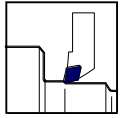
The radius cutter "**D20 DIN 844**" is located in magazine position "**T02**".

**Element Groups** Information belonging together is given in groups for better readability. These groups make up operational units and must be kept together:

Exm.: ( VICE RS 110  
           CLAMP HEIGHT E+029.000  
           OFFSET V+000.000                    ]                    Group  
           ...  
       ( TOOLS  
           T01 DRILL D06  
           T02 RADIUS CUTTER D20 DIN 844       ]                    Group  
           ...

**Comments** If you want to include comments or remarks in the Setup Sheet, insert a further bracket "(" in front of the comment. Certain comments, e.g. "left front top corner of workpart", are inserted automatically.

## CNC Turning: Setup Sheet Keywords



### Configuration

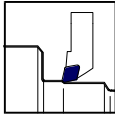
### Blank

### Material

### Fixture

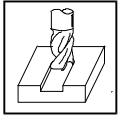
### Clamping Method

Keyword	Meaning
()	Start/end marker of the Setup Sheet; these must not be deleted
( CONFIGURATION	Group identifier for configuration
( MACHINE	The MTS file name denoting the desired machine tool configuration
( CONTROL	The MTS file name denoting the desired control configuration
( CYLINDER	Definition of blank, followed by blank dimensions
( CYLINDER WITH CENTERING	Definition of blank, followed by blank dimensions
( TUBE	Definition of blank, followed by blank dimensions
( TUBE WITH BEVEL	Definition of blank, followed by blank dimensions
( BLANK GEOMETRY	Definition of blank geometry, followed by details of the geometry as an NC program
( BLANK FILE NAME	Definition of workpart file, followed by the file name
( MATERIAL	Definition of workpart material
( FIXTURE	Group identifier, clamping elements in next line
( LATHE CHUCK	Definition of lathe chuck, followed by chuck designation
( STEPPED JAW	Definition of stepped jaws, followed by designation: jaw set
( LATHE CENTER	Definition of lathe center, followed by designation: lathe center
( FACE DRIVER	Definition of face driver, followed by designation: face driver
( CLAMPING METHOD	Group identifier, next line indicates clamping method
( OUTSIDE CLAMPING	Defines clamping method (no param.)
( INSIDE CLAMPING	Defines clamping method (no param.)
( OUTSIDE STEP	Defines stepped jaws (no param.)
( INSIDE STEP	Defines stepped jaws (no param.)
( TAILSTOCK POSITION	Defines tailstock position, followed by coordinates
( SET SLEEVE	Defines the sleeve (no parameter)
( DO NOT SET SLEEVE	Defines the sleeve (no parameter)
( CLAMP DEPTH	Defines the clamping method, followed by clamping depth
( RECLAMP	Defines reclamping (no parameters)

**Tools****Compensation value register**

Keyword	Meaning
<b>( CURRENT TOOL</b>	Defines the work position, followed by the turret position under "T".
<b>( TOOLS</b>	Group identifier turret assignment; details: under ID letter "T" the turret position, followed by the tool type and number.
RIGHT-HAND CORNER TOOL LEFT-HAND CORNER TOOL INTERNAL CUTTING TOOL PREAXIAL INTERNAL CUTTING TOOL POSTAXIAL ROUND-NOSE EXTERNAL CUTTING TOOL COPYING TOOL INTERNAL RECESSING TOOL PREAXIAL INTERNAL RECESSING TOOL POSTAXIAL AXIAL RECESSING TOOL EXTERNAL RECESSING TOOL RIGHT-HAND THREADING TOOL LEFT-HAND THREADING TOOL TWIST DRILL CENTER DRILL THREADING DRILL REVERSIBLE TIP DRILL TAPPING TOOL PREAXIAL TAPPING TOOL POSTAXIAL	[tool designations]
<b>( T.. EMPTY</b>	no tool inserted
<b>( ZERO ENGAG. TIMES</b>	Zero the tool engagement times (no param.)
<b>( CORRECT COMPENSATION VALUES</b>	Add the compensation values of the tools used to the associated registers (no parameters)
<b>( COMPENSATION VALUES</b>	Group identifier. Tool comp. values; details: under "D" the register and then the compensation values.

## CNC Milling: Setup Sheet Keywords



### Configuration

### Blank

### Material

### Chuck

String	Meaning
<b>()</b>	Start/end of the Setup Sheet; the markers must not be deleted.
<b>( CONFIGURATION</b>	Group identifier for configuration
<b>( MACHINE</b>	The MTS file name denoting the desired machine configuration
<b>( CONTROL</b>	The MTS file name denoting the desired control configuration
<b>( BLANK DIMENSIONS</b>	Blank definition, followed by the dimensions
<b>( BLANK FILE NAME</b>	Definition of the workpart file, followed by the file name and extension
<b>( WORKPART POSITION</b>	Definition of workpart position, followed by the X/Y coordinates referred to machine datum
<b>( MATERIAL</b>	Definition of workpart material
<b>( VICE</b>	Group identifier for clamping with vice, followed by the vice number
<b>( CLAMP HEIGHT</b>	Definition of clamping height, followed by the value under E
<b>( OFFSET</b>	Definition of clamping, offset of vice in v
<b>( ALIGNMENT</b>	Definition of clamping, alignment of the vice in °
<b>( MODULAR CLAMPING</b>	Group identifier for modular clamping, in the following lines are: dimension and position of the chucking modules by X/Y/Z coordinates referred to machine datum
<b>( CLAMP HEIGHT</b>	Definition of clamping height, followed by the value under "E"
<b>( MAGNETIC PLATE CLAMPING</b>	Definition of clamping type (no parameters)
<b>( RECLAMP</b>	Group identifier for reclamping (no parameters)
<b>( HORIZONTAL</b>	Reclamping mode (no parameters)
<b>( VERTICAL</b>	Reclamping mode (no parameters)
<b>( PLANE SELECTION</b>	Definition of plane (G17/G18/G19)
<b>( CURRENT TOOL</b>	Definition of working tool, followed by the magazine position under "T".
<b>( TOOLS</b>	Group identifier for magazine assignment; details: under ID letter "T" the magazine position, followed by the tool type and number



**Compensation value  
register**

SHANK-TYPE END MILLS [Tool designations]	
FLUTING CUTTER	
T-SLOT CUTTER	
SHELL END MILL	
FACE END MILL	
RADIUS MILLING CUTTER	
ANGULAR MILLING CUTTER FORM A	
ANGULAR MILLING CUTTER FORM B	
REAMER	
TAP	
REVERSIBLE TIP DRILL	
STEP DRILL	
COUNTERSINK	
CONCAVE MILLING CUTTER	
SIDE MILLING CUTTER	
TWIST DRILL	
<b>( T.. EMPTY</b>	no tool inserted
<b>( CORRECT COMPENSATION VALUES</b>	Add the compensation values of the tool used to the associated registers (no parameters)
<b>( COMPENSATION VALUES</b>	Group identifier. Tool compensation values; in the following lines the register is entered under "D" followed by the correction values.
<b>( ZERO ENGAG. TIMES</b>	Zero the tool engagement times (no param.)

Figure 3.1.2-1:  
CNC Turning,  
Setup Sheet,  
Menu of default  
settings

Program management (C) 1993 MTS GmbH - Berlin  
Modifications of the setup form

Switch to generate data in the setup form

Workpart	:	yes
Clamping devices	:	yes
Tools	:	yes
Offset value storage	:	Valid offset values
Configuration	:	no

Create setup form : in english

Translate setup form : in english

1 2 3 4 5 6 7 8 Accept

### 3.1.2 Setup Sheet Default Settings

Before the Setup Sheet is automatically generated, you can specify in the submenu "Settings" which data of the current machine status you want to save. To do this, select the corresponding data in the input fields of the upper screen area:

#### Procedure



Use the **TAB** key or the key combination **Shift** + **TAB** to activate the input fields one by one.



or



Use the **↓** and **↑** cursor keys to select the available data in the input field. The following settings are possible:

Input Field	Settings
Workpart	<b>yes/no</b> : this setting specifies whether the workpart data are to be included.
Fixture	<b>yes/no</b> : this setting specifies whether the clamping fixture data are to be included.
Tools	<b>yes/no</b> : this setting specifies whether the tool data are to be included.
Offset Values	<b>Correct comp. values/List of storage/no</b> : this setting specifies whether the compensation values are to be included.
Configuration	<b>yes/no</b> : this setting specifies whether the configuration files are to be included.
Save	<b>in [language]/no language</b>
Setup Sheet	The [language] setting (e.g. "English") saves the text information in the current language; "no language" means that the text information is coded. In this case the Setup Sheet can be used in various different language versions of the CNC Simulator.
Target language for translation	<b>to [language]/no language</b> To translate a text Setup Sheet to a coded version (or vice versa), choose the corresponding setting.



or



**Cancel:** **F7** or **ESC** cancels the function.



or



**Return:** **F8** or **↵** accepts the current settings and returns you to the starting point.

Figure 3.1.3-1:  
CNC Turning  
Creating the Setup  
Sheet;  
Selecting the NC  
program in which to  
include the current  
machine tool status as  
a Setup Sheet.

```

Path : dprog\*.dnc

          setup form

          GENERATE

          Selection: %228.DNC ____

Name      1. Programming      Size
-----
%228.DNC  10.05.94   9.10      1984
%229.DNC   8.03.94   9.34      2856
%501.DNC  20.07.94  18.01     4425
%502.DNC  20.07.94  16.10     5742
DMAX.DNC   8.03.94   9.34       853
DSTART.DNC 8.03.94   9.34     1025
HANS1.DNC  2.07.94  17.46     3018
HANS2.DNC  2.07.94  17.17     7293
U229.DNC  15.12.93  14.58       449

Select with arrow keys, Abort - ESC, Accept - Enter

```

### 3.1.3 Create Setup Sheet

Before creating a Setup Sheet you need to specify the NC program to which it is to be prefixed, or to enter a new program name:



**Create:** to create a Setup Sheet recording of the current machine tool status, press **F1**.

The File Selection window (see chapter 2) is then displayed listing all NC programs in the current path:



or



You can go through the list using the **↓** and **↑** cursor keys to select an existing NC program.

[File Name]

Or you can enter the name of the NC program file directly in the "Selection" input line.

**You have to enter the file name always directly when creating a Setup Sheet for a new NC program.**



Then confirm your input with **↵**.

The Setup Sheet is then automatically generated and prefixed to the NC program. If the selected NC program already contains a Setup Sheet, the existing setup information is overwritten without prompting the user for confirmation.

New Program

When entering a **new** file name, not only the Setup Sheet but also the NC program has to be created as a new file, therefore, you are prompted as follows:

The program does not exist!  
Create program? (Y/N)



or



Press **N** or **ESC** to cancel the procedure.



or



Press **Y** or **↵** to confirm that you want to create the program.  
The new NC program and Setup Sheet are then created.

### 3.1.4 Delete Setup Sheet

This function allows you to delete an existing Setup Sheet from an NC program. It works in the same way as above (activate function, enter file name, confirm).



**Now try to use the "Create Setup Sheet" and "Delete Setup Sheet" functions with a new NC program!**

**Also switch on the Setup Sheet Interpreter, to generate Setup Sheets for various machine tool states and load these NC programs in Automatic Mode. This practical exercise demonstrates how the CNC Simulator is automatically set up.**

Figure 3.1.5-1:  
CNC Turning,  
a sample Setup Sheet  
with text information  
(excerpt).

```

M T S - GmbH Berlin
Program name : %228.DNC          Syntax : correct
(
( @'80:17' T4
( @'80:4'
(      T01 @'80:129' ALT/545
(      T02 @'80:123'      ALT/508
(      T03 @'80:130'      ALT/603
(      T04 @'80:123'      ALT/506
(      T05 @'80:123'      ALT/517
(      T06 @'80:188' ALT/047/1
(      T07 @'80:127'      ALT/519
(      T08 @'80:131'      ALT/539
(
(      T09 @'80:125'      ALT/521
(
(      T10 @'80:127'      ALT/520
(      T11 @'80:139'      ALT/057
(      T12 @'80:123'      ALT/506
(      T13 @'80:135'      ALT/054
(      T14 @'80:123'      ALT/598
(      T15 @'80:134'      ALT/075
1Link  2 group- 3 editing 4renumber 5 WOP  6 help  7 search  8 save/
Programs operation range          quit
    
```

Figure 3.1.5-2:  
CNC Turning,  
a sample Setup Sheet,  
translated version in  
coded form (excerpt).

```

M T S - GmbH Berlin
Program name : %228.DNC          Syntax : correct
(
( CURRENT TOOL T09
( TOOLS
(      T01 INSIDE RECESSING TOOL POSTAXI. ALT/545
(      T02 LEFT CORNER TOOL      ALT/508
(      T03 FRONT GROOVING TOOL  ALT/603
(      T04 LEFT CORNER TOOL      ALT/506
(      T05 LEFT CORNER TOOL      ALT/517
(      T06 INTERN: THREADING TOOL POSTAX. ALT/047/1
(      T07 COPYING TOOL          ALT/519
(      T08 RECESSING TOOL        ALT/539
(
(      T09 INSIDE TURNING TOOL POST ALT/521
(
(      T10 COPYING TOOL          ALT/520
(      T11 REVERSIBLE TIP DRILL  ALT/057
(      T12 LEFT CORNER TOOL      ALT/506
(      T13 CENTER DRILL          ALT/054
(      T14 LEFT CORNER TOOL      ALT/598
(      T15 TWIST DRILL           ALT/075
1Link  2 group- 3 editing 4renumber 5 WOP  6 help  7 search  8 save/
Programs operation range          quit
    
```

### 3.1.5 Translate Setup Sheet

The CNC Simulator is available in various language versions, including German and English. To allow you to use existing NC programs and Setup Sheets when changing from one language into another these can be translated into a "language-neutral" version first. You can then switch into the language of your option and load the language-neutral versions and have them translated into the current language.

In this automatic translation process, all keywords of the Setup Sheets are either translated into coded strings of characters or translated from a coded version into a language version.



In the submenu "Settings" you first specify which version the Setup Sheet of the NC program is to be translated into.

**F5**



**Translate:** To have the Setup Sheet of an existing NC program translated, press **F5**.

The File Selection window (see chapter 2) is then displayed, listing all NC programs in the current path:



or




Use the  and  keys to go through the list to select an existing NC program.

[File Name]

Or enter the name of the NC program file directly in the "Selection" input line.



Then confirm your input with . The Setup Sheet is then automatically translated.

### 3.1.6 Automatic Setup

To facilitate setting up the CNC Simulator, an existing setup status that has been specified in the configuration (see Configuration Instructions) can be automatically loaded:

**F4**

**Automatic Setup:** this function is activated by pressing the **F4** function key from the main menu. The CNC Simulator is then set up automatically.



The purpose of the "Automatic Setup" function is to minimize recurring setup work. It is particularly useful for the CNC training when the focus is on NC programming using standard machine settings. In the configuration you specify which existing Setup Sheet is to be loaded automatically.

Figure 3.2-1:  
CNC Turning,  
Status Manager

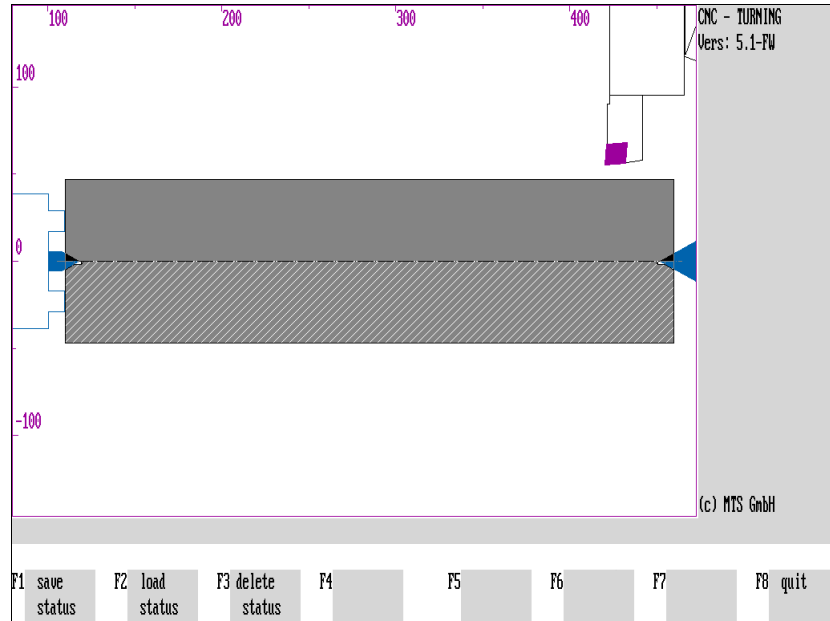


Figure 3.2-2:  
CNC Milling,  
Status Manager

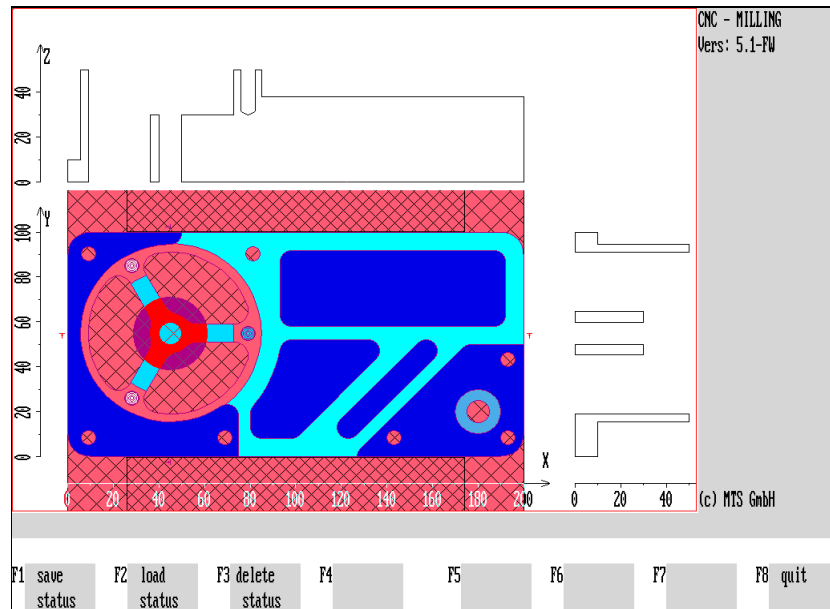
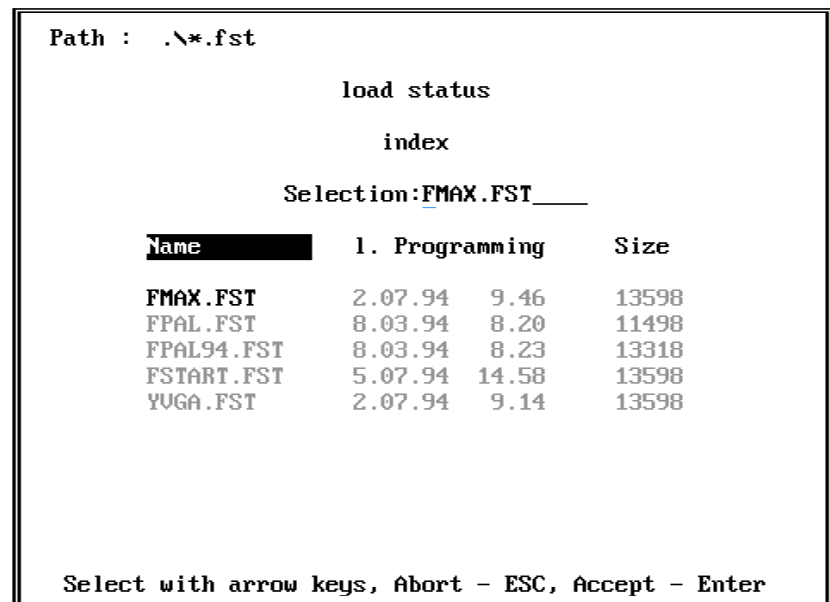


Figure 3.2-3:  
CNC Milling,  
File Selection window  
with existing status  
files





## 3.2 Status Manager

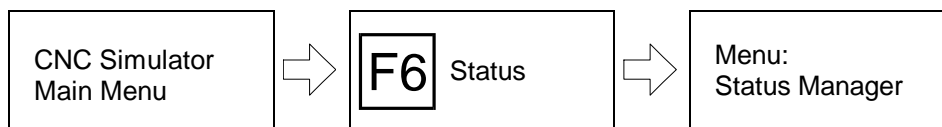
In addition to the Setup Sheet function, the CNC Simulator also allows you to save any editing status as a "Status File" and to load it again later on.

The editing status includes:

- exact workpart geometry
- clamping method
- turret/magazine configuration, compensation values and current working tool
- current technology values

The machining of workparts can thus be interrupted and resumed at a later time, or it can be done in sections without having to repeat previous operation steps.

### Selection:



**F6**

**Status:** the Status Manager is activated by pressing **F6** from the main menu of the CNC Simulator.

The following options are then available:

**F1**

**Save status:** to save the current editing status, press **F1**.

**F2**

**Load status:** to load a stored editing status, press **F2**.

**F3**

**Delete status:** to delete a saved editing status, press **F3**.

**F8**

**Return:** **F8** returns you to the Main Menu.

The procedures in the status manager are analogous to the Setup Sheet functions.



Like the Setup Sheet, the Status Manager saves time. with the difference, however, that a processing status usually also includes the workpart machining steps stored in an NC program.

To keep your system well-organized, we recommend to save the current status always together with the reference to the NC program.

### Note

The configuration of the CNC Turning or Milling Simulator allows you to load a certain status as soon as you start the system (see Configuration Instructions, Machine Tool Configuration). If the configuration contains a Status File and a Setup Sheet at the same time, the Simulator is automatically set up according to the Status File.

## Load Status



Press **F2** to activate the "Load Status" function. All saved status files are listed for your selection (see figure).



or



Move the cursor to the desired status file or enter the corresponding **file name**.



Then press **Enter** to confirm the input. You then return to the Main Menu and the selected status is loaded.

## Save Status



To save the current processing status in a file, press **F1**. Like in the "Load status" function, a list of existing status files is displayed in the selection window.

[File Name]

Enter the **name you want to save the current status with** .



Then confirm your input with **Enter**. The processing status is automatically saved and you return to the Main Menu.  
A saved status can be loaded again at any time.

### Note

If you enter an already existing file name, the (existing) file is **overwritten**, i.e. the old status information is deleted. You are **not prompted to confirm!** You should therefore always check your inputs before confirming with **Enter**.  
We also recommend to use file names that can be **clearly associated with their function**. This makes it easier to identify the files later on.

## Delete Status



Press **F3** to delete an existing status file. As in the "Load Status" function, the File Selection window is displayed.

[File Name]

Enter the **name of the file you want to delete** .



If you confirm this input with **Enter**, you are prompted as follows:

Do you really want to delete this status?

**ESC** = NO, **Enter** = YES



If you want to check your inputs one more time, cancel the procedure with **ESC**.



If you confirm the prompt by pressing **Enter**, the status file is deleted and you return to the Status Manager.

### Note

The **status manager path** shown at the top left corner of the selection window is a default set during software installation. It is not possible to choose an other directory for this.

## 4. Setup Mode

The setup or manual mode is used for **preliminary work** such as:

- Inserting the blank/preproduced workpart,
- Selection of the clamping method and clamping method management
- Tooling (configuration) of the turret/magazine and tool manager,
- Entry of the tool compensation values,
- Go to the reference points,
- Determining the workpart zeropoint etc.

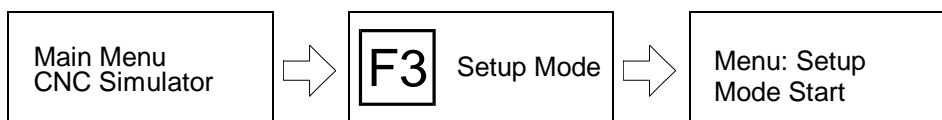
The CNC Simulator thus includes the same operation steps as are required to set up a machine tool. Here all functionally necessary activities (referencing, entry of the tool compensation values) are **obligatory**.

Of course, it is possible to **process with manual control** as well.

### Note

The entries you make in the setup mode have to agree with the presettings of the configuration. Otherwise the corresponding error message is displayed. The currently selected configuration files (cf. configuration instructions) are relevant for processing.

### Selection:



Proceeding from the main menu, you call up the setup mode with the function key **F3** (PC keyboard)

Figure 4.-1:  
CNC Turning and Milling,  
schematic of the Setup  
Mode menu

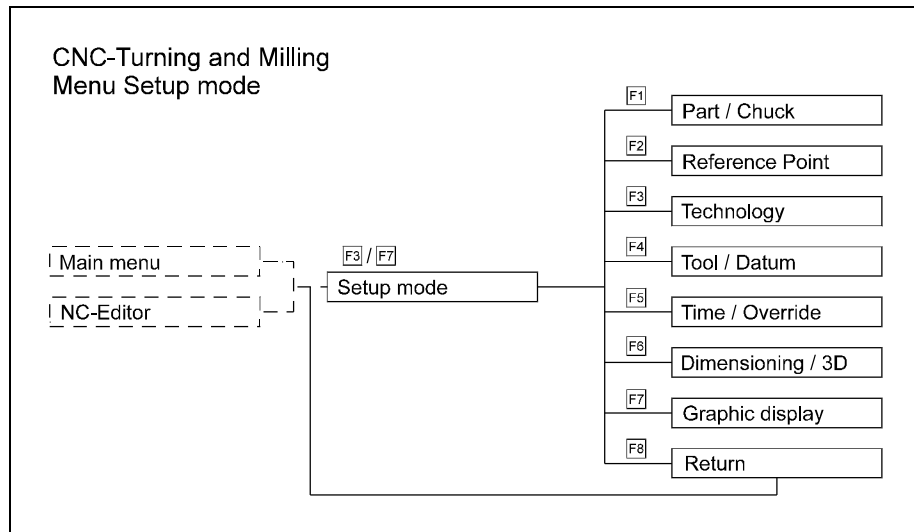
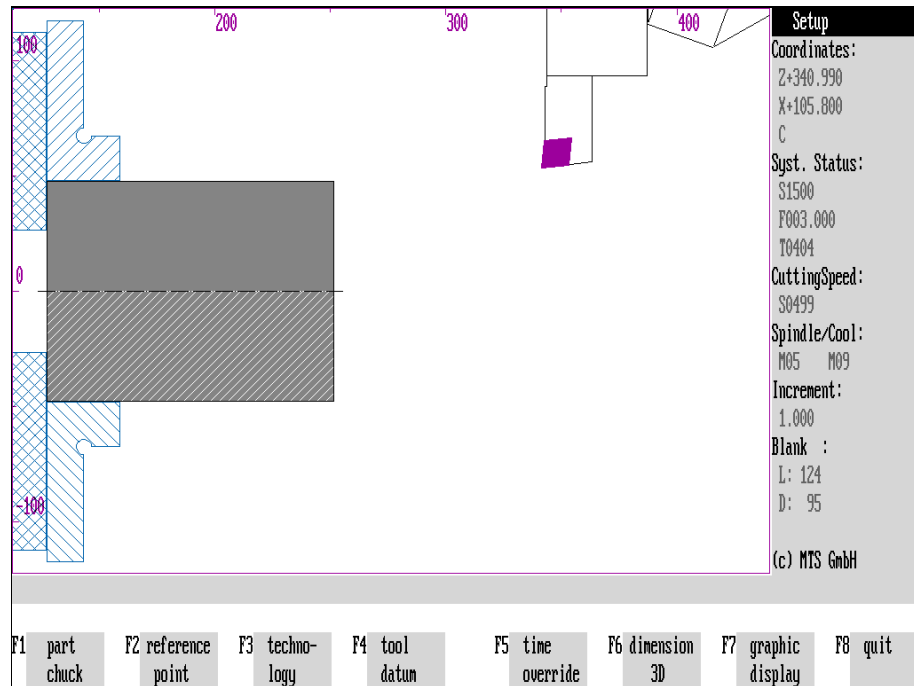


Figure 4.-2:  
CNC Turning,  
Setup Mode menu.



## Setup Mode Menu

Due to the extensive processing possibilities of the version 5 in the setup mode it was necessary to make **various submenus to the Setup Mode menu**. The following program functions are called from within the Setup Mode menu with the **function keys**:

F1

**Workpart/Clamping Fixture:** With **F1** you select the workpart and clamping fixture manager to determine a new blank, to select the clamping method and to determine the clamping.

F2

**Reference Point:** You move to the reference point with **F2**.

F3

**Technology:** To enter the technology values, select **F3**.

F4

**Tool/Zeropoint:** With **F4** you switch to the tool manager to determine the zeropoint.

F5

**Override/Times:** To change the feedrate override and to determine the engagement times of the tools, select **F5**.

F6

**[Measuring] 3D view:** With **F6** you switch to the "Measuring" functions (only CNC Turning) or 3D view.

F7

**Graphical Display:** To change the display, select **F7**.

F8

**Return:** **F8** returns you to the main menu.

Figure 4.-3:  
CNC Milling,  
Setup Mode menu.

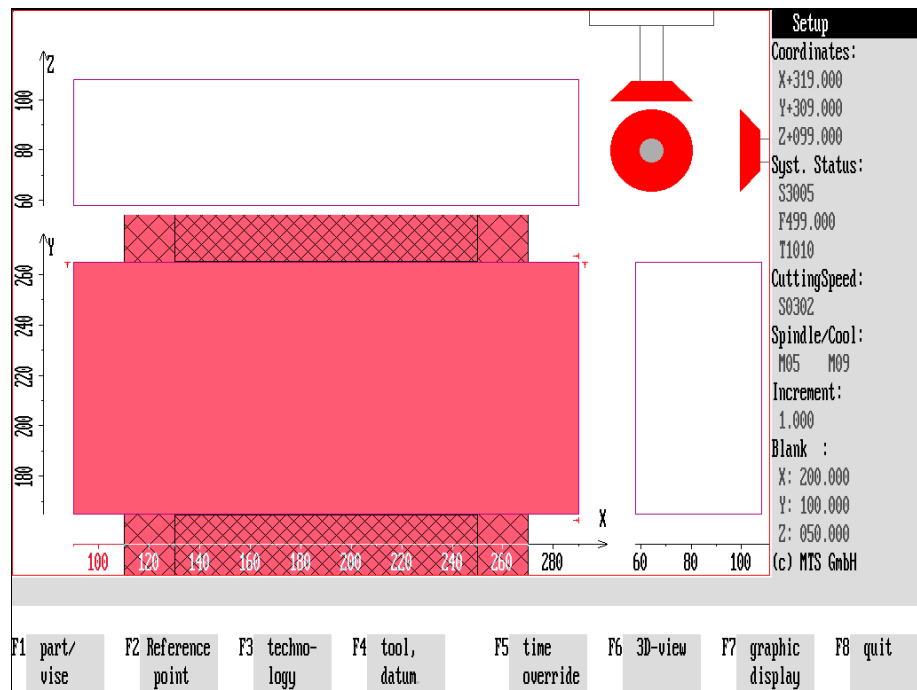
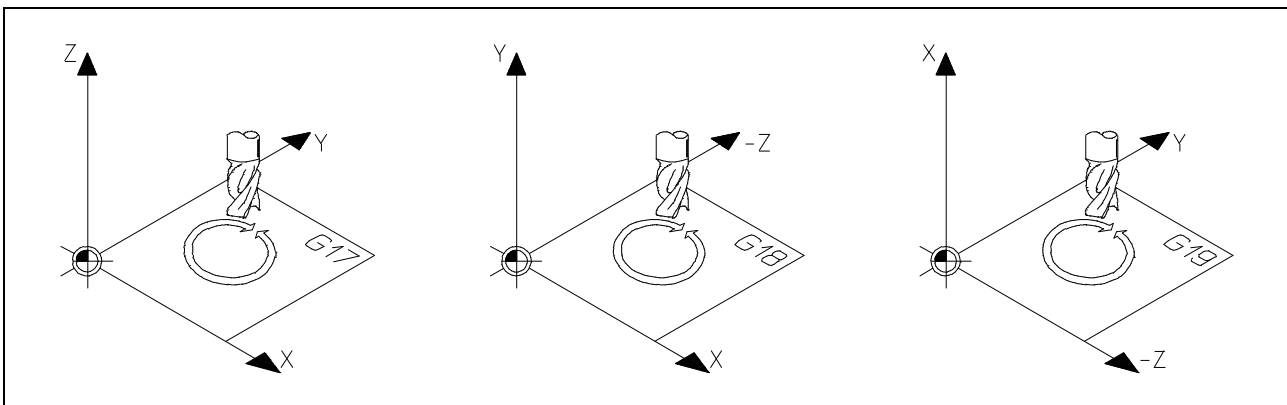
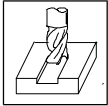


Figure 4.-4:  
CNC Milling;  
Display of a circular movement in the three machining planes G17, G18, G19:



Plane selection		longitudinal	lateral	vertical
	<b>G17</b>	X	Y	Z
	<b>G18</b>	X	Z	- Y
	<b>G19</b>	Z	Y	- X

## Screen Display

### Screen Windows

The **screen display layout** corresponds to that of the main menu: the left screen window shows a section of the work space of the machine tool with a blank or a workpart, clamping fixtures and - according to its position - a tool. Basically the current working situation is displayed graphically in this screen window.

### Note

As already described, the screen display can be modified both in turning (sectional displays, zooming etc.) and in milling (sectional and axial display etc.) (cf. [Chapter 10](#)).

### Information Column

Unlike the main menu, information on the current machine status is given in the right column:

Machine Status	CNC Turning	CNC Milling
<b>Coordinates<sup>1</sup></b>	X / Z / C current tool position [in mm]	X / Y / Z current tool position [in mm] <sup>2</sup>
<b>System Status</b>	S : Spindle speed max. <sup>3</sup> : 9000 rpm min. <sup>3</sup> : 0010 rpm F : Feedrate max. <sup>3</sup> : 50,0 mm/rev min. <sup>3</sup> : 0,001 mm/rev Txxyy : Working tool xx : Turret position yy : Compensation value reg.	S : Spindle speed max. <sup>3</sup> : 6000 rpm min. <sup>3</sup> : 0010 rpm F : Feedrate max. <sup>3</sup> : 999,0 mm/Min. min. <sup>3</sup> : 0,001 mm/Min. Txxyy : Working tool xx : Magazine position yy : Compensation value reg.
<b>Cutting Speed</b>	in m/Min.	in m/Min.
<b>Spindle/Coolant</b>	M03 - M05 / M07 - M09	M03 - M05 / M07 - M09
<b>Increment [in mm]</b>	1.0 / 0.1 / 0.01 / 0.001	1.0 / 0.1 / 0.01 / 0.001
<b>Blank [in mm]</b>	D : Diameter L : Length	X : Length <sup>4</sup> Y : Width <sup>4</sup> Z : Height <sup>4</sup>
<b>Override</b>	10 - 150 %	10 - 150 %

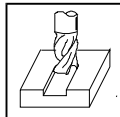
<sup>1</sup> Under consideration of tool compensation values and workpart zero

<sup>2</sup> Observe plane selection

<sup>3</sup> As to standard configuration

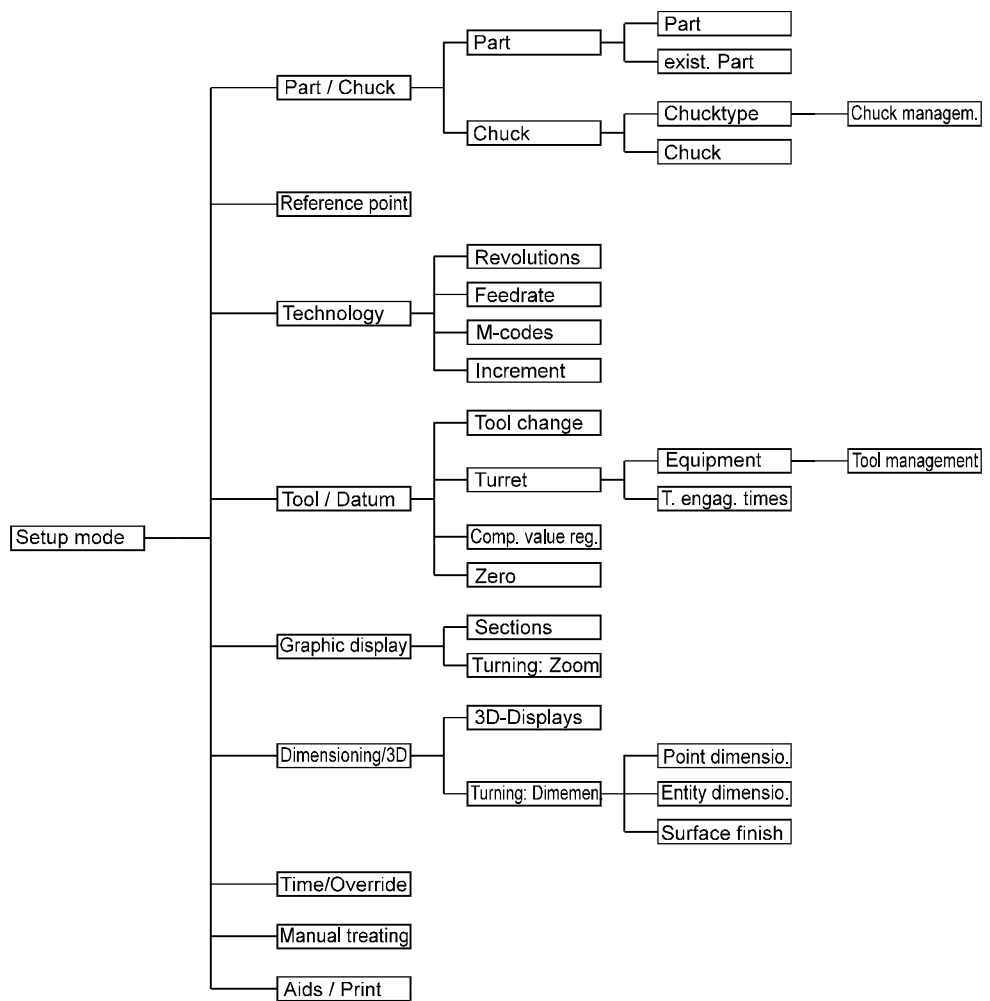
<sup>4</sup> As to plane selection G17

### CNC Milling



Note that the **plane selection** ("G17", "G18" or "G19") made in the configuration is taken into account in the coordinate data (cf. [the Table on the opposite page](#)).

Figure 4.-2:  
Overview of the  
processing possibilities  
in the Setup Mode.





## Processing

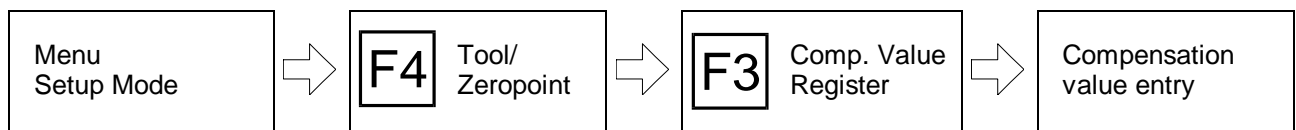
After the start, the CNC Simulator is set up as standard (provided no status file or another setup sheet is stored in the configuration, cf. configuration instructions) in accordance with the setup sheet "DSTART.DNC/FSTART.FNC" with blank, clamping, turret or magazine configuration. Naturally, the simulated machine tool can be re-equipped in the setup mode.

### Submenus

Many processing functions are available for setting up the simulated CNC machine. Please refer to the overview on the opposite page for the menu structure of the Setup Mode.

The submenus are called up, as a rule, by using the function keys:

### Example of Selection:



For the purpose of the CNC Simulator to be as identical as possible with the CNC machine tool, the CNC symbols and further icons can be displayed on the screen instead of the labelling of the function keys.

Figure 4.1-1:  
CNC Turning,  
Schematic of the  
"Workpart and  
clamping fixture  
definition" menu.

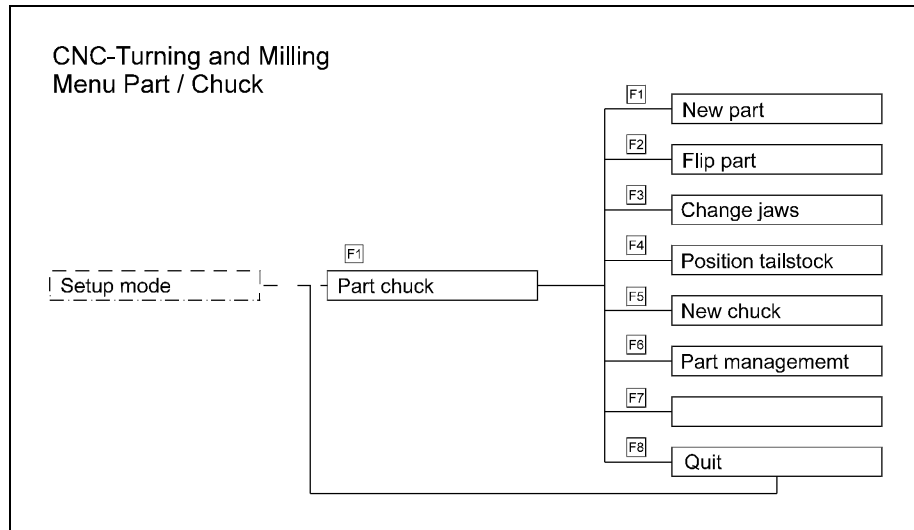


Figure 4.1-2:  
CNC Turning,  
Setup Mode;  
"Workpart and  
clamping fixture  
definition" menu.

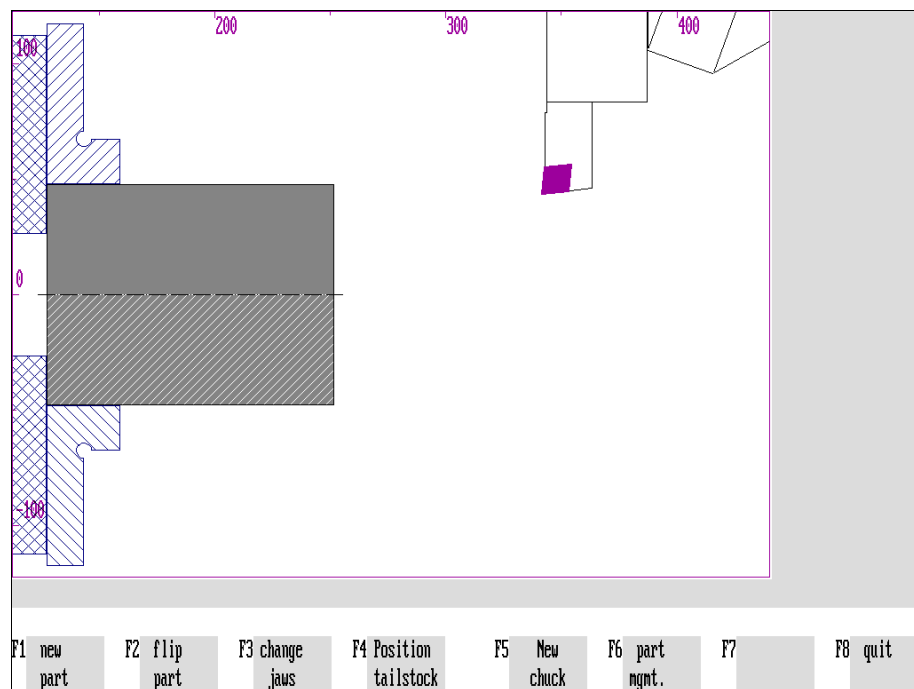
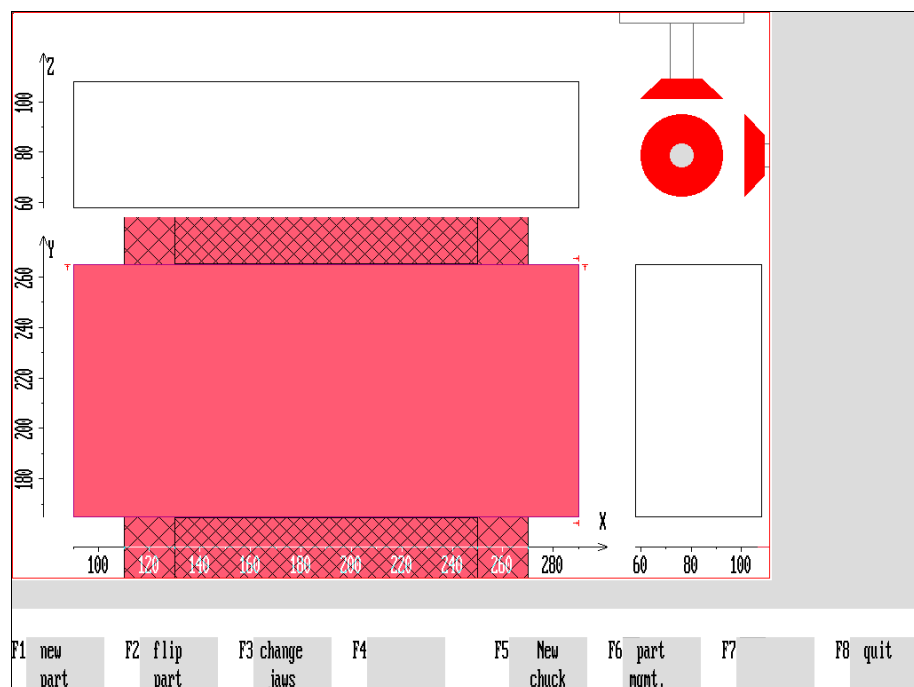
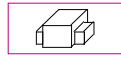
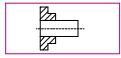


Figure 4.1-3:  
CNC Milling,  
"Workpart and  
clamping fixture  
definition" menu.



## 4.1 Blank / Workpart



The "Workpart and clamping fixture definition" is performed in a separate **submenu** called from within the Setup Mode menu with **F1**. In this menu, the boxes at the bottom margin of the screen indicate the next operation steps available:

**F1**

**New workpart:** To call up the submenu for defining the blank, press **F1**.

**F2**

**Reclamp:** To reclamp a workpart, press **F2**.

**F3**

**Change clamping fixture:** To change the clamping fixture and the clamping method, press **F3**.

**F5**

**New clamping:** If you want to clamp a workpart anew, switch with **F5** directly to the "Define clamping" menu.

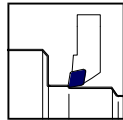
**F6**

**Workpart Manager:** To save the current workpart, call up the Workpart Manager with **F6**.

**F8**

**Return:** **F8** returns you to the main processing level of the setup mode.

### CNC Turning



In addition to the above processing possibilities, the turning CNC Simulator also offers the option of setting a tailstock:

**F4**

**Tailstock position :** With **F4** select the submenu to set or drop the tailstock.

### Note

In the menu sequence: "Determine blank" --> "Select clamping fixture" --> "Determine clamping" the execution of the work steps depending functionally on each another is prompted automatically **in the required sequence and order** .

Figure 4.1.1-1:  
CNC Turning,  
Schematic of the  
menu sequence for  
blank changing.

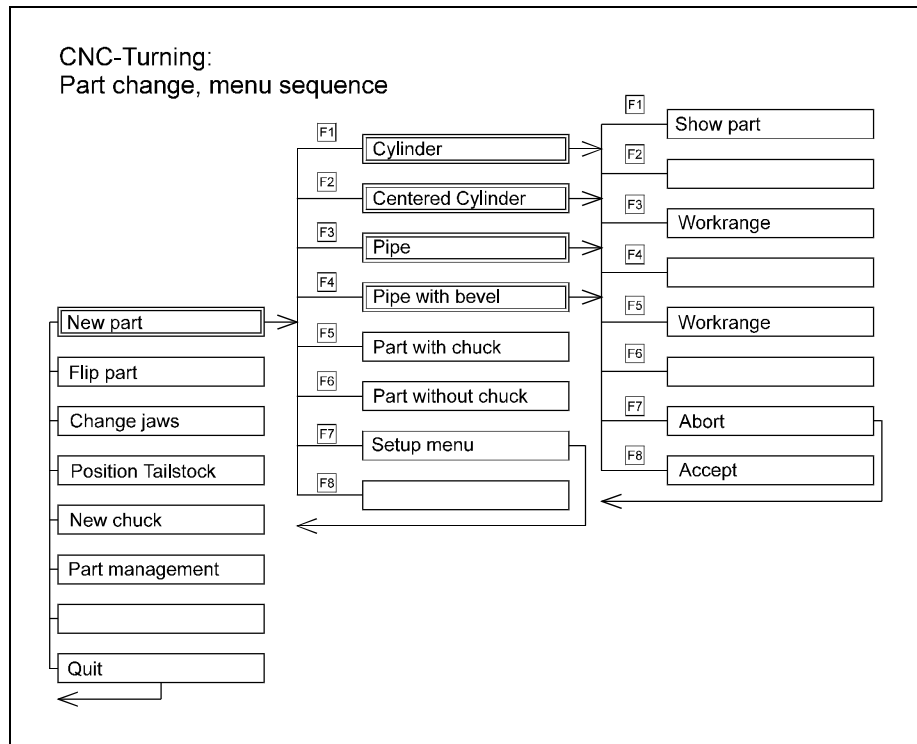
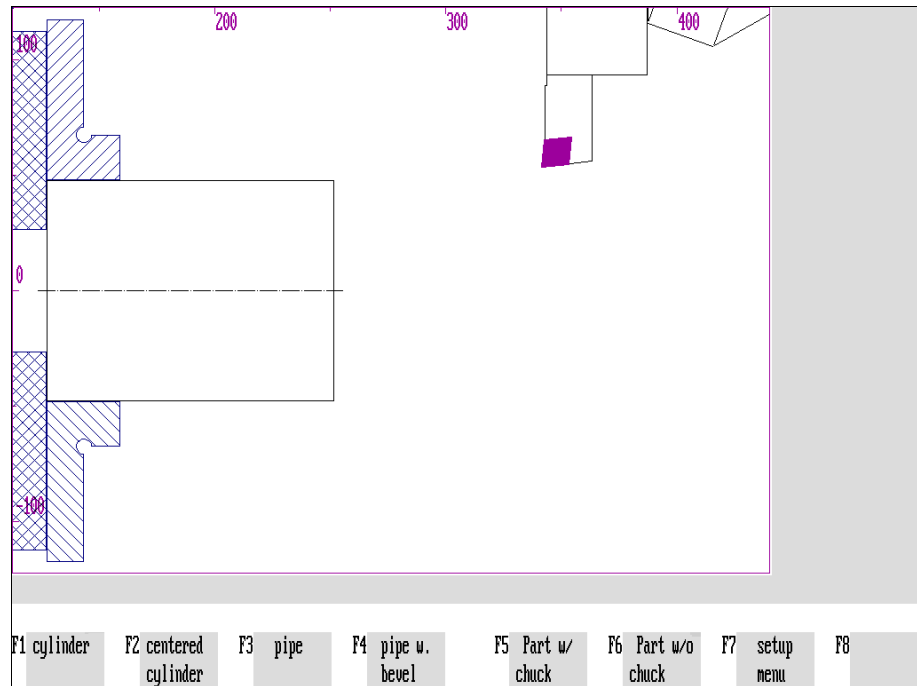
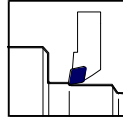


Figure 4.1.1-2:  
CNC Turning,  
Workpart and  
clamping fixture  
definition;  
"Change blank"  
menu.



### 4.1.1 CNC Turning: Blank Changing

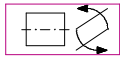
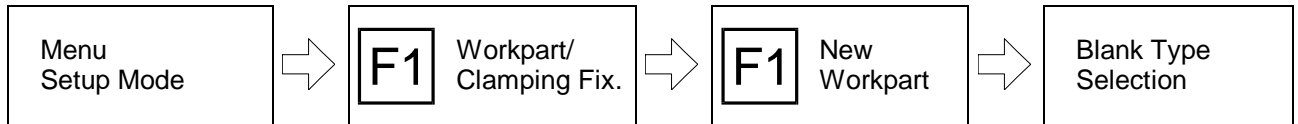


The "Blank magazine" has four different **blank types** you can equip the CNC Simulator with.

Before entering the blank dimensions, first select the required blank type, and then enter the **blank dimensions** in the following menu.

Preproduced workparts can also be read-in.

#### Selection:



Proceeding from the "Workpart and Clamping Fixture Definition" menu, you switch with **F1** ("New workpart") to the "Blank Changing" menu:

**F1**

**Cylinder:** To insert a cylinder, press **F1**.

**F2**

**Cylinder centered:** Should the cylinder be readily centered on the front face or chuck side, press **F2**.

**F3**

**Tube:** To insert a tube, press **F3**.

**F4**

**Tube with bevel:** Should the tube be readily bevelled inside, press **F4**.

**F5**

or

**F6**

**Workpart with/without clamping fixture:** Apart from determining a blank, the CNC Simulator workpart manager is able to store readily machined parts as files and to load them again with or without clamping fixture. The workpart manager saves time and work when preproduced parts (e.g. castings or forgings) are used. To insert a preproduced part, call up the Workpart Manager with **F5** or **F6**.

**F7**

or

**ESC**

**Setup Mode:** With **F7** you interrupt processing and return to the starting situation.

Figure 4.1.1-3:  
CNC Turning,  
Blank Definition;  
Entry of the  
dimensions for an  
already centered  
blank.

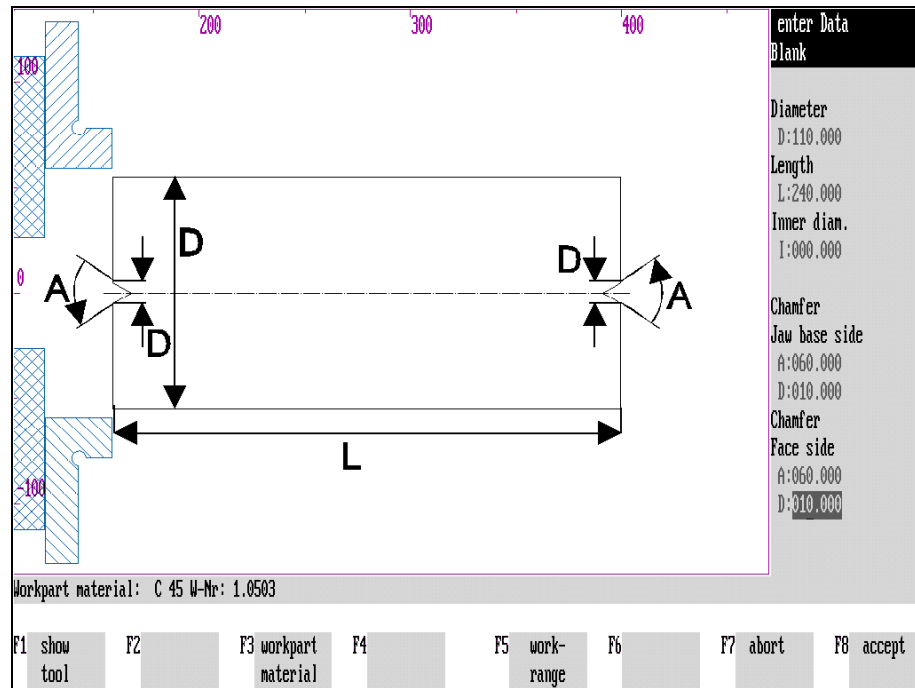
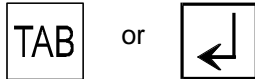


Figure 4.1.1-4:  
CNC Turning,  
Blank Dimensions.

Blank Dimensions	Cylinder	centered Cylinder	Tube	Tube w. bevel
Length	L	L	L	L
$\varnothing$ (Diameter)	D	D	D	D
$\varnothing$ , internal			I	I
Chuck side, bevel angle		A		A
Chuck side, $\varnothing$ of the bevel		D		D
Front face, bevel angle		A		A
Front face, $\varnothing$ of the bevel		D		D

## Enter Blank Dimensions

After having selected one of the described blank types, the dimensions of the required blank can be entered in the following menu in the **information column on the right side of the screen**. Proceed as follows:



You move from input field to input field with the **TAB** key (or **⇧** and **TAB**) or **⇐**.

[Value statement]

Enter the desired dimensions in the input fields.

The following menu functions are also available:



**Display workpart:** To display now the new blank in the dimensions entered, press **F1**.



**Material:** Under **F3** "Material", enter the material of the blank and confirm with **⇐**.



**Work space:** To display the entire work space, press **F5**. You can also switch back to the standard display with **F5**.



or **ESC**

**Interrupt:** Interrupt processing with **F7** and return to the starting situation.



**Accept:** After you have entered all values, confirm your entry with **F8**.

**The next menu for selecting the clamping fixture is then loaded automatically!**

Figure 4.1.2-1:  
CNC Milling,  
Schematic of the  
menu sequence for  
blank changing.

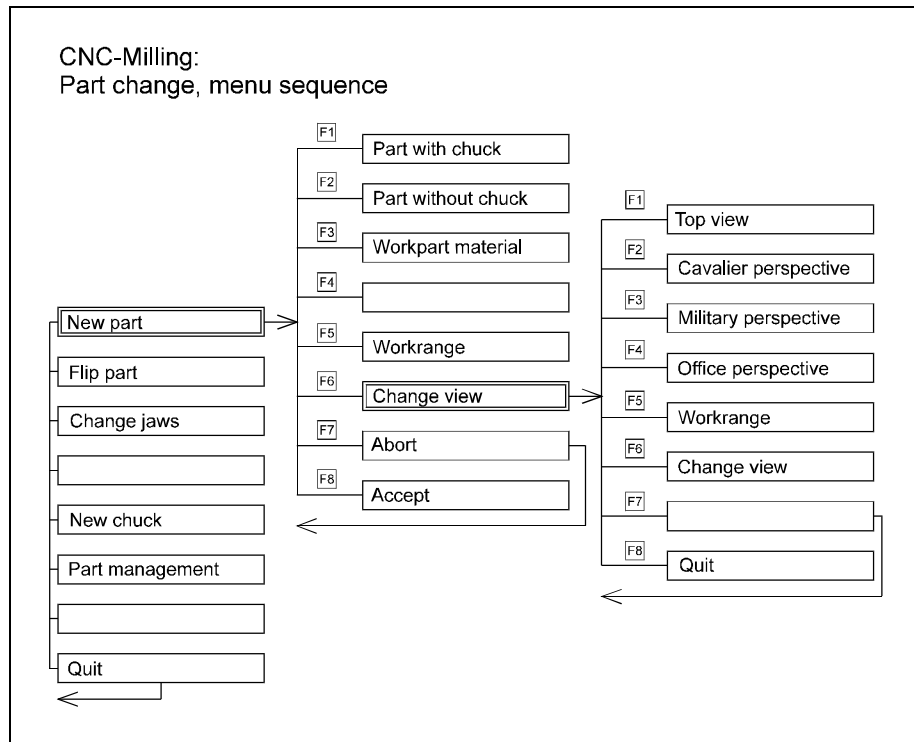
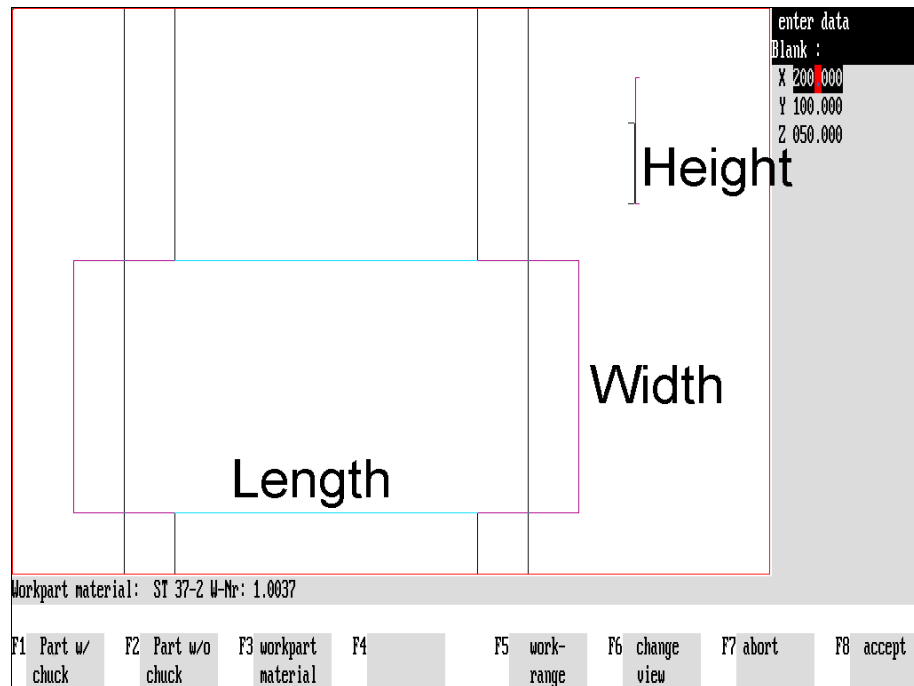


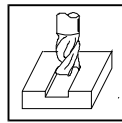
Figure 4.1.2-2:  
CNC Milling,  
Workpart and  
clamping fixture  
definition;  
Entry of the blank  
dimensions.



Blank Dimensions	G17	G18	G19
Length	X	X	Z
Width	Y	Z	Y
Height	Z	Y	X

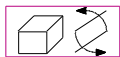
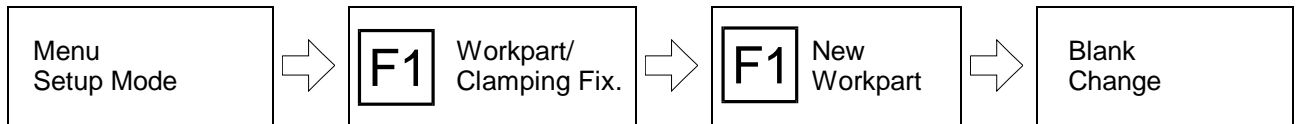


## 4.1.2 CNC Milling: Blank Changing



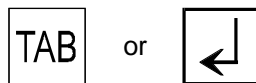
A new blank is selected for the milling CNC Simulator by **entering the blank dimensions** (in mm).  
Preproduced workparts can be read-in as well.

### Selection:



Proceeding from the "Workpart and Clamping Fixture Definition" menu, you change to the "Blank Change" menu with **F1** ("New Workpart"). Then enter the dimensions of the blank required in the **information column on the right side of the screen**:

### Specification of the Blank Dimensions



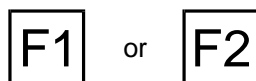
or

You move from input field to input field with the **TAB** key (or **△** and **TAB**) or **←**.

[Value Statement]

In the input fields you enter the desired dimensions (in mm): **length**, **width** and **height**. Note the plane selection specified in the configuration ("G17", "G18" or "G19"). It needs to be taken into account when entering the dimensions ([cf. overview opposite page](#))!

Furthermore, the following menu functions are available:



or

**Workpart with/without clamping fixture**: Apart from defining a blank, the CNC Simulator workpart manager is able to store ready machined parts and to load them again with or without clamping fixture. The workpart manager saves time when preproduced parts (e.g. castings or forgings) are used. To insert a preproduced part, call up the workpart manager with **F1** or **F2**.



**Material**: In this subsequent menu you specify the material of the blank/workpart and confirm it with **←**.



**Work space**: To display the entire work space, press **F5**. You also switch back to the standard display with **F5**.



**Change view**: You can change the view onto the workpart with **F6**.



or



**Cancel**: You cancel processing with **F7** or **ESC** and return to the starting situation.



**Confirm**: After having entered all values, confirm your entries with **F8**.  
**The next menu for selecting the clamping method is then loaded automatically!**



**You can also change the viewing perspective of the workpart while determining the blank and clamping method. This display option remains active only during the current processing.**

Figure 4.1.2-3:  
CNC Milling;  
Blank and clamping  
method definition;  
Changing the  
perspective onto the  
workpart: in this  
example the "military  
perspective" is  
selected.

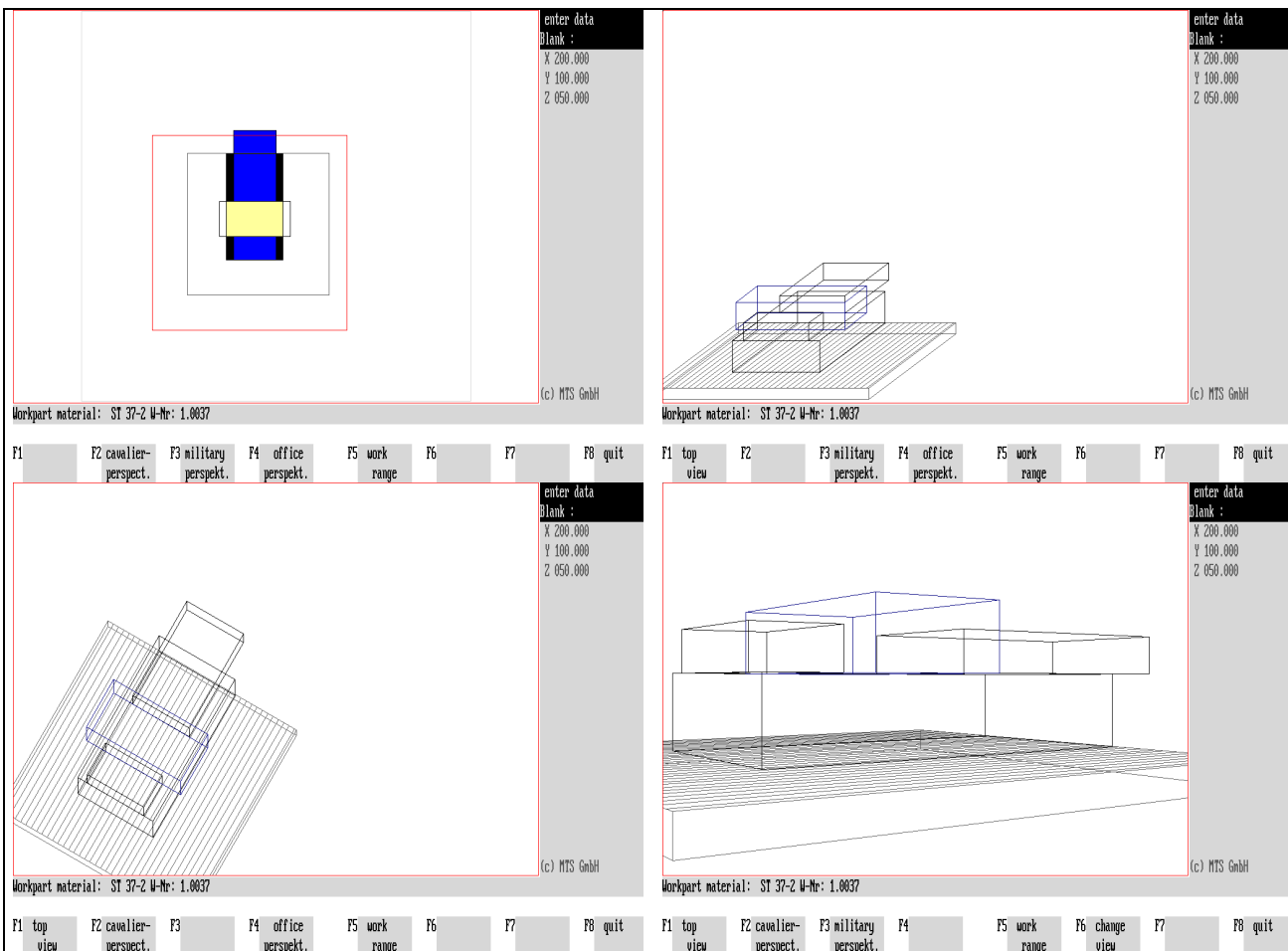
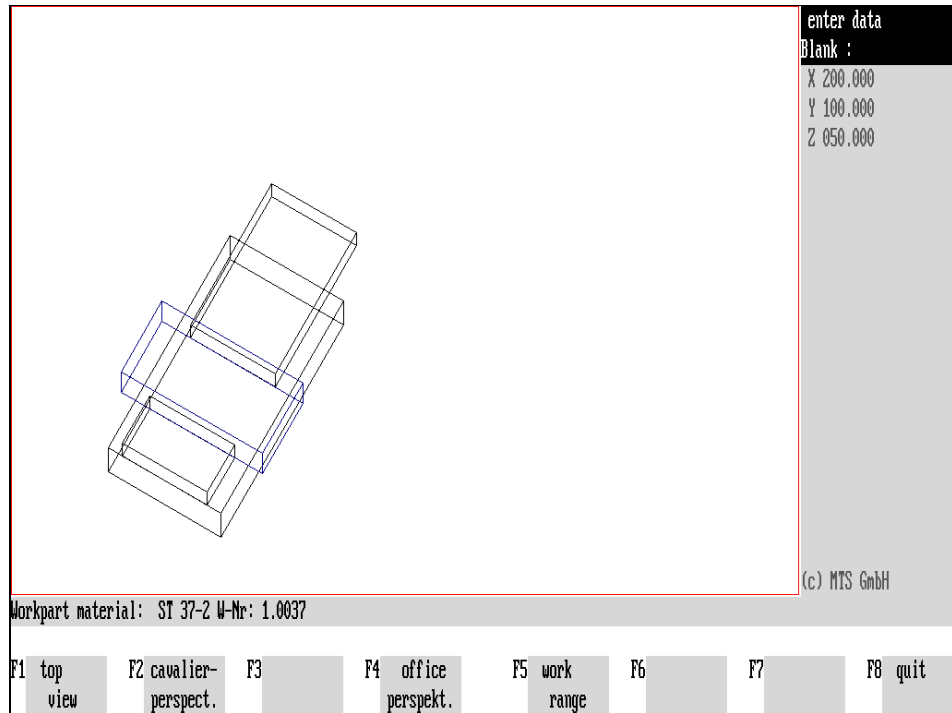


Figure 4.1.2-4:  
CNC Milling, different views of the workpart in the blank and clamping method definition:  
Above left: top view of the entire work space; above right: cavalier perspective; below left: military perspective; below right: office perspective.

## Perspective Views

PER-  
SPEKTIVE

To illustrate the clamping situation (helpful especially in modular clamping), you can display different views of it. The "Change View" menu is called from the "Blank Change" menu with **F6**. You can then choose between the following views:

**F1**

**Top view:** If you have selected another perspective, switch back to the standard display: top view onto the workpart with **F1**.

**F2**

**Cavalier perspective:** The "cavalier perspective" shows the workpart and clamping fixture from a slightly elevated side-view perspective: the work space of the machine tool being in the front sideways..

**F3**

**Military perspective:** The "military perspective" shows the workpart and clamping fixture from a highly elevated top-view perspective: the work space of the machine tool being in the front laterally.

**F4**

**Office perspective:** The "office perspective" shows the front perspective elevation of the workpart and clamping fixture. This display view can be further modified with the "Change Viewpoint" function (cf. below).

**F5**

**Work space:** You display the dimensions of the work space (in X/Y direction according to G17) in the screen with this function. The work space is highlighted in color and cross-hatched.

To cancel this type of display, press **F5** again.

**F6**

**Change viewpoint:** This function is activated only if office perspective has been selected. You can then modify this display with **F6**.

**F8**

**Return:** After having selected the perspective, **F8** returns you to the starting situation.

### Note

The currently active display type is highlighted in the function key menu.

## Change Viewing Point

Only in case you have first selected the **office perspective** it is possible to modify further the screen display with **F6** "Change Viewpoint":

**F1**

or

**F2**

**Turn to the left/right:** You shift the viewing position to the left or right with **F1** or **F2** respectively.

**F3**

or

**F4**

**Raise/lower view:** You shift the viewing position higher or lower with **F3** or **F4** respectively.

**F5**

or

**F6**

**Increase/reduce distance:** You shift the viewing position closer to the workpart or increase the viewing distance to the workpart with **F5** or **F6** respectively.

**F8**

**Return:** **F8** returns you to the starting position.

### Note

The terms "cavalier perspective", "office perspective" and "military perspective" are technical terms from projective geometry. The cavalier perspective and the military perspective are parallel projections with the projection plane parallel to the X/Z plane (cavalier perspective) or X/Y plane (military perspective).

The office perspective is a central projection (defined by the viewing point and the projection plane), rendering closest the natural picture of the object. In the office perspective, the viewing distance and perspective of the observer can be changed with the cursor keys **F1** - **F4**.



Perspective displays are possible for blank and clamping method definition. Preproduced workparts can be shown only in the top view.

Figure 4.1.2-1:  
CNC Turning,  
Schematic of the  
menu sequence for the  
Workpart Manager

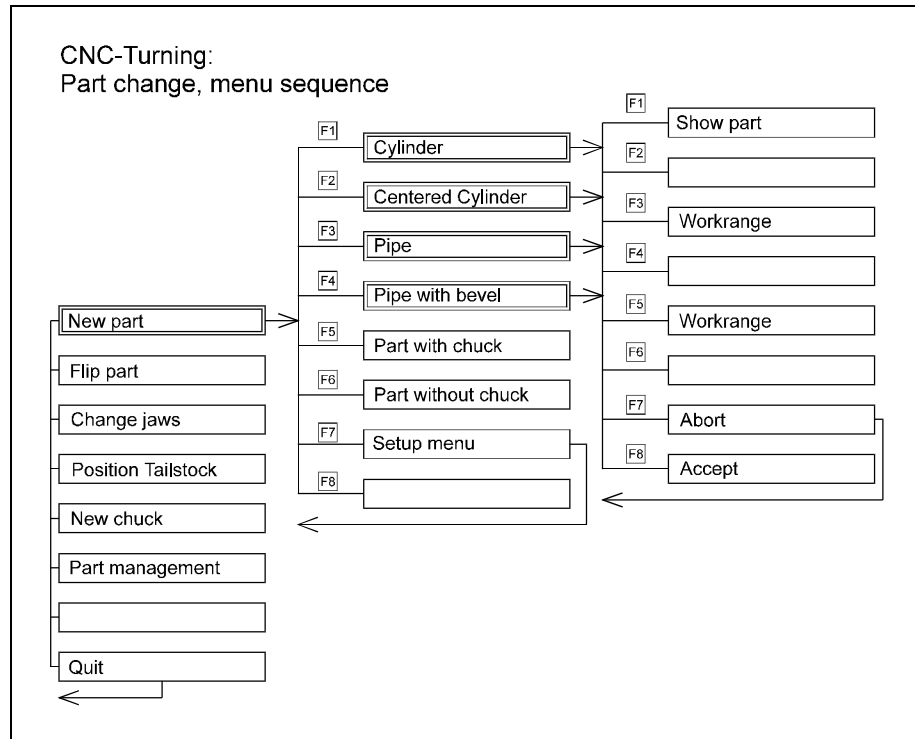


Figure 4.1.2-1:  
CNC Milling,  
Schematic of the  
menu sequence for the  
Workpart Manager.

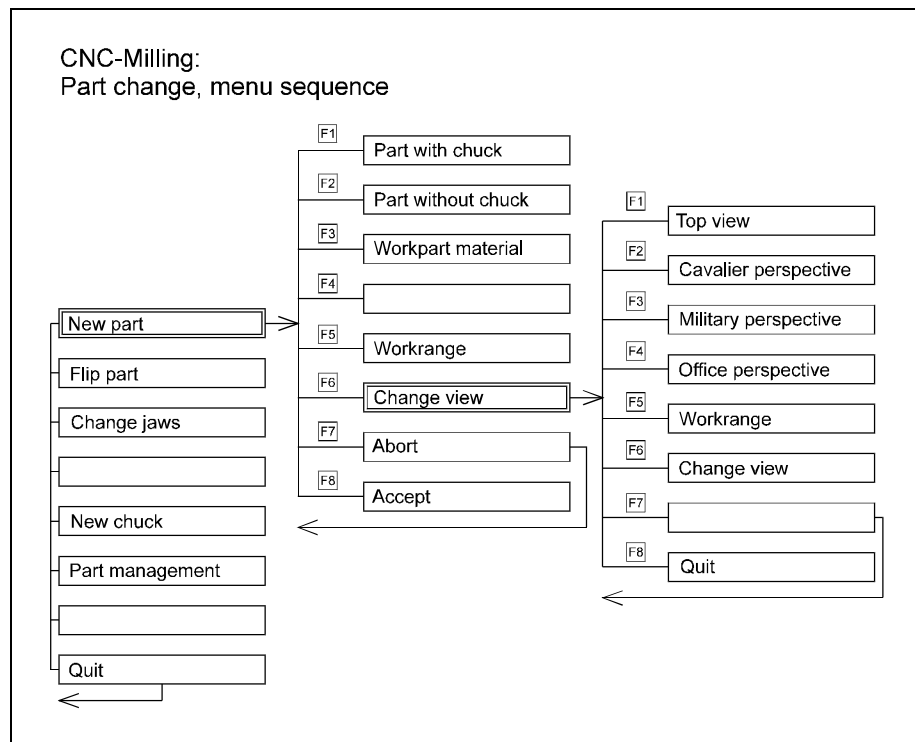
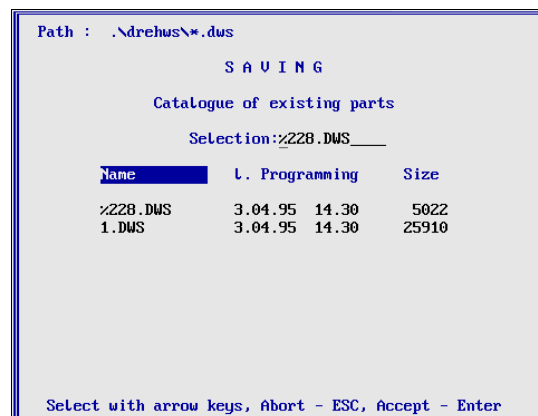


Figure 4.1.3-3/4:  
CNC Turning,  
Workpart Manager;  
File information for  
loading a reproduced  
workpart.

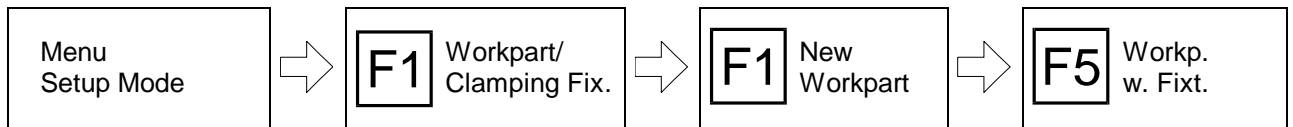


## 4.1.3 Workpart Manager

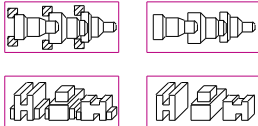
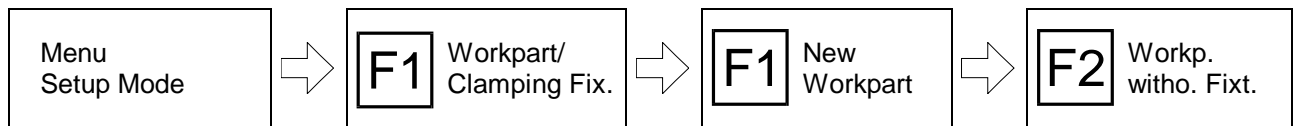
### Inserting an Existing Workpart

In addition to the blank definition, the CNC Simulator includes a workpart manager to store already machined parts in files and to load them again with or without clamping fixture. The workpart manager is a great support when working with preproduced parts (e.g. castings or forgings).

#### Selection Example CNC Turning:



#### Selection Example CNC Milling:



To insert a preproduced workpart, select the function "Workpart with/without clamping fixture" with **F5** or **F6** (CNC Turning) or with **F1** or **F2** (CNC Milling) in the "Blank Change" menu.

The **file selection window** is then loaded for the selection of the workpart. Enter the file name of the desired workpart in its input field or search for the desired part with the cursor keys (please refer to [Chapter 2](#) for handling of the file selection window).



You cancel selection of a workpart with **ESC**.



You confirm the selected workpart with **↵**.



**To avoid collision, be sure to leave enough distance between the tool (-holder) and the clamping fixture when inserting a new workpart.**

Figure 4.1.1-4:  
CNC Turning,  
Workpart Manager;  
Save or delete a  
workpart

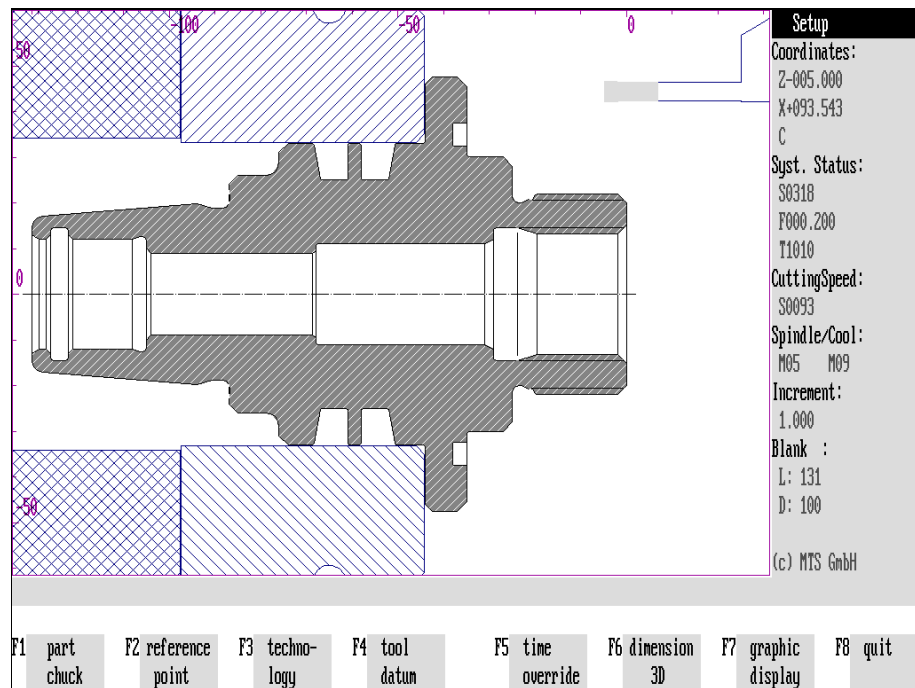
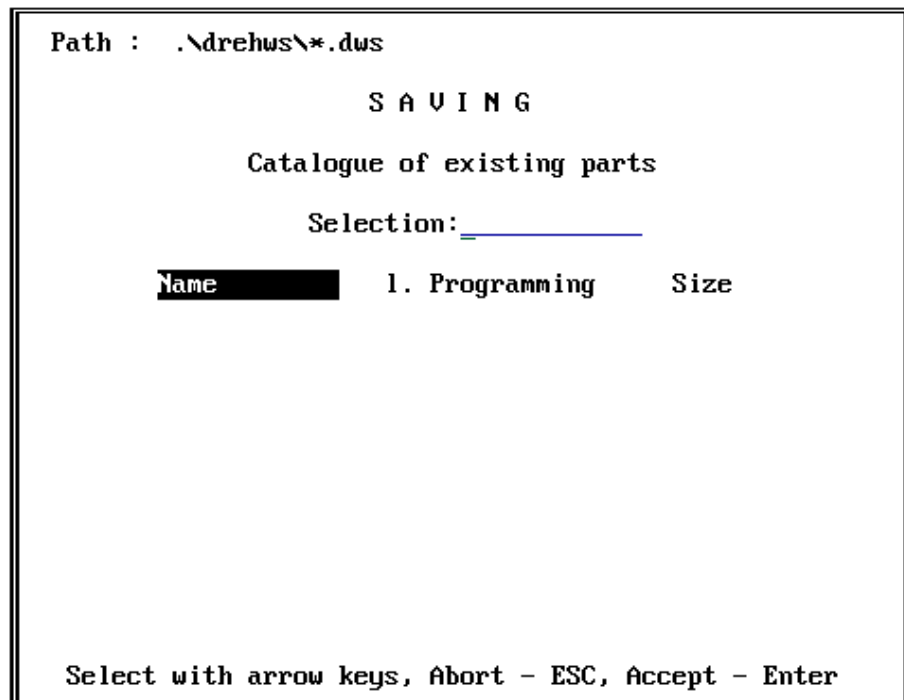
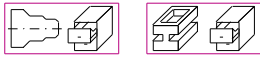


Figure 4.1.3-5:  
CNC Turning,  
Workpart Manager;  
File information to  
save a preproduced  
workpart.

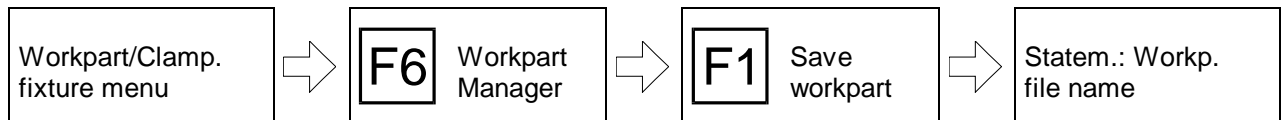


## Saving/Deleting a Preproduced Workpart



To save or delete a preproduced workpart, select the function "Workpart Manager" with **F6** in the "Blank and Clamping Method Definition" menu.

### Selection Example CNC Milling:



This then loads a submenu in which you state whether the workpart should be saved or an already existing one deleted:

**F1**

**Save workpart:** With **F1** you save the current workpart as a file.

**F2**

**Delete workpart:** With **F2** you select the existing workpart file to delete it.

**F8**

**Return:** **F8** returns you to the starting situation.

After having chosen the processing function, the **file selection window** is loaded. Enter the file name of the workpart to be saved or deleted (cf. above, [Chapter 2](#) for handling the file selection window) in its input field.

**ESC**

With **ESC** you cancel the selection of a workpart.

**↵**

With **↵** you confirm the selected workpart.

### Control Enquiry

After you have once pressed the key to delete a file an additional **control query** is displayed before a workpart file is deleted or overwritten (this would be the case if the name entered for the file to be saved is identical with an already existing file name), to enable you to cancel the function in time.

**ESC**

With **ESC** you cancel processing.

**↵**

With **↵** you confirm the control query.

You then return to the starting position.

Figure 4.2-1:  
CNC Turning,  
Schematic of the  
"Workpart and  
Clamping Fixture  
Definition" menu.

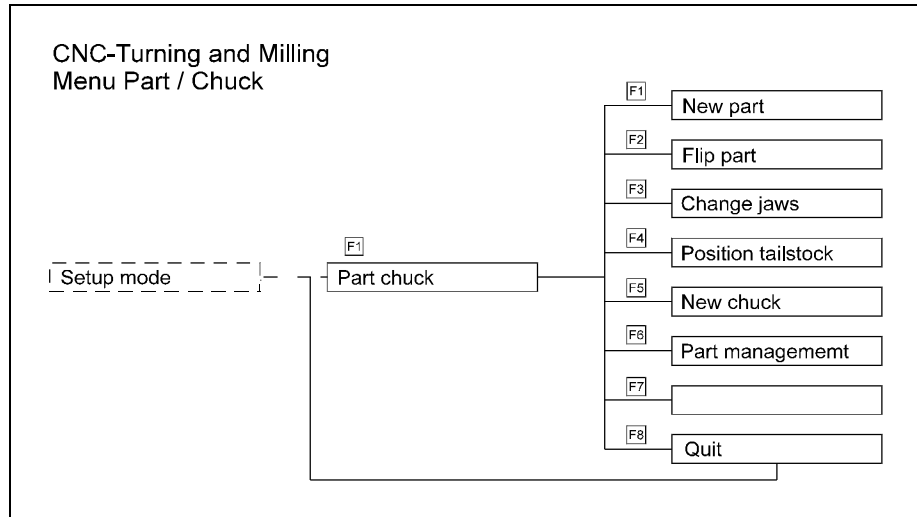


Figure 4.2-2:  
CNC Turning,  
clamping between  
centers.

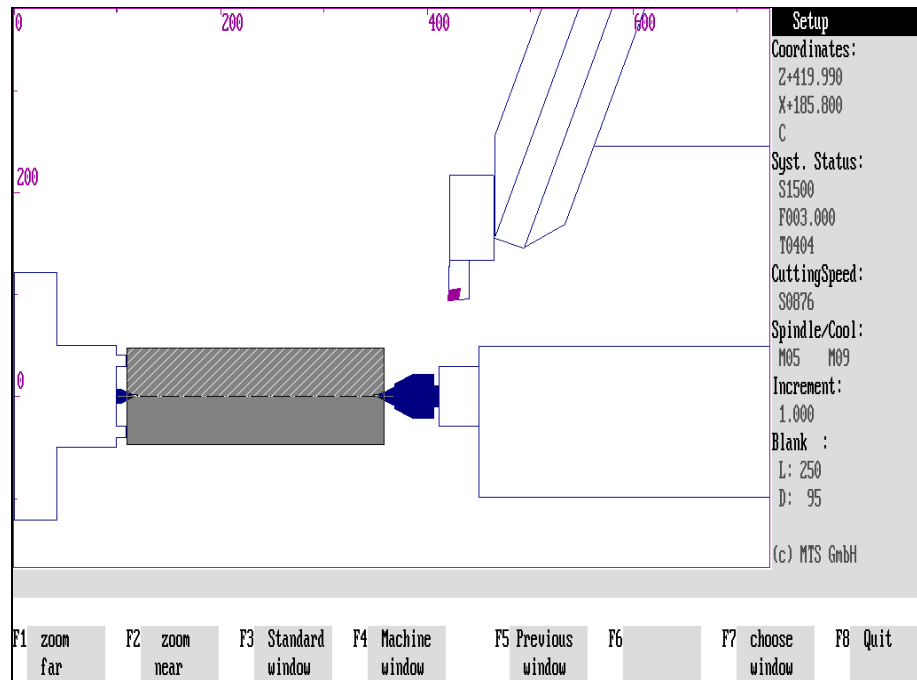
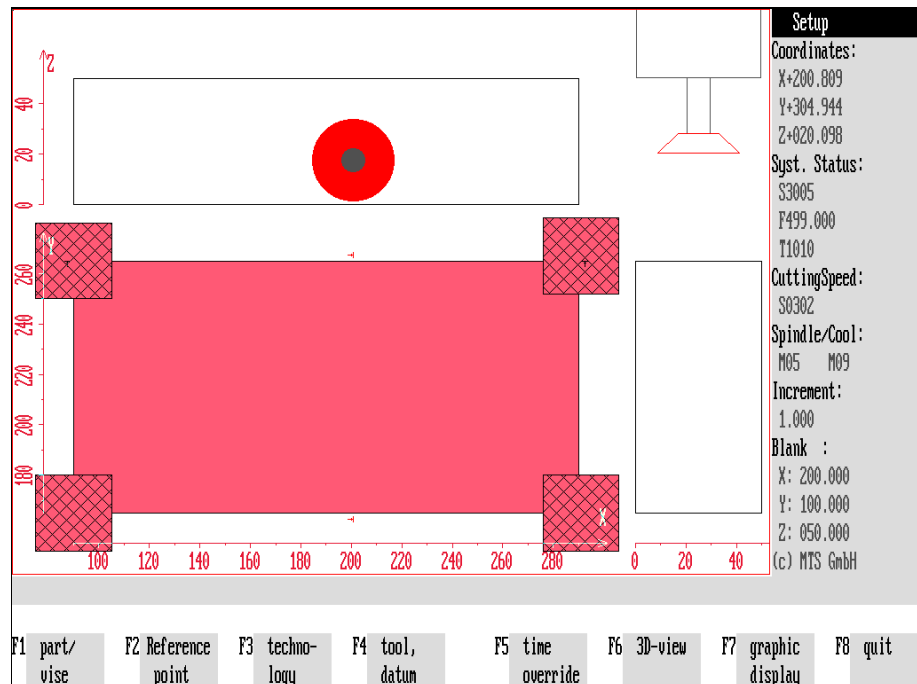


Figure 4.2-3:  
CNC Milling,  
modular clamping.





## 4.2 Clamping Fixtures

The clamping fixtures (clamping devices) required for workpart clamping depend on the type of machining and on the geometry of the workpart. MTS Simulators permit most of the current customary clamping situations used in production. The version 5 has its own extensive clamping fixture manager including several options for workpart clamping.



The starting point for the selection of the clamping fixture and specification of the clamping method is the "Workpart and Clamping Fixture Definition" menu. Proceeding from the Setup Mode menu, it is called with **F1**:

**F1**

**New workpart:** With **F1** you call the submenu to define the blank.

**F2**

**Reclamp:** To reclamp a workpart, press **F2**.

**F3**

**Change clamping fixture:** To change the clamping fixture and the clamping method, press **F3**.

**F5**

**New clamping:** If you want to change only the clamping method of a workpart, switch with **F5** directly to the "Define Clamping Method" menu.

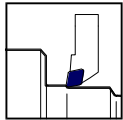
**F6**

**Workpart Manager:** To save the current workpart, call the workpart manager with **F6**.

**F8**

**Return:** **F8** returns you to the main processing level of the Setup mode.

### CNC Turning



In addition to the above listed processing options, the turning CNC Simulator also offers the use of the tailstock:

**F4**

**Tailstock position:** With **F4** you call the submenu to set or drop the tailstock.

### Procedure

In general, there are two processing paths to determine the clamping method:

- 1.) If you define a new blank, the menus for "Selecting the clamping fixture" and "Defining the clamping method" are loaded **in direct sequence** and you can perform the necessary work steps consecutively.
- 2.) On the other hand, it is also possible to call up the selection of the clamping fixtures or the definition of the clamping method **separately**. In this case, the clamping fixture manager is activated to load or redefine individual clamping elements for turning after selection of the clamping fixture.

Figure 4.2.1-1:  
CNC Turning,  
Schematic of the menu  
sequence for selecting  
the clamping fixture  
and defining the  
clamping method after  
a blank change.

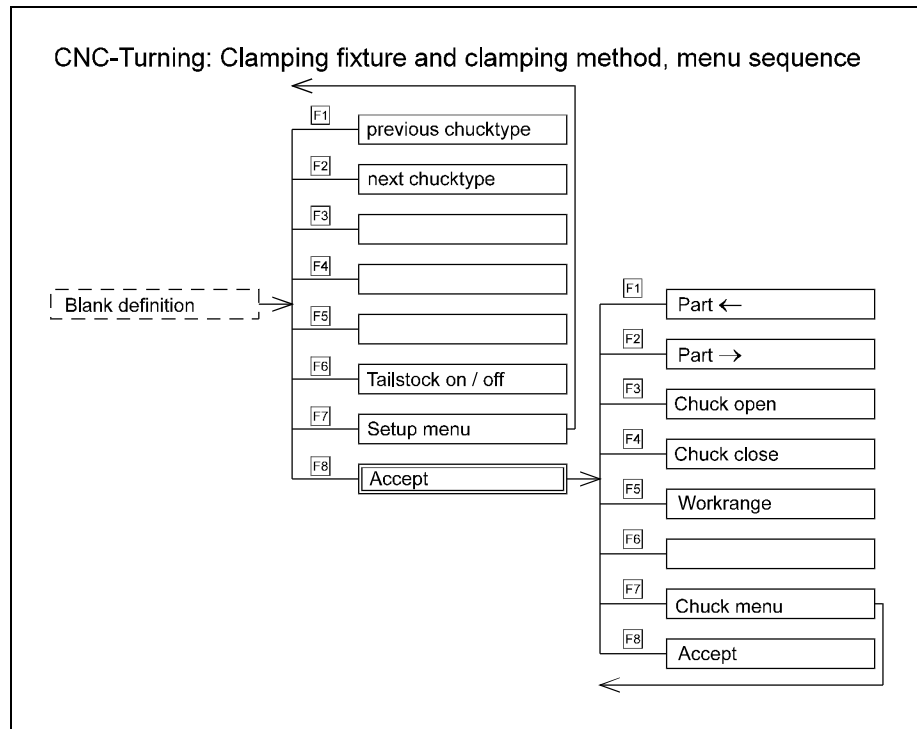
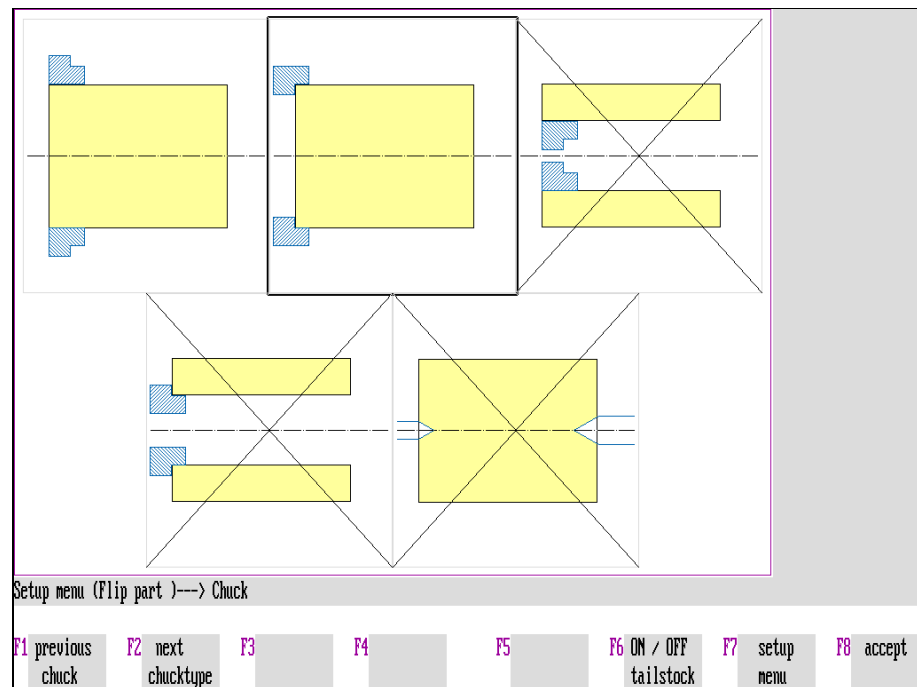
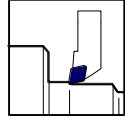


Figure 4.2.1-2:  
CNC Turning,  
workpart and clamping  
definition;  
"Clamping Fixture  
Selection" menu.



## 4.2.1 CNC Turning: Clamping Fixtures

### Selection of Clamping Fixtures



In the Simulator for Turning, the clamping system consists of a lathe chuck with stepped jaws (multiple jaw chuck); as an alternative, clamping between centers with front driver and tool arbor/lathe centers is also possible.

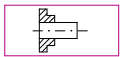
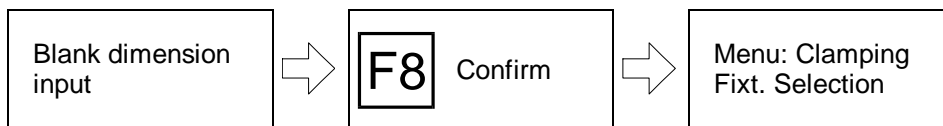
All clamping elements can be defined and assembled individually in the clamping fixture manager.

The user specifies the geometrical dimensions of the tailstock and of the spindle in the configuration.

#### Selection:



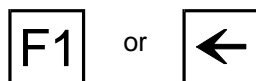
#### Automatic Menu Guidance:




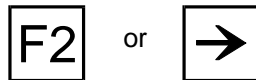
The menu for clamping fixture selection is loaded either automatically after blank definition or it is loaded proceeding from the "Workpart/Clamping Fixture" menu with [F3] "Change Clamping Fixture".


The clamping fixture selection is supported by five displayed graphic presentations of the types of clamping.

Depending on the blank/workpart inserted, those **clamping methods which are not available for workpart clamping are crossed out** in the display. Select the desired clamping method by marking the corresponding presentation with the **color-highlighted frame**:



**Previous clamping type:** Move the frame counterclockwise with [F1] or the cursor key .

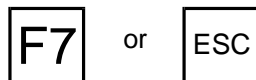


**Next clamping type:** Move the frame clockwise with [F2] or the cursor key .

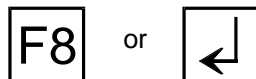


**With/without tailstock:** If you intend to set the tailstock as well, press [F6]. In this case a lathe center is additionally displayed in the front area of the clamping presentations.

To drop the tailstock, press [F6] again.



**Return:** With [F7] or [ESC] you cancel processing and return to the starting situation.



**Accept:** You confirm the marked type of clamping with [F8] or .



**Subsequently either the menu for defining the clamping method or the Clamping Fixture Manager is loaded automatically.**

Figure 4.2.1-3:  
CNC Turning,  
Clamping Fixture  
Manager;  
Schematic of the  
menu sequence for  
changing a clamping  
element.

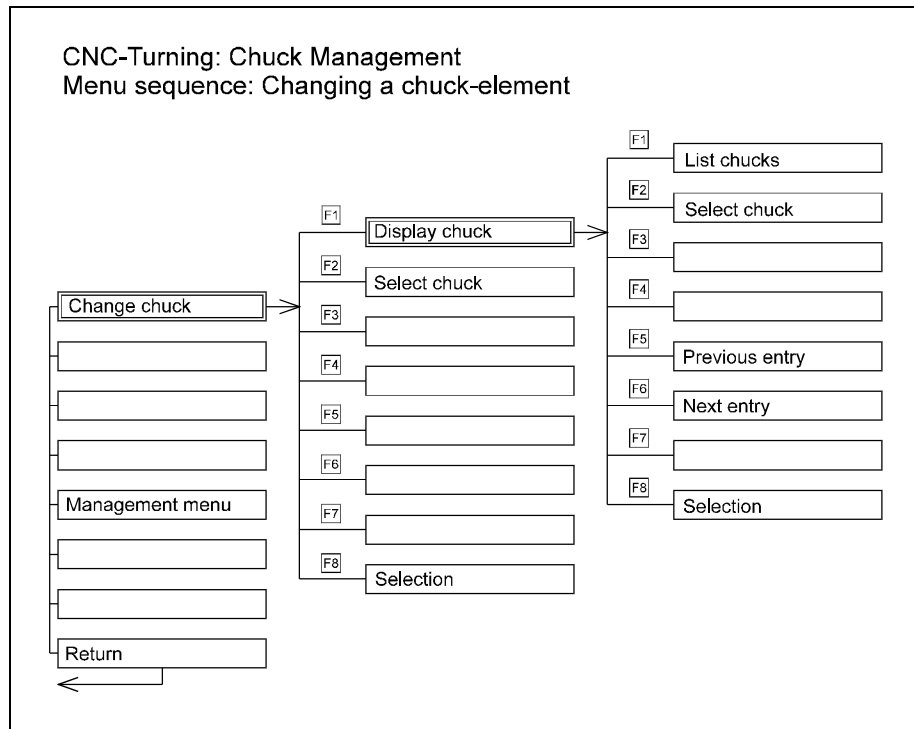
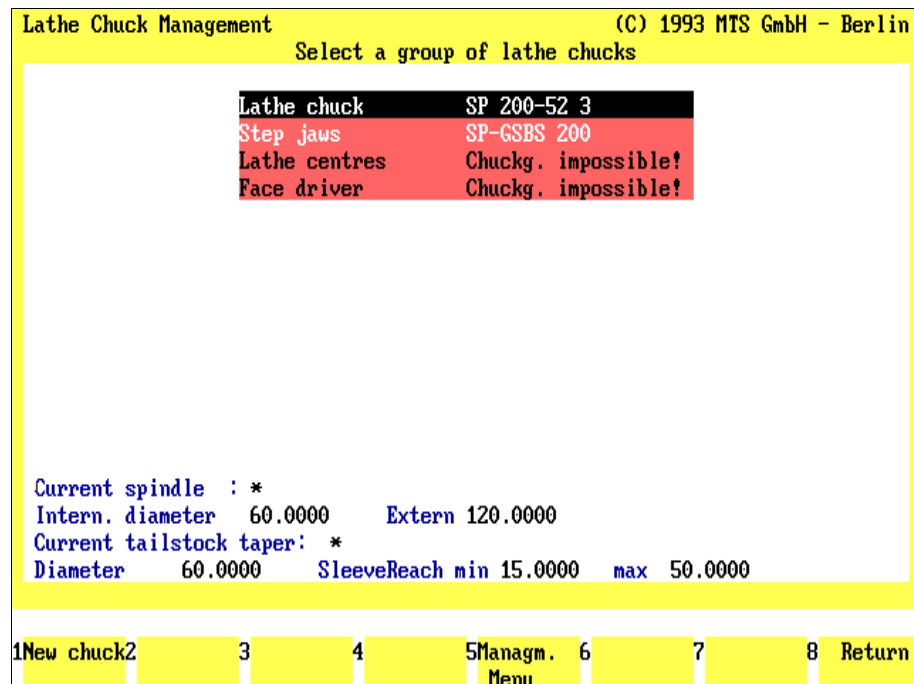


Figure 4.2.1-4:  
CNC Turning,  
clamping fixture  
manager;  
Main menu.

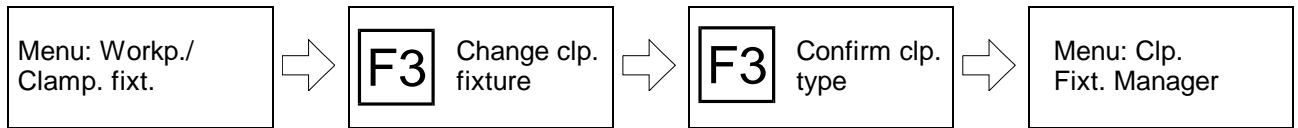


## Changing a Clamping Element



The turning Simulator has an independent clamping fixture manager for defining and replacing of individual clamping elements.

### Selection:



After having selected the clamping type, the clamping element manager is loaded automatically. **However, if you have already defined a blank, this menu is skipped.** In this case select the clamping fixture manager with **[F3]** via the menu item "Change Clamping Fixture".



The clamping fixture manager is described in this chapter only as far as necessary for changing clamping fixtures. Later on, it is discussed in detail..

### Clamping Fixture Manager

The menu structure of the clamping fixture manager is similar to those of the tool manager (see below, p. 89):

- 1.) The upper screen area shows auxiliary graphics, information and input fields.  
In the main menu the clamping fixture groups are displayed in this area, subdivided into the groups of **currently used clamping element** (or: "**non-inserted**") and of "**non-insertable clamping elements**" (due to the selected type of clamping).  
In addition, information on spindle and tailstock/sleeve is given in the bottom part of the screen.
- 2.) The function keys are displayed in the lower screen margin.

**[F1]**

**Change clamping fixture:** With **[F1]** you can use a new clamping element of the marked clamping fixture group.

**[F5]**

**Management menu:** To define or to change or to delete a clamping element, press **[F5]**. To avoid unauthorized interventions in the clamping fixture manager, this function can be protected with a "password" (cf. Configuration Instructions).

**[F8]**

**Return:** With **[F8]** you confirm the listed combination of the clamping elements and switch to the menu for defining the clamping method.

### Procedure

To insert a new clamping element, first select the applicable clamping fixture group and then switch to the submenu of this group:

**↓**

or

**↑**

With the cursor keys **↑** or **↓** you mark the desired clamping fixture group. In this case those clamping elements cannot be selected, of course, which are not available for the specified clamping element.

**[F1]**

**Change clamping fixture:** To pick up a new clamping element from this group, switch with **[F1]** onto the selection menu of this clamping fixture group.

**ESC**

You cancel processing and return to the starting situation with **[ESC]**.

Figure 4.2.1-5:  
CNC Turning,  
Clamping Fixture  
Manager,  
stepped jaws;  
Selection of a  
clamping element.

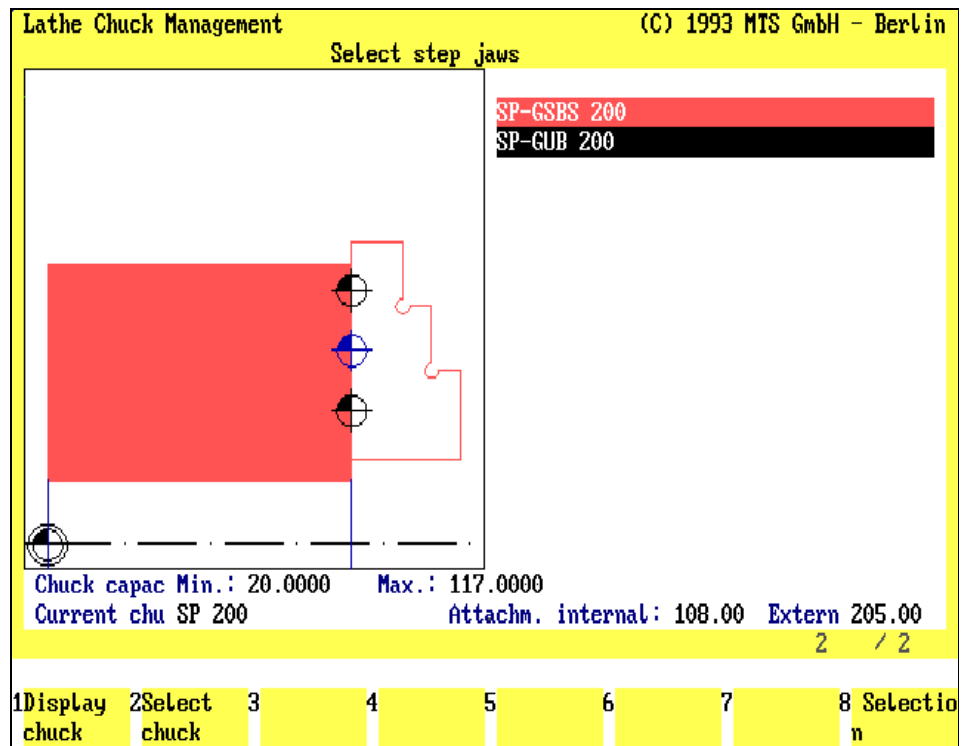
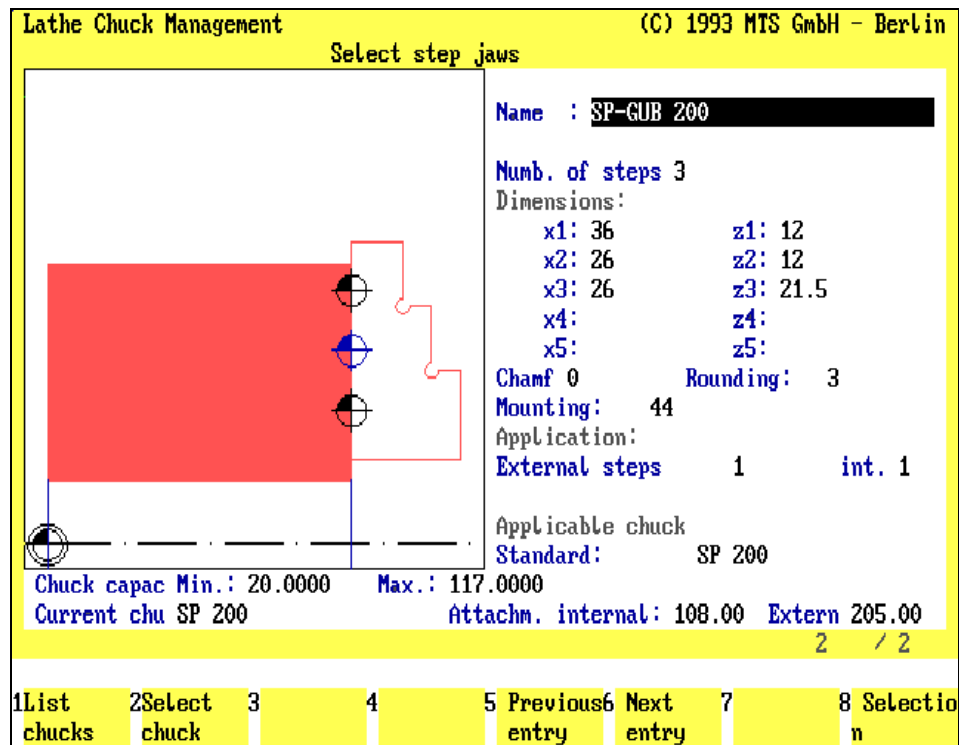


Figure 4.2.1-6:  
CNC Turning,  
Clamping Fixture  
Manager,  
stepped jaws;  
Display clamping  
element



## Overview

In the selection menu of the clamping fixture manager, a graphic presentation of the currently selected clamping element is shown next to the information field (or input fields) (cf. Figure) for a better overview. Furthermore, the names of the available clamping elements are listed.



**Please note that after having selected a lathe chuck only the acceptable stepped jaws are offered.**

The following operation functions are available:

F1

**Show clamping fixture:** With **F1** you call up information on the currently selected chuck/clamping element.

F2

**Select clamping fixture:** To select from the existing clamping elements those meeting the geometrical criteria, you can have these searched for and listed. A detailed description of this function is given in the following chapter.

F8

**Confirm:** With **F8** you accept the selected clamping element.

ESC

With **ESC** you cancel processing and switch back to the starting position.

## Procedure

To select a new clamping element, first mark the desired element in the overview and then confirm your choice:



or



With the cursor keys **↓** or **↑** (in extensive lists you may also use **Page↓** or **Page↑**), you mark the clamping element you want to select.

F8

**Selection:** With **F8** you confirm the selected clamping element; then you return to the main menu of the Clamping Fixture Manager.

## Individual Information

The selection of the clamping element from the listed overview can alternatively be done by calling detailed information on each individual clamping element: This display function is available in the overview of the Clamping Fixture Manager for checking the geometrical dimensions of the clamping element.

F1

**Show clamping fixture:** Detailed information is called with **F1**.

## Procedure:

A clamping element is selected from the "individual display" in a similar way as described above:

F1

**Element list window:** You return to the overview of all clamping elements with **F1**.

F2

**Make element list:** To select from the list of all clamping elements the clamping elements meeting the geometrical criteria you can have these searched for and listed as a new list.

F5

**Previous entry:** With **F5**, the geometrical data of the listed clamping element preceding the currently marked one are displayed.

F6

**Next entry:** With **F6**, the geometrical data of the listed clamping element following the currently marked one are displayed.

F8

**Selection:** With **F8** you select the currently displayed clamping element.

ESC

With **ESC** you cancel the current processing and return to the Selection menu.

Figure 4.2.1-7:  
CNC Turning,  
Schematic of the  
"Defining the clamping  
method" menu;  
workpart clamping with  
stepped jaws.

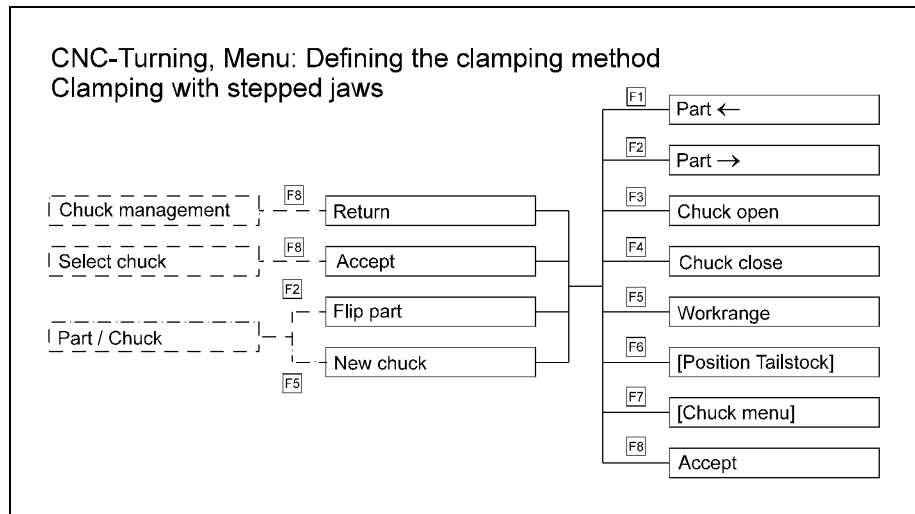
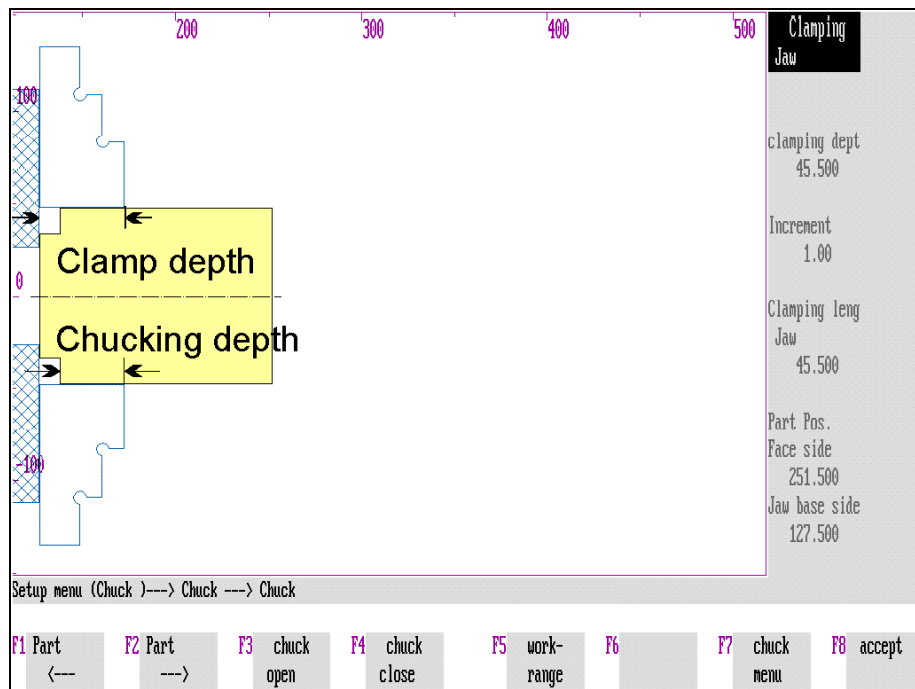


Figure 4.2.1-8:  
CNC Turning,  
defining the clamping  
method with a stepped  
jaw chuck.



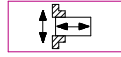
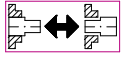


## Defining Clamping Method

### Clamping Types

The definition of the clamping method depends on the clamping type:

- Clamping with a jaw chuck or
- Clamping between centers.



The "Defining the clamping method" function is **offered automatically** after the Clamping Fixture Manager or after the blank definition/selection of the clamping type.

This function is also loaded if you have selected "**Reclamp**" with **F2** or "**New clamping**" with **F5** proceeding from the "Workpart/Clamping Fixture" menu.

## Clamping with Jaw Chuck

When using a jaw chuck, you need to position the workpart and to close/open the stepped jaws; for this the following functions are available:

**F1**

or



**Workpart -->/Workpart <--**: Shift the workpart to the desired clamping position with **F1** or **F2** or the cursor keys **←** or **→**.

**F2**

or



The information messages "**Clamping depth**" and "**Chucking length**" indicate how the workpart is clamped (cf. Figure); "**Workpart Pos.**" shows the workpart position in relation to the machine zeropoint; "**Increment**" shows how many units the clamping fixture has been shifted.

**F3**

or



**Clamping fixture open/close**: To close or open the clamping fixture,

**F4**

or



press the function keys **F3** or **F4** or the cursor keys **↑** or **↓**. If the clamping fixture is closed, this is indicated by a corresponding message.

**F5**

**Work space**: If you want to display the entire work space, press **F5**. You can also return to the standard display with **F5**.

**F6**

**[Tailstock position]**: If both the tailstock and the jaw clamping can be set, switch to the "Set/Drop Tailstock" menu (cf. below) with **F6**.

**F7**

or



**Clamping Fixture menu**: To replace clamping elements or to change the entire type of the workpart clamping, return with **F7** or **ESC** to the selection of the clamping fixture.

**F8**

or



After defining the clamping method, confirm this with **F8** or **↵**.

### Note

To position the workpart more precisely, you can additionally change the "increment". The unit indicating how much the workpart has been shifted after pressing once either of the keys **F1** or **F2** is called "increment" (please refer to Section 4.5 below for setting the increment).



After having confirmed the clamping method, the starting menu "Workpart and Clamping Fixture Definition" is automatically loaded. **F8** returns you from there to the main menu of the Setup Mode.

Figure 4.2.1-9:  
CNC Turning,  
Schematic of the  
"Defining the clamping  
type" menu for  
workpart clamping  
between centers.

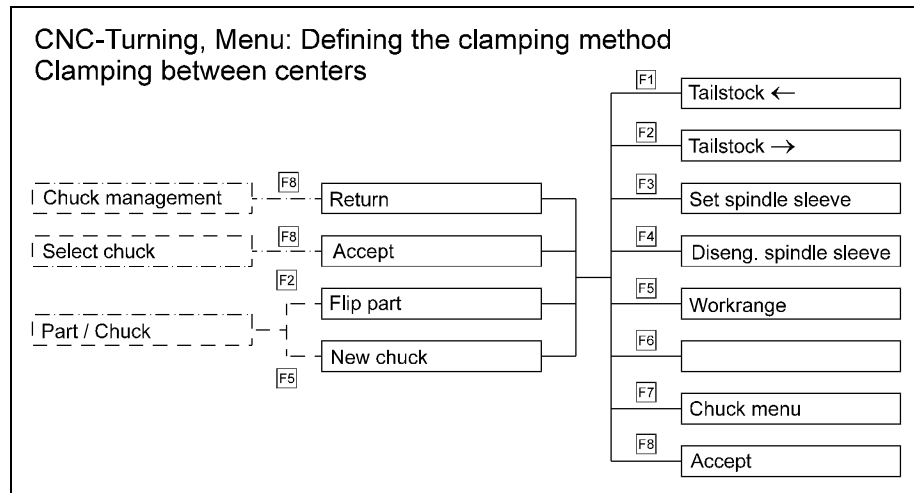


Figure 4.2.1-10:  
CNC Turning,  
defining the clamping  
method for clamping  
between centers.

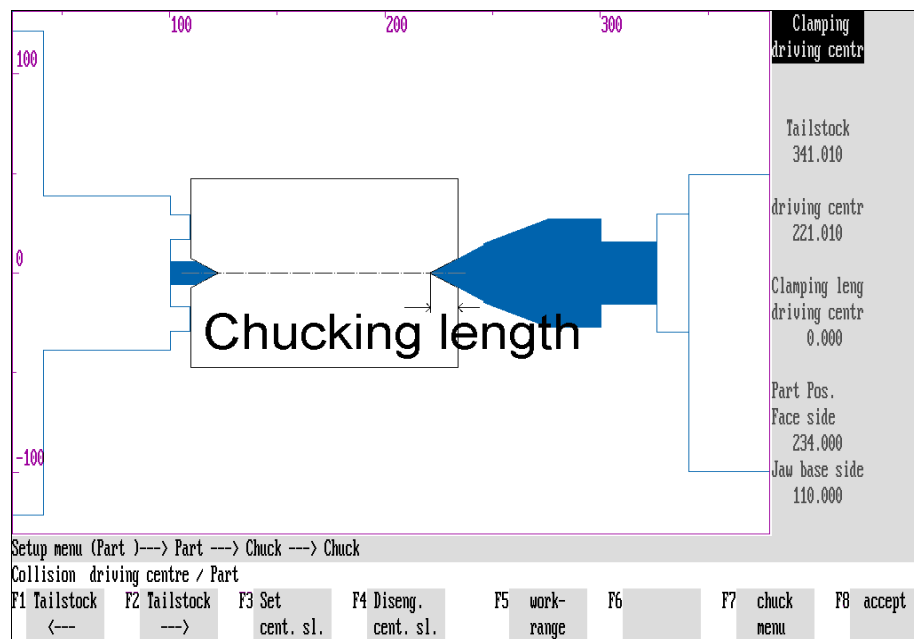
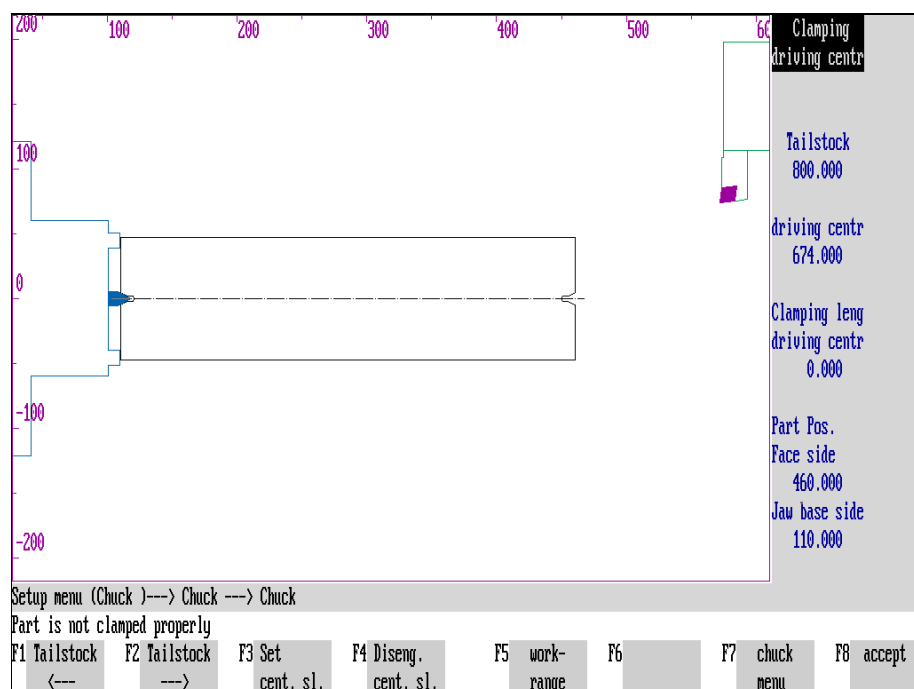


Figure 4.2.1-11:  
CNC Turning,  
workpart and clamping  
fixture definition;  
If the workpart cannot  
be clamped with the  
selected clamping  
devices, the error  
message "**Workpart  
cannot be clamped**" is  
displayed and the  
program switches  
automatically back to  
selection of the  
clamping method. In  
this example the  
selected face driver is  
too large. In such a  
case, change either  
the blank dimensions,  
type of clamping or  
clamping fixture.



## Positioning of Tailstock and Sleeve

When clamping between centers, the face driver is inserted automatically and you set the tailstock and sleeve manually.



This menu is available if either the "With tailstock" option or clamping between centers (possible for centered or machined parts) has been selected at the clamping type selection.

### Note

In the machine configuration you enter whether you want to simulate a lathe with tailstock and define the geometrical dimensions of tailstock and sleeve as well as the increment for positioning the tailstock.

**Two operational steps** are necessary to set or drop the tailstock and the sleeve:

**F1**

or



**Tailstock -->/Tailstock <--** : At first the tailstock is moved with the function keys **F1** and **F2** or the cursor keys **←** and **→** close enough to the workpart sleeve to be

**F2**

or



set. At the same time the "shift" of the tailstock is displayed on the screen.

The information messages "**Tailstock**" and "**Sleeve tip**" indicate the respective positions with regard to the workpart zeropoint (cf. Figure).

**F3**

or



**Set sleeve/open sleeve**: The sleeve is then set to clamp the workpart with **F3**. The sleeve is opened again with **F4**.

If the workpart is clamped, this is indicated with a corresponding message.

The information message "**Sleeve tip chucking length**" indicates the chucking length (cf. Figure).

**F5**

**Work space**: If you want to display the entire work space, press **F5**. You can also switch back to the standard display with **F5**.

**F7**

or



**Clamping Fixture menu**: Cancel processing with **F7** or **ESC** and return to the selection of the clamping fixture.

**F8**

or



After clamping the workpart, confirm this with **F8** or **↓**.

The "Workpart and Clamping Fixture Definition" menu is then automatically loaded. **F8** returns you from there to the main menu of the Setup Mode.



Before you set the tailstock, be sure to have the tool positioned far enough from the rotation axis and to have the spindle switched off.

Figure 4.2.1-12:  
CNC Turning,  
Schematic of the  
"Workpart and  
Clamping Fixture  
Definition" menu.

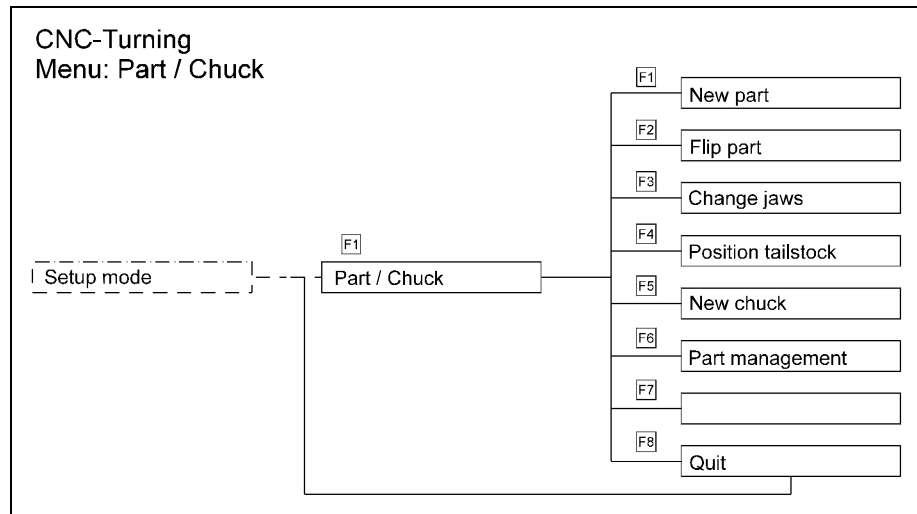
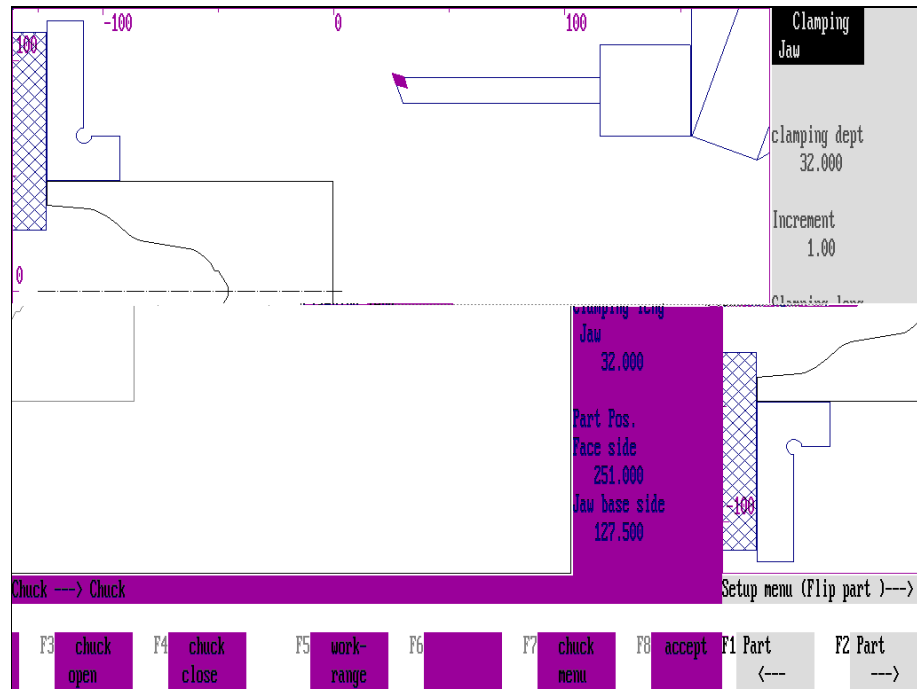


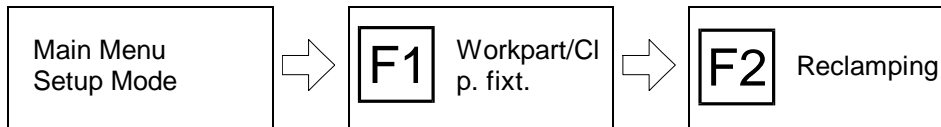
Figure 4.2.1-13:  
CNC Turning,  
reclamped workpart.



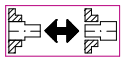
## Reclamping a Workpart

Workparts can be reclamped for further machining on both sides as many times as required:

### Selection:



### Procedure



After calling up the "Workpart and Clamping Fixture Definition" menu from the Setup Mode, you can reclamp the currently clamped workpart with **F2**.

The workpart is displayed on the screen rotated 180°.

The **menu for defining the clamping method** is then loaded automatically (cf. above).

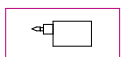
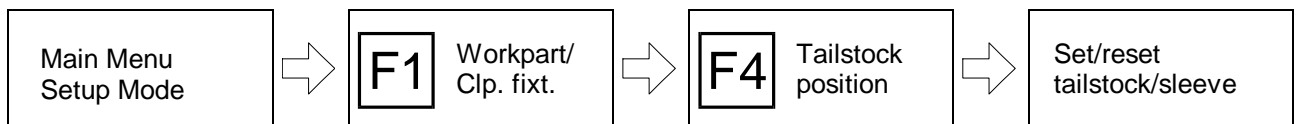
## Tailstock

A guide through the tailstock tip is used not only to clamp between centers but can also be selected to clamp with the stepped jaw chuck, e.g. to machine shafts.



**This clamping method can be defined only if the function "With tailstock" has been activated in the "Clamping Type Selection" menu.**

### Selection:



To use or drop the tailstock, call up the function "Tailstock position" with **F4** in the "Workpart/Clamping Fixture Definition" menu.

The further handling steps are identical with those of defining the clamping between centers (cf. above) and are therefore not repeated here. To conclude this subject item, some general notes on handling the tailstock menu for your reference:

- To clamp with tailstock, the tailstock needs first to be moved next to the workpart to be able to set the sleeve.
- To cancel the "clamping with tailstock" function, the sleeve needs first to be opened and after that the tailstock to be withdrawn.
- As soon as the tailstock is back in its starting position, the starting menu "Workpart and Clamping Fixture Definition" is loaded automatically.
- The geometrical and technical data of the tailstock and of the sleeve are defined in the configuration.

Figure 4.2.2-1:  
CNC Turning,  
Schematic of the  
menu sequence for the  
Clamping Fixture  
Manager, simplified  
presentation.

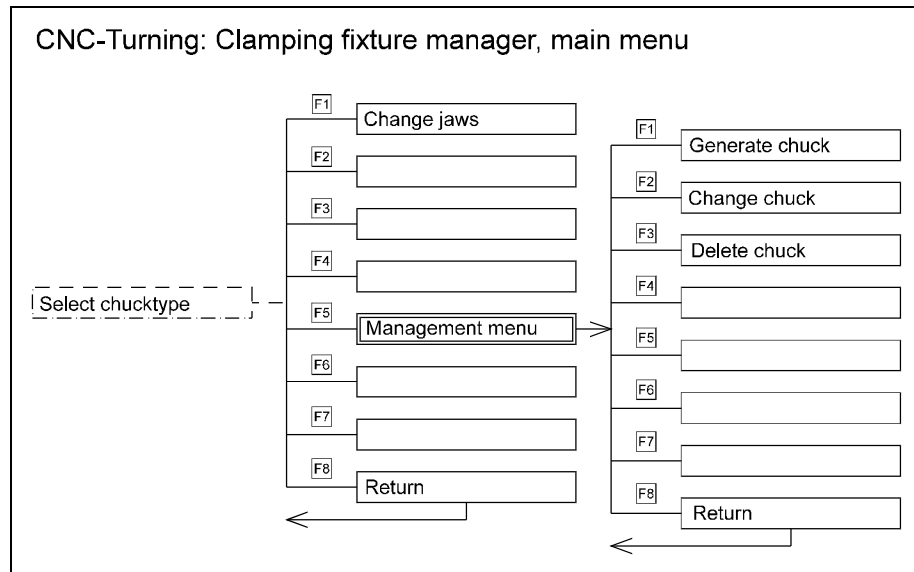
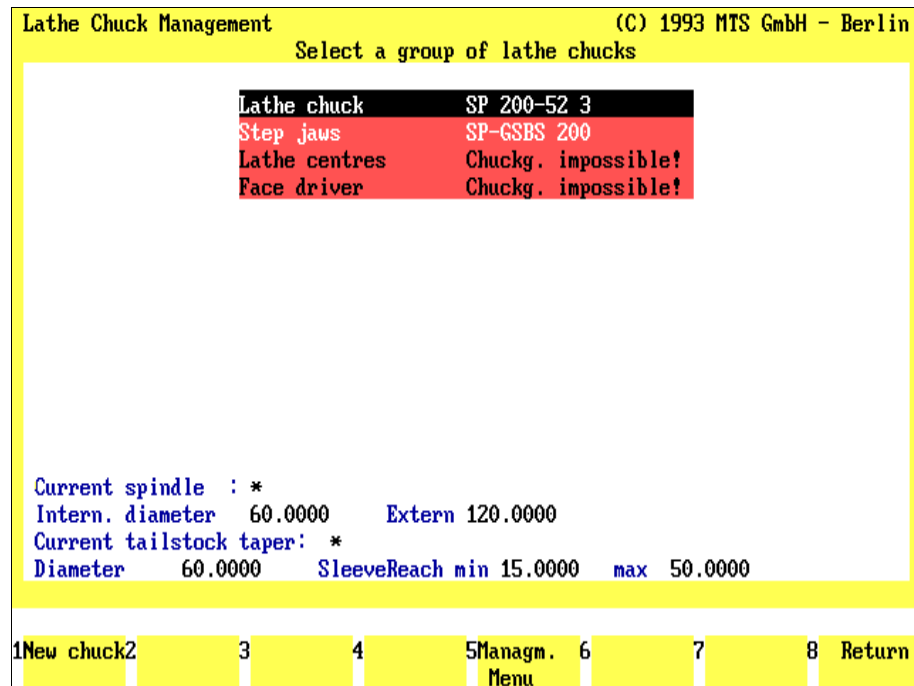
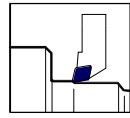


Figure 4.2.2-2:  
CNC Turning,  
Clamping Fixture  
Manager,  
main menu.



## 4.2.2 CNC Turning: Clamping Fixture Manager



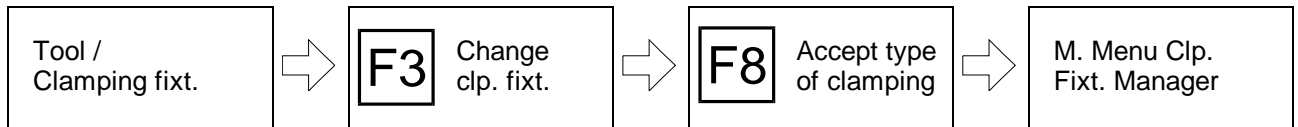
Apart from the individual assembly of the clamping elements to form a clamping system (see above), all elements: lathe chuck, stepped jaws, face driver and lathe centers can be redefined, changed and deleted in the Clamping Fixture Manager.

The Clamping Fixture Manager is loaded automatically after the clamping type has been selected proceeding from the "Change Clamping Fixture" function.



**If the blank has already been defined, the Clamping Fixture Management menu is skipped.**

### Selection:



## Clamping Fixture Manager: Main Menu

### Screen Display

The layout of the screen display was described in [Section 4.2.1](#) (Setup Mode). In general, the upper screen area is reserved for graphics, information or input fields, whereas the function key fields are displayed at the lower end of the screen, providing the following operation alternatives:

**F1**

**Change clamping fixture:** You can select and insert a new clamping element out of the marked clamping fixture group with **F1**.

**F5**

**Management menu:** To create a new clamping element or to change or delete an existing one, press **F5**. To avoid unauthorized interventions in the clamping fixture manager, this function can be protected with a "password" (cf. Configuration Instructions).

**F8**

**Return:** With **F8** you confirm the listed combination of the clamping elements and return to the menu for defining the clamping method.

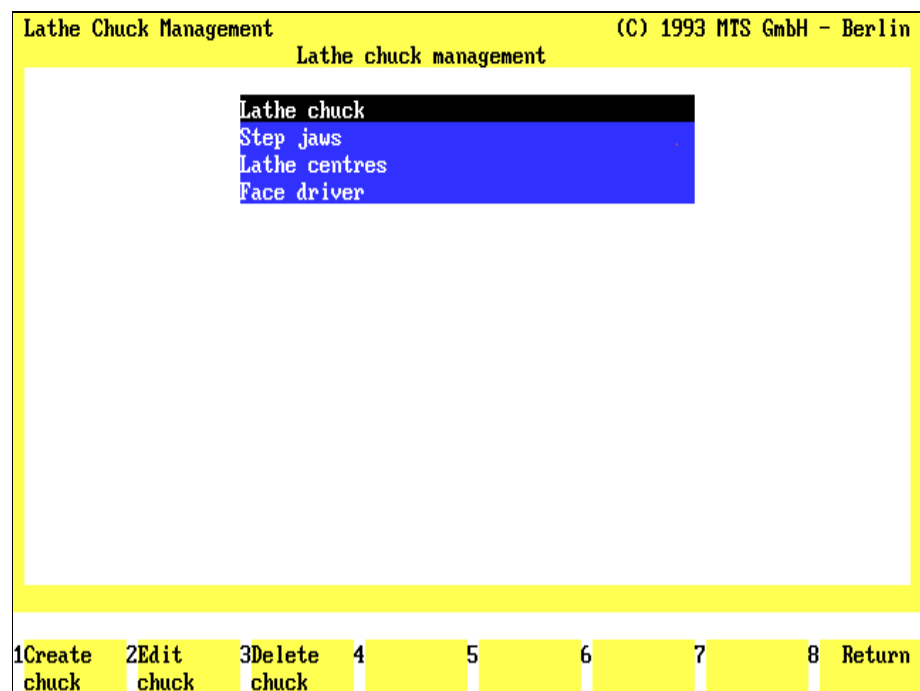
**ESC**

With **ESC** you cancel processing and return to the starting situation.

### Note

The clamping fixture manager is structured according to uniform criteria: the same menu structure and handling is applied for all groups of clamping fixtures. For the sake of clarity, the following systematic description is therefore structured on the basis of practical handling features. In case of analogous processing steps only the nonanalogous steps are described, to avoid repetition.

Figure 4.2.2-3:  
CNC Turning,  
Clamping Fixture  
Manager;  
Management menu.





## Clamping Fixture Manager: Management Menu

The management of the clamping elements is called up with **F5** from the main menu of the Clamping Fixture Manager.

### Password

To avoid unauthorized manipulation of the Clamping Fixture Manager, this function can be protected with a "password". If you have defined a password in the configuration, enter it after selecting the Clamping Fixture Manager and confirm your entry with **↵**.

The management menu is then loaded:

The clamping fixture groups are displayed in the upper screen area, subdivided into the elements of:

- Lathe chuck
- Stepped jaws
- Face driver
- Lathe centers

Irrespective of the current clamping type selection, you have the following processing options for the clamping fixture file:

**F1**

**Create clamping fixture:** Press **F1** to define a new clamping element.

**F2**

**Change clamping fixture:** Press **F2** to change an existing clamping element.

**F3**

**Delete clamping fixture:** Press **F3** to delete an existing clamping element.

**F8**

or

**ESC**

**Return:** **F8** or **ESC** returns you to the main menu.

### Procedure

To edit the clamping fixture file, first mark the corresponding clamping fixture group and then select the desired processing function:

**↑**

or

**↓**

Mark the desired clamping fixture group with the cursor keys **↑** or **↓**.

**F1**

or

**F2**

After you have marked a group, switch on to the desired processing.

**F3**

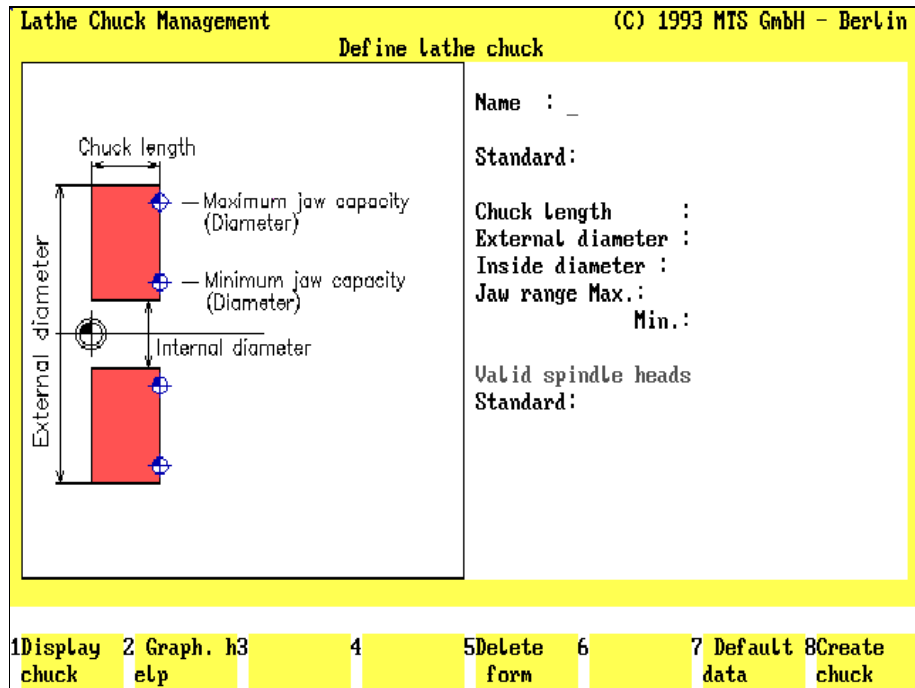
### Note

**Please note that after changes of the clamping fixture file, also certain entries in existing setup sheets (e.g. data on clamping depth etc.) or existing NC programs may have to be changed as well.**

**Therefore, change the clamping fixture file only as far as necessary and check subsequently the setup sheets and NC programs containing data on these clamping elements.**

**When in doubt, generate a new clamping element!**

Figure 4.2.2-4:  
CNC Turning,  
Clamping Fixture  
Manager;  
Create clamping  
element,  
example of lathe  
chuck.



## Clamping Fixture Manager: Define Clamping Element

### Screen Display

Define a new clamping element by calling up the corresponding menu with **F1** from the main menu of the Clamping Fixture Manager.

The input fields for entering the name, dimensions etc. of the clamping element are in the upper screen area. On the left, next to this there is a window for displaying the graphic presentation or an auxiliary graphic of the defined clamping element.

The menu functions in the lower part of the screen indicate the available further operation steps:

**F1**

**Clamping fixture graphic display:** As soon as you have completed the definition of a clamping element, you can have this displayed with **F1**. In case of incomplete data records, the corresponding error message is displayed.

**F2**

**Auxiliary graphic:** An auxiliary graphic clarifying the entry parameters of the elements exists for each clamping fixture group. **F2** displays the auxiliary graphic.

**F5**

**Delete form:** All entries in the input fields are deleted with **F5**.

**F7**

**Copy data:** If you want to take up the geometrical data of an already existing clamping element, press **F7**.

**F8**

**Create clamping fixture:** With **F8** you confirm your entered data to create a new clamping element.

### Procedure

When entering the parameters, etc., please note the following operating steps:

**TAB**

You move from input field to input field with the **TAB** key (or **△ shift** and **TAB**).

[Value Statement or Name]

The input is entered as numbers, a decimal point with **<,>** or **<.>**.

**ESC**

With **ESC** you cancel current processing and return to the management menu.

**F8**

Confirm your entries with **F8** only after all parameters have been entered.



After the new clamping element has been saved, the input menu is loaded again with empty input fields for the definition of a further element.

**ESC** returns you to the management menu.

Figure 4.2.2-5:  
CNC Turning,  
Parameters for  
determining the lathe  
chuck jaws

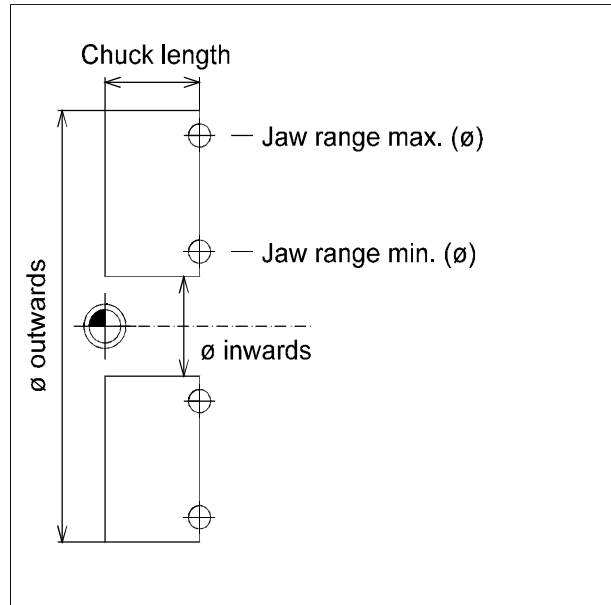


Figure 4.2.2-6:  
CNC Turning,  
Parameters for  
determining stepped  
jaws

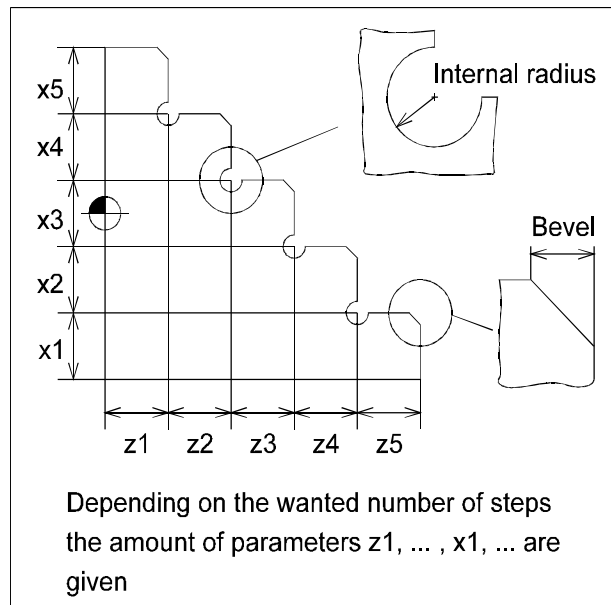
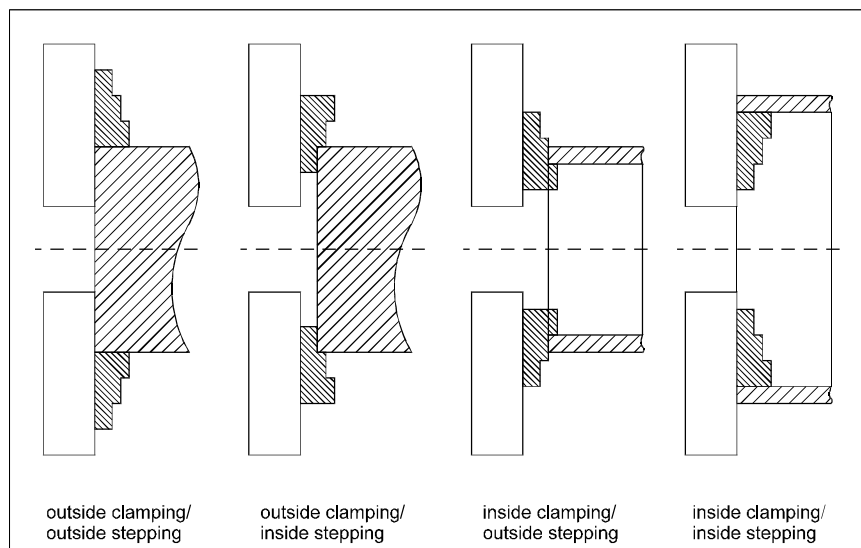


Figure 4.2.2-6a:  
CNC-Turning,  
application options for  
stepped jaws



The following parameters are entered to determine a clamping element:

### Lathe Chuck Input Parameters

<b>General Entries</b>	Name:	arbitrary entry
	Standard designation:	arbitrary entry (Standard: DIN 4612)
<b>Dimensions</b>	Chuck length:	in Z ( $Z > 0$ )
	External diameter:	in $X_A$ ( $X_A > 0$ )
	Internal diameter:	in $X_I$ ( $X_I < X_A$ )
	Jaw range (Max.):	in $X_{max.}$ ( $X_A > X_{max.} > X_{min.}$ )
	Jaw range (Min.):	in $X_{min.}$ ( $X_I < X_{min.} < X_{max.}$ )
	<b>Permissible Spindle Heads:</b>	Standard designation:

### Stepped Jaws Input Parameters

General Entries:	Name:	arbitrary entry
<b>Dimensions:</b>	Number of steps:	numerical value: 1 - 5
	$X_1$ and $Z_1$ :	X and Z values of the first step ( $> 0$ )
	$X_2$ and $Z_2$ etc.:	X and Z values of the 2nd step ( $> 0$ ) etc.
	Bevel:	Bevel length of step outer edge ( $\cdot 0$ )
	Radiusing:	Radiusing of the inner edges ( $\cdot 0$ )
	Mounting:	Jaw mounting onto chuck in X ( $> 0$ )
	<b>Use:</b>	Stepping outwards <sup>2</sup> :
Stepping inwards <sup>2</sup> :		1 = Yes / 0 = No
<b>Permissible Chuck</b>	Standard designation:	arbitrary entry (e.g. DIN 4612) <sup>1</sup>

<sup>1</sup> The statement "\*" replaces an arbitrary character string.

<sup>2</sup> The jaws can be used with outside and/or inside stepping for outside and inside clamping. The clamping type is defined in the "Clamping Fixture Selection" menu.

Figure 4.2.2-7:  
CNC Turning;  
Parameters for  
determining the lathe  
centers

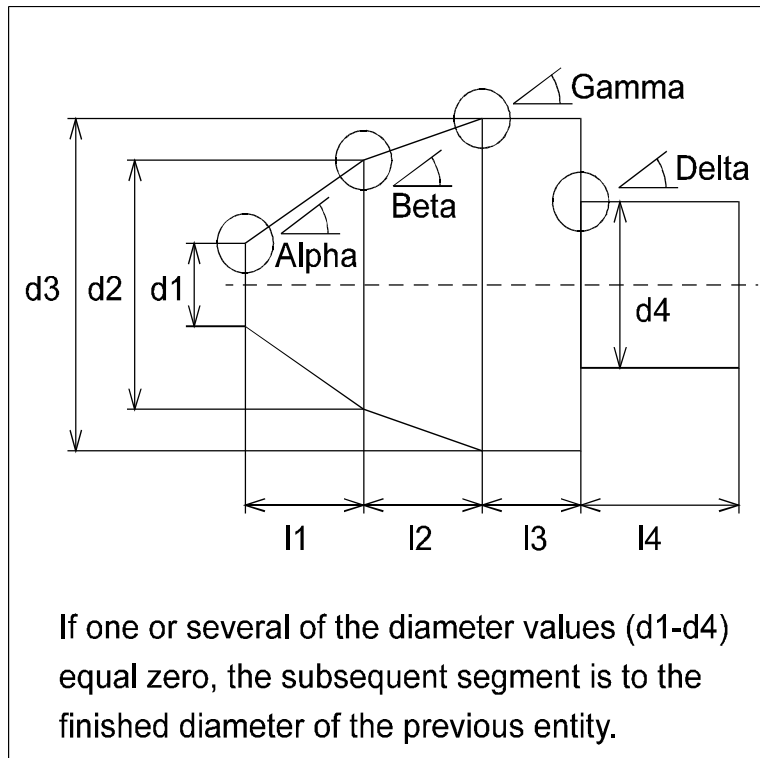
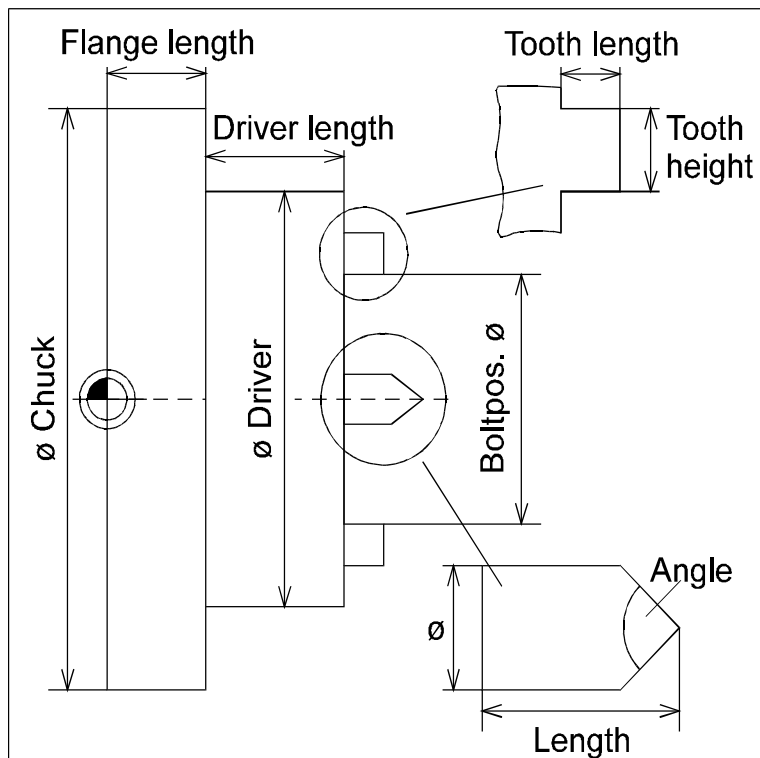


Figure 4.2.2-8:  
CNC Turning;  
Parameters for  
determining the face  
drivers



## Lathe Center Input Parameters

<b>General Entries</b>	Name:	arbitrary entry
<b>Dimensions</b>	The centers can have a maximum of four gradations:	
	Length:	$l_1$ to $l_4$ (in Z · 0)
	Diameter:	$d_1$ to $d_4$ (in X; at $d_i = 0$ the diameter of the previous segment is taken)
	Angle:	In each case total angle of a segment part (at angle size = 0, the parallel line to the Z axis is taken over)
<b>Assignment to a Sleeve</b>	Standard designation:	arbitrary entry (e.g. DIN 4712) <sup>1</sup>



**Length, diameter and angle depend upon one another!**

## Face Driver Input Parameters

<b>General Entries</b>	Name	arbitrary entry
	Rotation direction to the right	1 = Yes / 0 = No
	Rotation direction to the left	1 = Yes / 0 = No
<b>Dimensions:</b>	Flange length:	in Z > 0
<b>Flange:</b>	Chuck diameter	in X > 0
<b>Driver Body</b>	Driver length	in Z ≥ 0
	Driver diameter	in Z > 0
<b>Driving Bolts</b>	Tooth position	in X > 0
	Tooth length	in Z > 0
	Tooth height	in X > 0
<b>Centering Tip</b>	Center length	in Z > 0
	Center diameter	in X > 0
	Center angle	Total angle
<b>Permissible Spindle Heads</b>	Permi. spindle heads	arbitrary entry (standard DIN 4812) <sup>1</sup>

<sup>1</sup> The statement "\*" replaces an arbitrary character string.

<sup>2</sup> The statement 0/0 is inadmissible.

Figure 4.2.2-9:  
CNC Turning,  
Clamping Fixture  
Manager;  
Definition of a new  
clamping element and  
reading in specific  
existing data records.

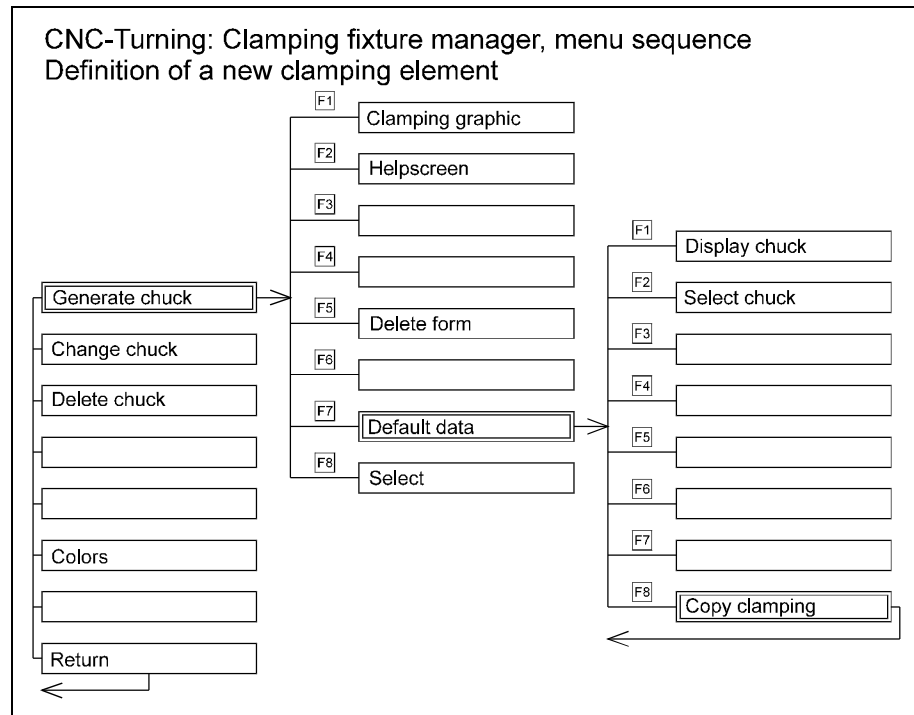
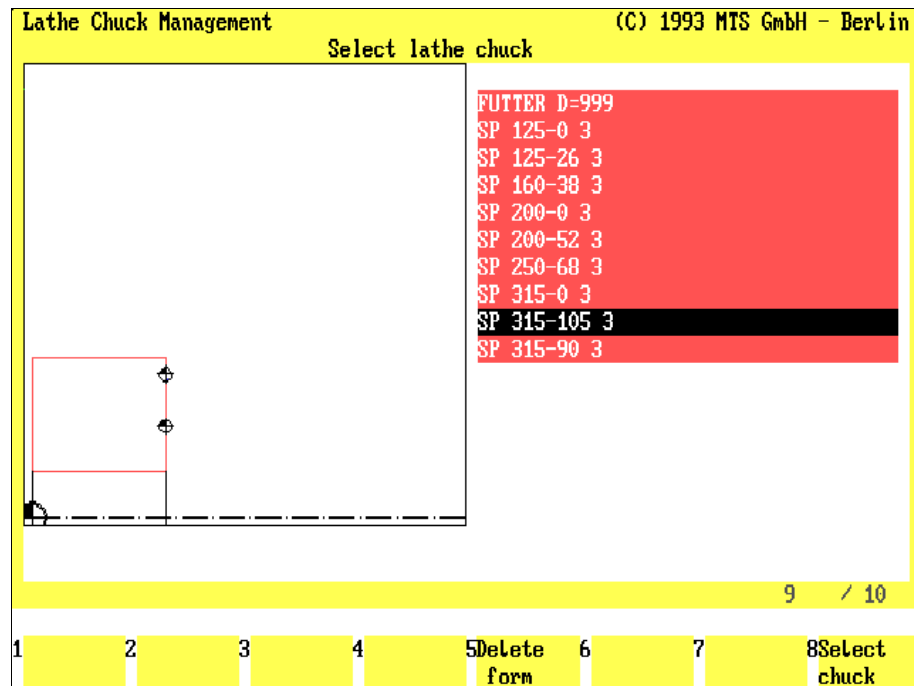


Figure 4.2.2-10:  
CNC Turning,  
Clamping Fixture  
Manager;  
Menu: Reading in  
specified data records.  
Example: Lathe Chuck





## Copy Data Record

If you want to use already existing data records for the definition of a new clamping element, you can read in the data and then change the parameters as desired:

The "Copy Data Record" function is called in the input menu with **F7**:

The existing clamping elements of the currently selected group are displayed in the upper field of the screen (cf. above, Change Clamping Fixture).

**F1**

**Show clamping fixture:** With **F1** you call up the detailed information for the currently marked chuck/clamping element.

**F2**

**Select clamping fixture:** To select out of the existing clamping elements those meeting certain geometrical criteria, you can have these searched and displayed as a new list.

**F8**

**Copy clamping element:** With **F8** you confirm and read-in the selected clamping element.

### Procedure

Please note the following steps for selecting existing data records:

**↑**

or

**↓**

Mark the desired clamping element in the selection window with the cursor keys **↑** or **↓** or **PageUp** or **PageDown**.

**F8**

**Copy clamping element:** **F8** returns you to the input menu and the specified data are copied to the input fields. You can then enter your data as described above to define a new clamping element.

**ESC**

With **ESC** you cancel processing and return to the starting situation.

Figure 4.2.2-11:  
CNC Turning,  
Clamping Fixture  
Manager;  
Schematic of the  
menu sequence:  
Modifying a clamping  
element.

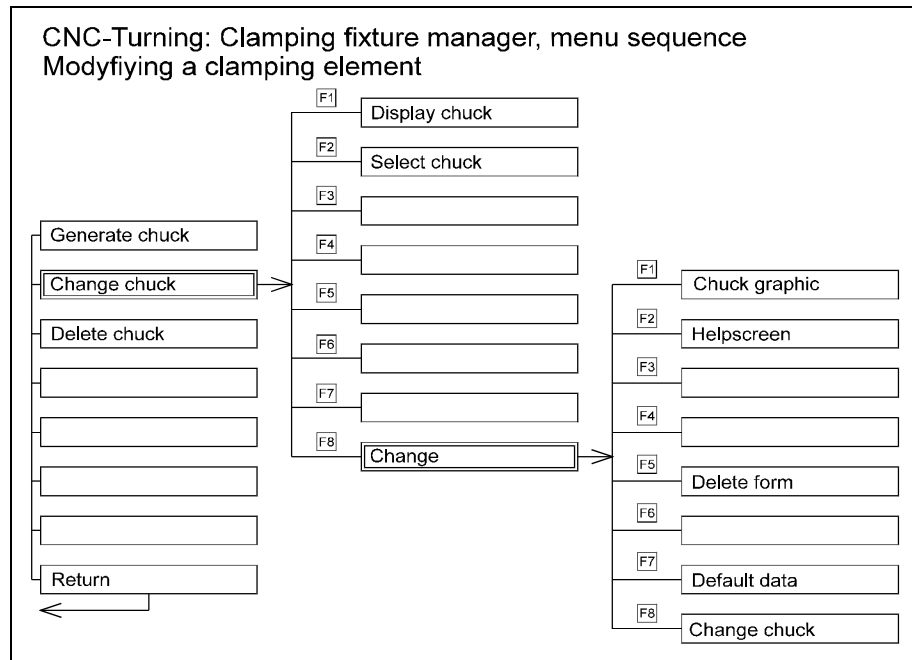


Figure 4.2.2-12:  
CNC Turning,  
Clamping Fixture  
Manager;  
Menu: Selection of a  
clamping element for  
modification.  
Example: Face driver

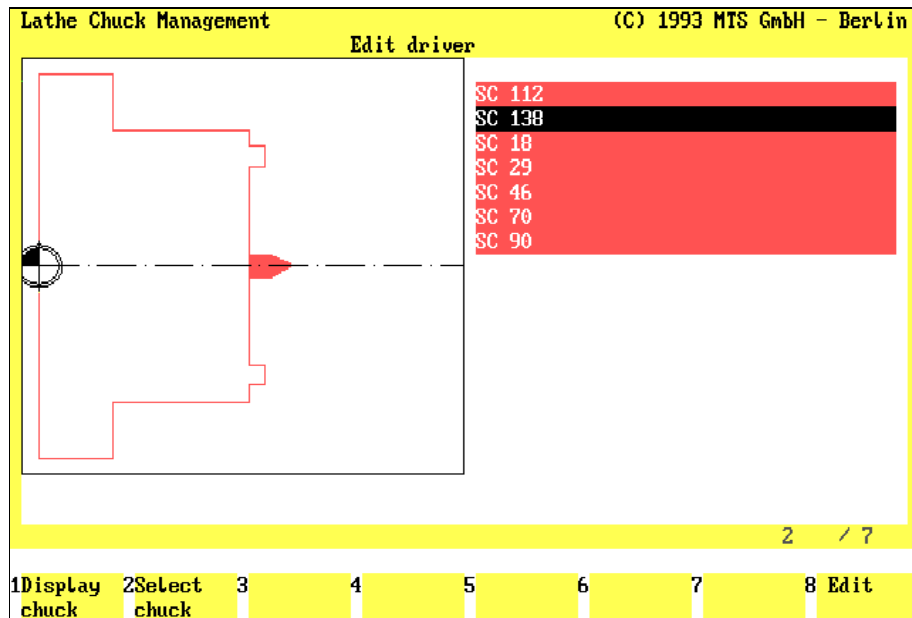
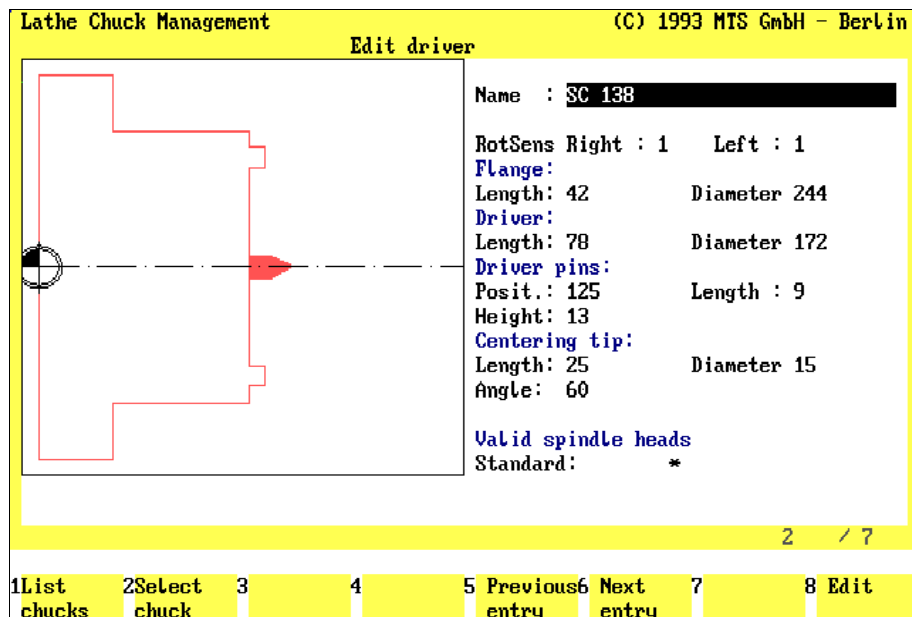


Figure 4.2.2-13  
CNC Turning,  
Clamping Fixture  
Manager;  
Modify clamping  
element,  
input menu for  
modification function.  
Example: Face driver



## Clamping Fixture Manager: Modify Clamping Element

The menu for modifying an existing clamping element is called with **F2** from the management menu of the Clamping Fixture Manager.

Processing takes place in two work steps:

1. First select the clamping element you want to modify and
2. then change the parameters as you wish:

### Select Element

The selection menu of the modification function is laid out analogously to the selection menu for copying data records (cf. preceding section):



or



Mark the desired clamping element in the selection window with the cursor keys **↑** or **↓** or **Page▲** or **Page▼**.



With **ESC** you cancel processing and return to the starting situation.



**Modify:** With **F8** you confirm the selected clamping element and switch on to the input menu to edit individual data.

### Edit Data

The input menu for data editing is laid out analogous to the input menu of a new clamping element (cf. above): to change input values, first activate the corresponding input field and then overwrite the existing values:



You move from input field to input field with the **TAB** key or **△** and **TAB**.



or



Move the cursor to a certain input position with the cursor keys **→** or **←**.

### [Value Statement or Name]

The input values are entered as numbers, a decimal point with **<, >** or **<.>** (the individual parameters are explained in the preceding section).



With **ESC** you cancel processing and return to the starting situation.



**Modify clamping fixture:** With **F8** you confirm your modifications.

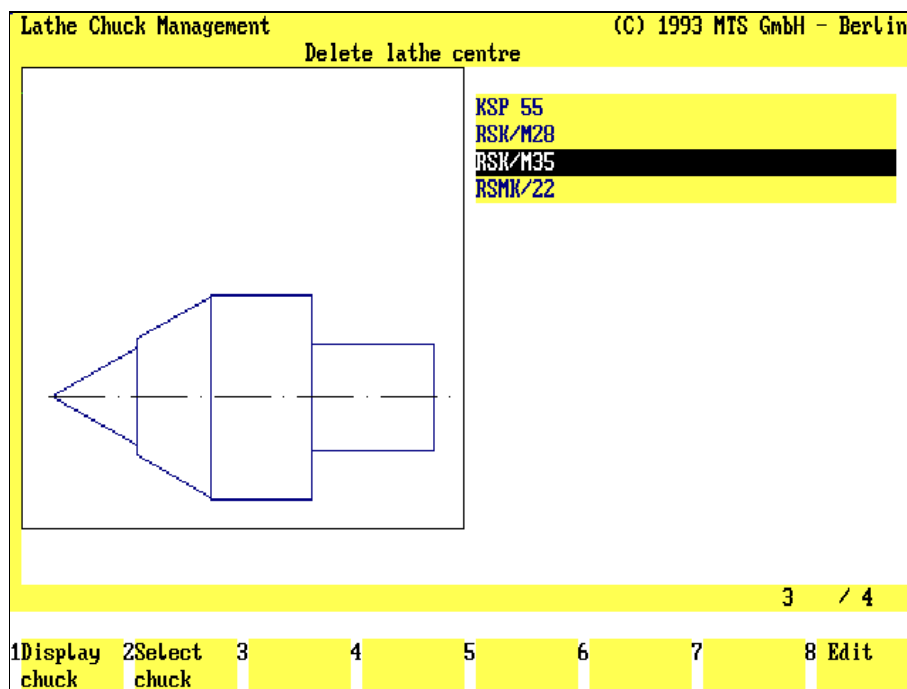
### Note

**Please note that after changes of the clamping fixture file, certain entries in existing setup sheets (e.g. data on clamping depth etc.) or existing NC programs may also have to be edited.**

**Therefore, you should edit the clamping fixture file only if necessary and then check the setup sheets and NC programs the clamping elements are used in.**

**If in doubt, generate a new clamping element!**

Figure 4.2.2-14:  
CNC Turning,  
Clamping Fixture  
Manager;  
Delete clamping  
element;  
Example: Lathe  
center.



## Clamping Fixture Manager: Delete Clamping Element

To delete an existing clamping element, a menu is called from the Management menu of the Clamping Fixture Manager with **F3**.

Processing takes place in two work steps:

- 1.) First select the clamping element to be deleted, and
- 2.) then confirm the control enquiry to delete this element:

### Select Element

The selection menu is analogously to the selection menu for copying element data (cf. last section but one):



or



Mark the desired clamping element in the selection window with the cursor keys **↑** or **↓** or **PageUp** or **PageDown**.



With **ESC** you cancel processing and return to the starting situation.



**Delete:** Confirm your selection with **F8**.

### Security Enquiry

Before the clamping element is deleted from the clamping fixture file, there is one further dialogue enquiry for control: **Do you want to delete this clamping element?**



**No:** **F7** returns you to the selection menu.



**Yes:** **F8** deletes the clamping element.

### Note

**Please note, that after a clamping element has been deleted, existing setup sheets may need to be changed in respect of the deleted element as well . Therefore edit the clamping fixture file only as far as necessary and then check the setup sheets and NC programs as to the clamping elements concerned.**

Figure 4.2.2-16:  
CNC Turning,  
Clamping Fixture  
Manager;  
Select clamping  
fixture;  
Example: Lathe  
center.

**Lathe Chuck Management** (C) 1993 MTS GmbH - Berlin

**Select lathe centre**

The diagram shows a lathe chuck profile with four segments. The diameters are labeled d1, d2, d3, and d4. The lengths of the segments are labeled l1, l2, l3, and l4. The angles are labeled Alpha, Beta, Gamma, and Delta. The segments are numbered 1, 2, 3, and 4 from left to right.

Name : \_

**Lengths:**

l1:                      l2:

l3:                      l4:

**Diameter**

d1:                      d2:

d3:                      d4:

**Angle:**

alpha:                      beta:

gamma:                      delta:

**Valid centre sleeve:**

Standard:

If one or several of the diameter values (d1-d4) equal zero, the subsequent segment is to the finished diameter of the previous entity

1
2
3
4
5Delete form
6
7
8Select chuck

## Clamping Fixture Manager: Select Clamping Fixture

From the extensive list of existing clamping fixtures this menu selects the elements meeting certain geometrical criteria and displays them as a new list for a better overview.

**F2**

**Select clamping fixture:** This function is called with **F2** in the selection menus and the display menu.

The selection menu is analogous to the input menus (cf. the preceding sections):

**F5**

**Delete form:** All current entries in the input fields are deleted with **F5**.

**F8**

**Select clamping fixture:** With **F8** you confirm your entries for the selection of the clamping fixture from the list.

### Procedure

The loaded menu is first without entries, You then make the entries in the relevant input fields; **only the data** to serve as selection criteria is entered, e.g.:

- all lathe chucks with internal diameter of 38 mm => "Internal dia.: 38";
- all three-step clamping jaws => "Number of steps: 3" etc.

**TAB**

You move from input field to input field with the **TAB** key.

### [Value Statement or Name]

The values are entered as numbers, the decimal point with <, > or <.>. Further input options are listed in the following table.

Elements can also be selected according to relational dependencies:

	Entry	Selection
numerical	"X ... Y"	Those elements are selected whose numerical value in the corresponding input field is larger than or equal to X and smaller than or equal to Y
	"> X" or ">= X"	> X or · X (see above)
	"< X" or "<= X"	> X or μ X (see above)
alphanumeric	" * "	arbitrary character string
	" ? "	arbitrary character

If several selection criteria are entered, then only those elements meeting all criteria are selected.

**ESC**

With **ESC** you cancel the current processing and return to the starting situation.

**F8**

Confirm your entries with **F8** only after all desired selection criteria are entered. You then switch back automatically to the preceding processing situation, the displayed list of the clamping elements now contains only those elements meeting the selection criteria.



To have the original complete list of all existing clamping elements displayed again, enter **no** selection criterion in the input fields.

Figure 4.2.3-1:  
CNC Milling,  
Schematic of the menu  
sequence for selecting  
a clamping system  
after a blank change.

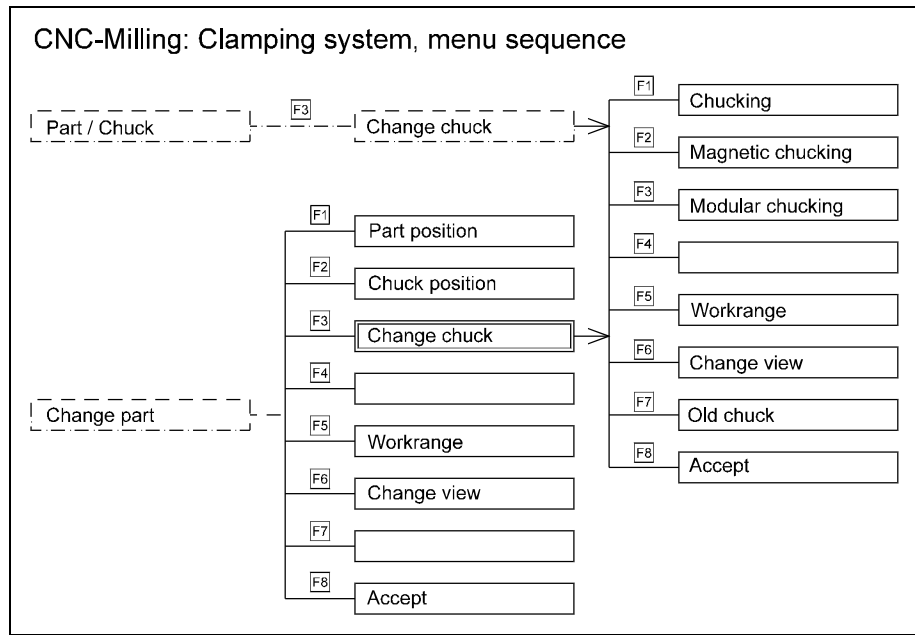
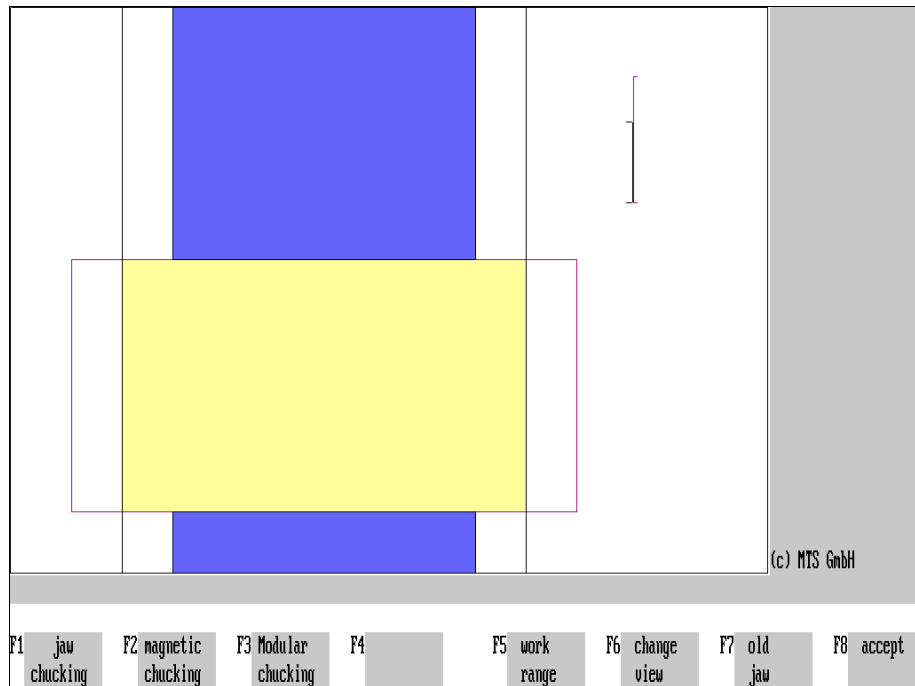


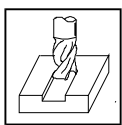
Figure 4.2.3-2:  
CNC Milling,  
Selection menu of the  
clamping systems.





## 4.2.3 CNC Milling: Clamping Fixtures

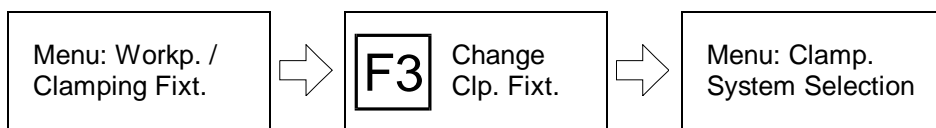
### Selection of the Clamping System



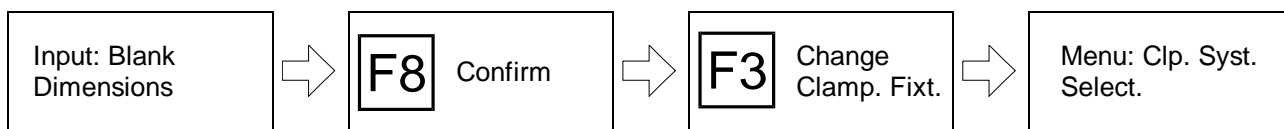
Milling machines, as a rule, have several options for clamping the workpart. For this reason the MTS Milling Simulator offers three different clamping systems: jaw clamping, modular clamping and magnetic plate clamping.

As in turning, the clamping elements can be defined in a separate Clamping Fixture Manager. If modular clamping is used, you can define simple clamping elements (clamps, bases, positioning and supporting elements) in a processing menu and you can then assemble these to form a module or a system fixture.

#### Selection:



#### Automatic Menu Guidance:



The menu for the clamping system selection is called with **F3** proceeding from the "Change Clamping Fixture" function:

**F1**

**Jaw clamping:** Select the vice manager with **F1**.

**F2**

**Magnetic plate clamping:** The workpart is magnetically clamped with **F2**.

**F3**

**Modular clamping:** Select processing of modular clamping with **F3**.

**F5**

**Work space:** To display the entire work space, press **F5**. You can also return to the standard display with **F5**.

**F6**

**Change view:** Similar to blank definition, you can change the view of the workpart when processing the clamping systems.


**F7**

**Previous clamping fixture:** If you have already selected a new clamping system, you can reverse this with **F7**.

**F8**

or



**Accept:** After the clamping system has been selected, press **F8** or  to move to the following menu "Defining the Clamping Method".

**ESC**

With **ESC** you cancel selection of the clamping system and return to the starting situation.

#### Note

The "Change View" function is described in [Section 4.1.2](#).



**To reduce the setup time, blanks or preproduced workparts can be saved and loaded together with the clamping system as a file. Alternatively, you can save a specific setup status as a status file or generate a setup sheet for this.**

Figure 4.2.3-3:  
CNC Milling,  
Clamping Fixture  
Manager;  
Schematic of the  
menu sequence for  
changing a vice.

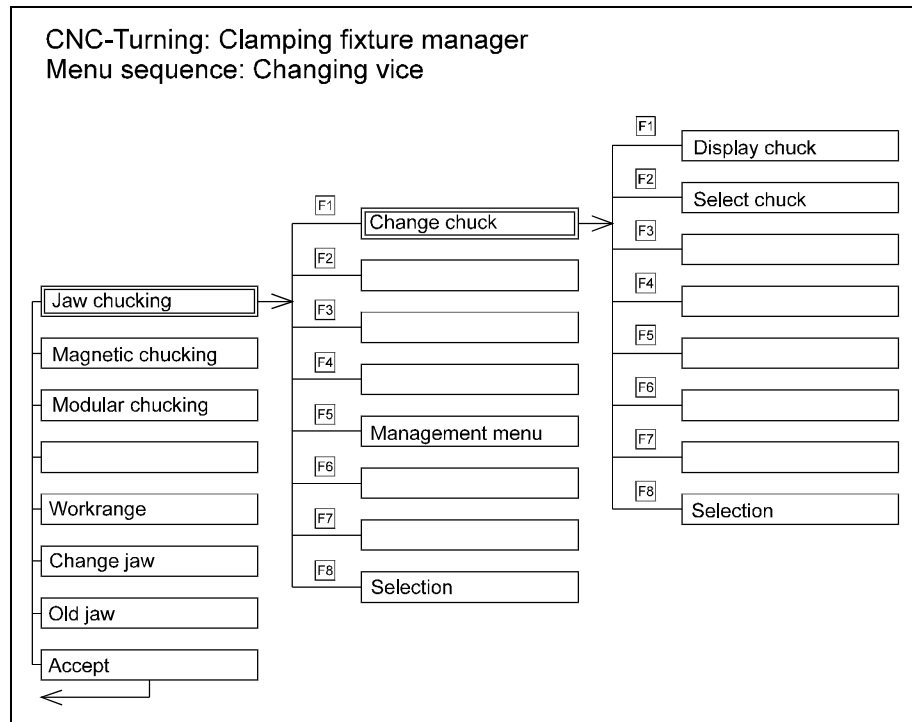
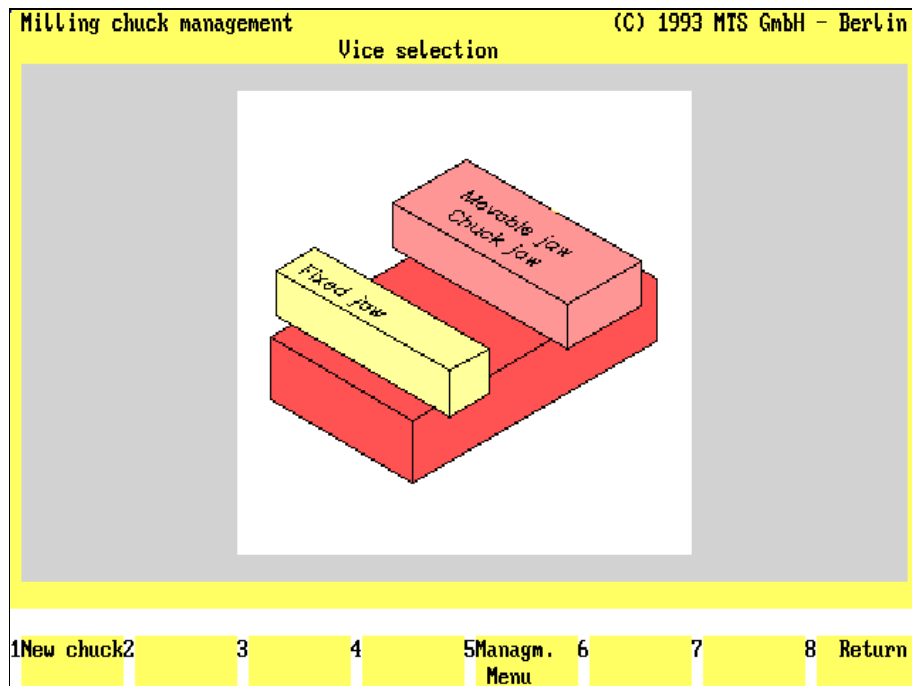


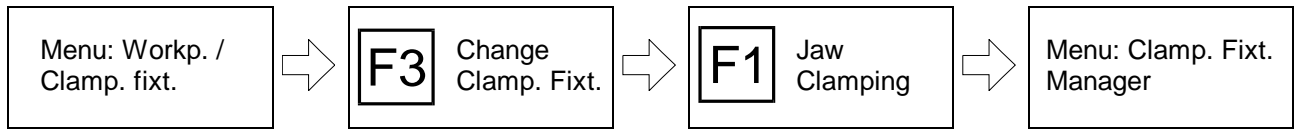
Figure 4.2.3-4:  
CNC Milling,  
Clamping fixture  
definition;  
Selection menu of the  
Clamping Fixture  
Manager.



## Changing the Vice

For jaw clamping, the MTS Simulator for Milling has a separate Clamping Fixture Manager for defining and replacing of individual vices.

### Selection:



After the menu item "Jaw Clamping" is selected with **F1**, the Vice Manager is automatically called.



To complete the general processing description of clamping fixtures before the detailed description of clamping fixtures, the Clamping Fixture Manager is discussed at this place only as far as necessary for changing the vice. The detailed description follows in [chapter 5](#).

### Clamping Fixture Manager

The Vice Manager menus are similar to those of CNC Turning Clamping Fixture Manager:

- 1.) The upper screen area is reserved for help graphics, information and input fields.
- 2.) The applicable function keys are displayed on the lower screen margin.

**F1**

**Change clamping fixture:** To insert a new vice, press **F1**.

**F5**

**Management menu:** To define or modify a vice, press **F5**. To avoid unauthorized manipulation of the Clamping Fixture Manager, this function can be protected with a "password" (cf. Configuration Instructions).

**F8**

**Return:** **F8** returns you to the menu for defining the clamping method.

**ESC**

**Cancel:** With **ESC** you cancel processing and return to the starting situation.

Figure 4.2.3-5:  
CNC Milling,  
Clamping fixture  
definition;  
Selection of a vice.

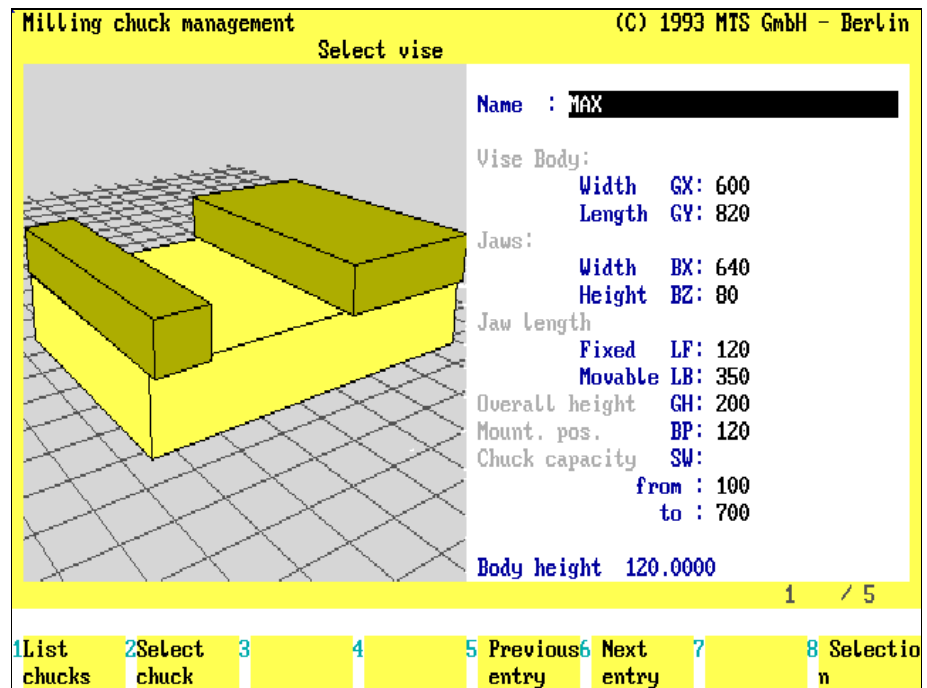
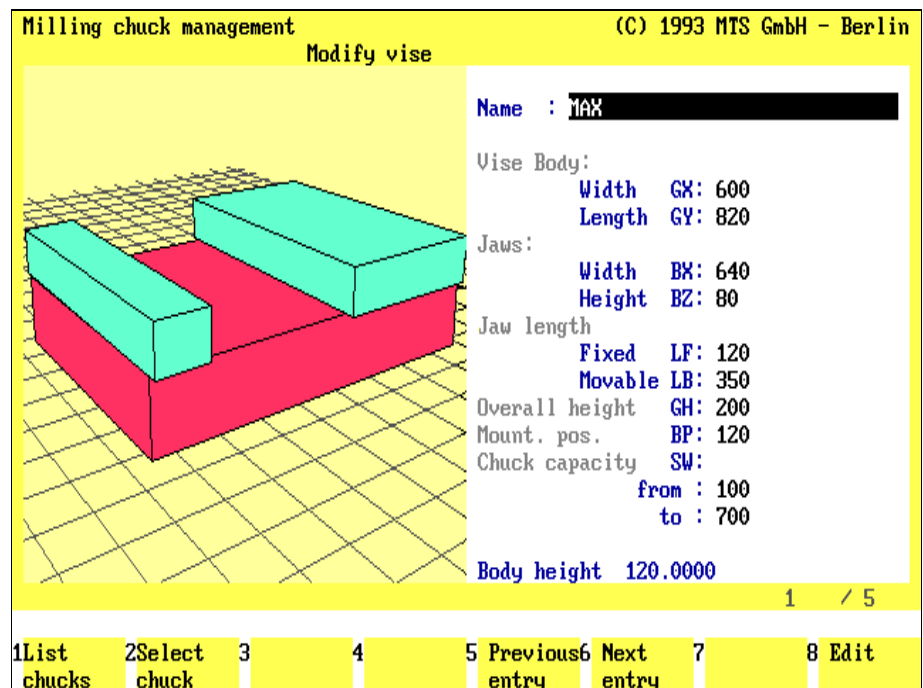


Figure 4.2.3-6:  
CNC Milling,  
Clamping fixture  
definition;  
Display of the data of  
an individual vice.



## Overview

In the selection menu of the Clamping Fixture Manager, a graphic representation of the currently selected clamping element is displayed for better understanding (cf. Figure). A list of the available vices is displayed next to it.

The following processing functions are now available:

**F1**

**Show clamping fixture:** With **F1** you call up the detailed information on the currently selected clamping element.

**F2**

**Select clamping fixture:** To select from the complete list of existing clamping elements those meeting certain geometrical criteria, you can have these searched and displayed as a new list. A detailed description of this function is given in the following chapter.

**F8**

**Selection:** With **F8** you confirm the selected clamping element.

**ESC**

With **ESC** you cancel processing and return to the starting situation.

## Procedure

To select the vice, first mark the corresponding element in the overview list and then confirm your selection:

**↑**

or

**↓**

Mark the desired vice for selection with the cursor keys **↑** or **↓** or **PageUp** or **PageDown**.

**F8**

**Selection:** Confirm your selection with **F8**. You then return to the main menu of the Clamping Fixture Manager.

## Detailed Information

As an alternative to selecting the vice in the general overview, this can also be done by displaying the detailed information. This display function is available in the overviews of the Clamping Fixture Manager to check the geometrical data of a clamping element:

**F1**

**Show clamping fixture:** Press **F1** to display the detailed data.

## Procedure:

Select the vice from the "Detail display", the same way described above:

**F1**

**List clamping fixtures:** Press **F1** to return to the overview of all vices.

**F2**

**Select clamping fixture:** To select from the complete list of existing clamping elements those meeting certain geometrical criteria, you can have these searched and displayed as a new list.

**F5**

**Previous entry :** **F5** or **Shift** and **↑** show the geometrical data of the clamping element preceding the currently specified element in the list.

**F6**

**Next entry :** **F6** or **Shift** and **↓** show the geometrical data of the clamping element following the currently specified element in the list.

**F8**

**Selection:** With **F8** you select the currently displayed clamping element.

**ESC**

With **ESC** you cancel current processing and return to the selection menu.

Figure 4.2.3-7:  
CNC Milling,  
Clamping fixture  
definition;  
After selection, you  
can also rotate the  
vice  
with **F3**.

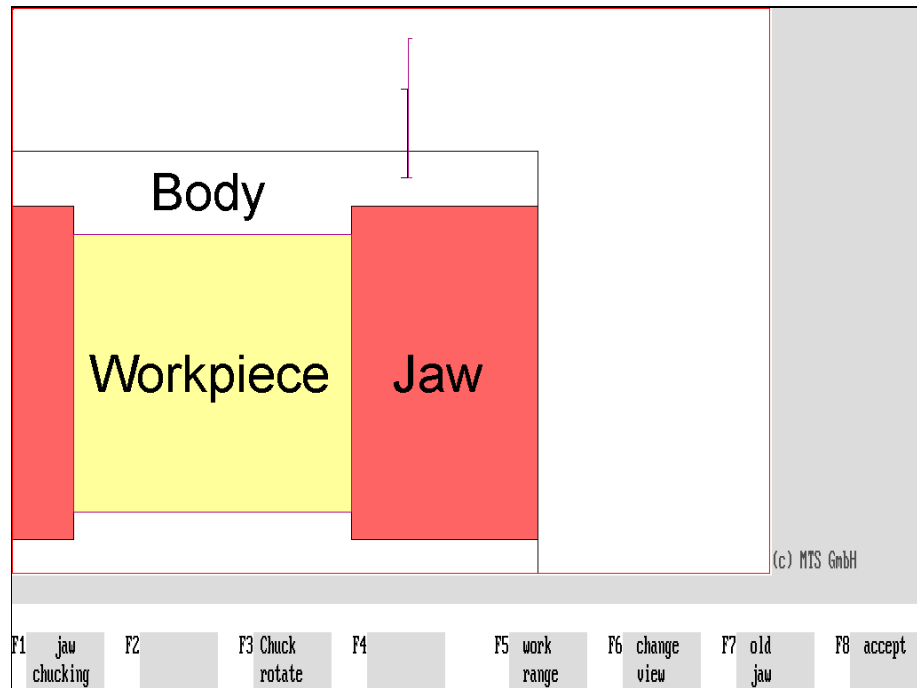
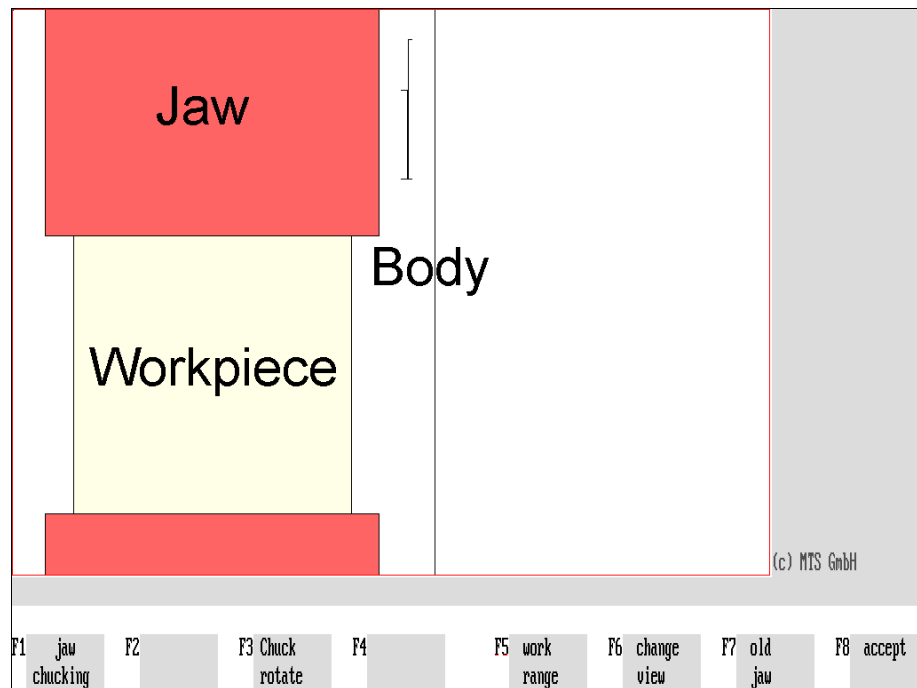
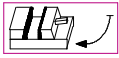


Figure 4.2.3-8:  
CNC Milling,  
Clamping Fixture  
Definition;  
Vice rotated by 90°.



## Rotating the Vice



After vice selection, an intermediate menu is loaded to rotate the vice 90°:

**F1**

**Jaw clamping:** Press **F1** to select the Vice Manager.

**F3**

**Rotate clamping fixture:** Press **F3** to turn the vice 90°. This function can be executed several times consecutively.

**F5**

**Work space:** To display the work space, press **F5**. You can return to the standard display with **F5**.

**F6**

**Change view:** While defining the clamping method, you can change the view onto the workpart with **F6**.


**F7**

**Previous clamping fixture:** Press **F7** to load the preceding clamping fixture again.

**F8**

or



**Confirm:** After having specified the clamping system, return with **F8** or  to the starting situation.

### Note

Should the span of the current vice not be sufficient to clamp the workpart after rotation, an error message is displayed:

**"Vice is too small for the workpart. Nevertheless confirm? (Y/N)"**

**N**

You can cancel processing with **N**.

**Y**

With **Y** the vice is rotated but the workpart is removed and you must then insert a new blank.

**F3**

**Rotate clamping fixture:** You rotate the vice further 90° with **F3**. This function makes it possible to rotate the vice 180° - even if the intermediate position would not be possible.

## Magnetic Plate Clamping



Unlike the vice selection, magnetic plate clamping is loaded proceeding directly from the "Selection of Clamping Systems" menu.

**F2**

**Magnetic plate clamping:** The availability of clamping with magnetic plates is switched on directly with **F2**. The menu for defining the clamping method is then loaded automatically.

Figure 4.2.3-9:  
CNC Milling,  
Clamping fixture  
definition;  
Modular Clamping;  
Menu for editing the  
clamping elements.

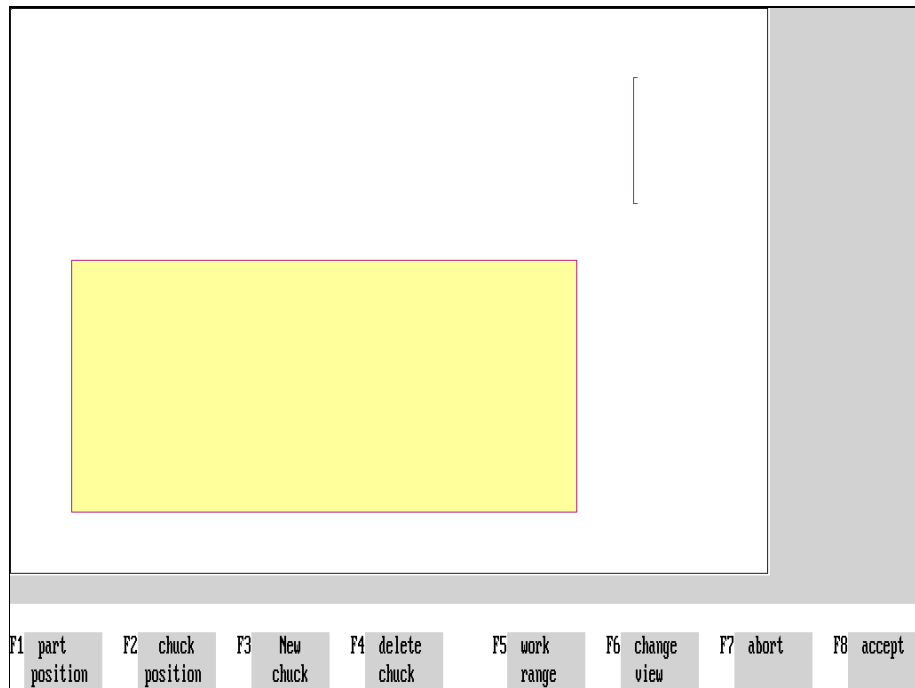
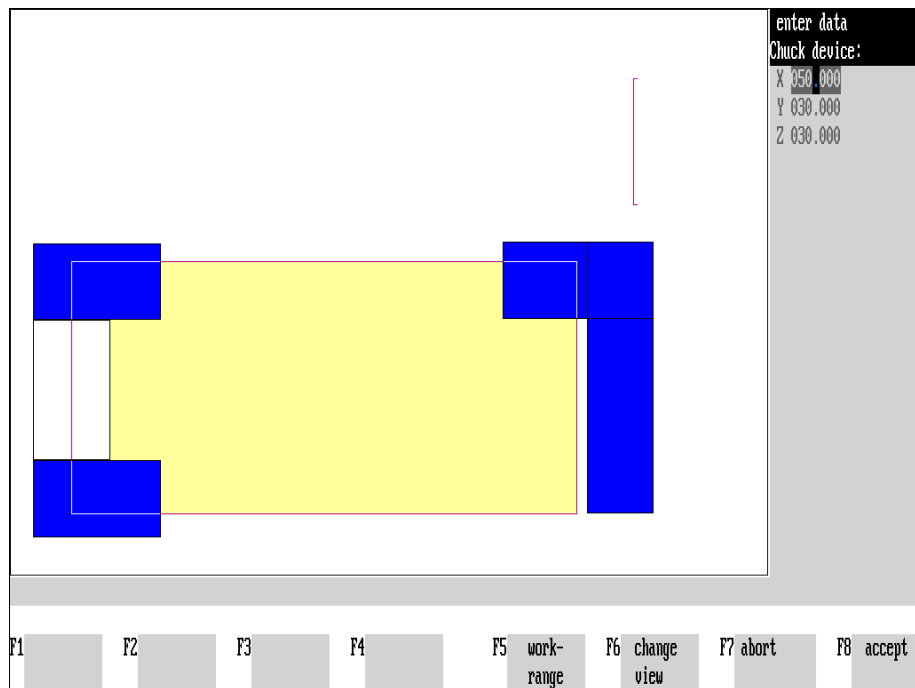
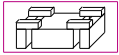


Figure 4.2.3-10:  
CNC Milling,  
Clamping fixture  
definition;  
Modular Clamping;  
Defining the size of a  
clamping element.





## Modular Clamping



When using modular clamping, you first define simple clamping elements (claws, bases, positioning and supporting elements) in processing menu and then combine these to form a modular clamping fixture system .

To keep handling of this type of clamping under control in the CNC Simulator, the elements are defined and displayed only as cuboids (max. external dimensions of the upper clamping device). Collision monitoring is based on these cuboid dimensions.

### Processing Menu

After selecting modular clamping with **F3**, the processing menu for modular clamping is loaded:

**F1**

**Workpart position:** Press **F1** to change the position of the workpart.

**F2**

**Position clamping element:** Press **F2** to change the position of the clamping element.

**F3**

**New clamping element:** Press **F3** to define a new element.

**F4**

**Delete clamping element:** Press **F4** to delete the clamping elements one by one again.

**F5**

**Work space:** To have the entire work space displayed, press **F5**.

**F6**

**Change view:** Change the view of the workpart with **F6** during processing of the clamping elements.

**F7**

or

**ESC**

**Cancel:** Press **F7** or **ESC** to cancel processing.

**F8**

or

**↵**

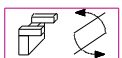
**Confirm:** Press **F8** or **↵** to confirm the defined modular clamping.

### Procedure

The definition of modular clamping consists of two work steps:

- 1.) You define the dimensions of the clamping elements and
- 2.) You position each element to clamp the workpart.

### New Clamp. Elem.



After you have called the "New clamping element" function with **F3**, you can define the size of the clamping element in the following menu.

For this purpose enter the corresponding values in the input fields in the information column (right-hand-side section of the screen):

**↑**

or

**↓**

You move from input field to input field with the cursor keys **↑** or **↓**.

**→**

or

**←**

You move the cursor to a certain input position with the cursor keys **→** or **←**.

### [Value Statement]

The values are entered as numbers, a decimal point with **<, >** or **<.>**.

**F7**

or

**ESC**

**Cancel:** Press **F7** or **ESC** to cancel processing.

**F8**

**Confirm:** After confirming your value entries with **F8**, the clamping element is displayed and the menu for positioning this clamping element is loaded automatically.

### Note

Cf. [Section 4.1.2](#) for the "Work space" and "Change view" menu functions.

Figure 4.2.3-11:  
CNC Milling,  
Clamping fixture  
definition;  
Positioning a clamping  
element of modular  
clamping.

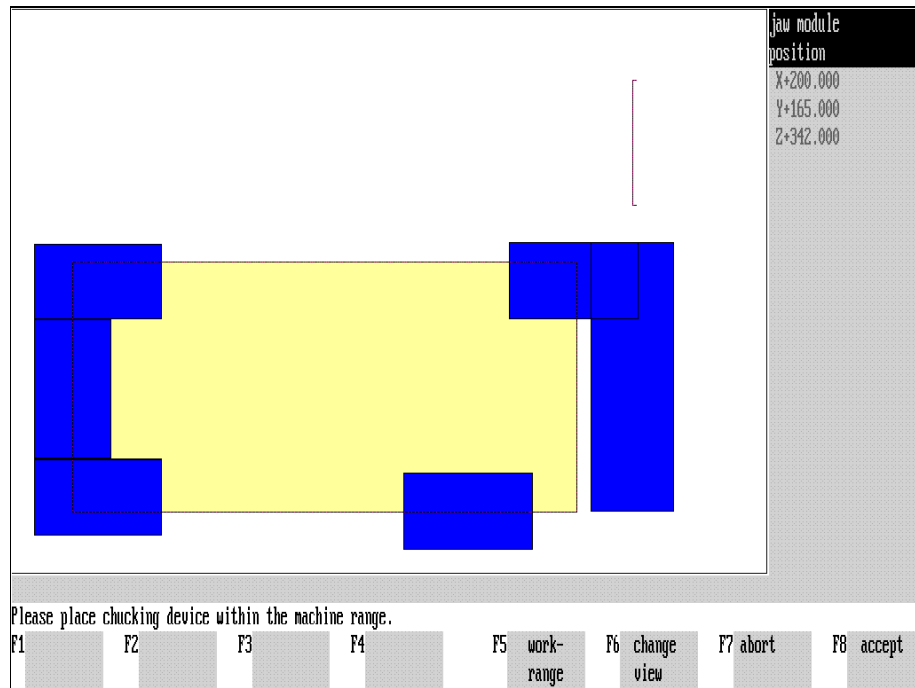
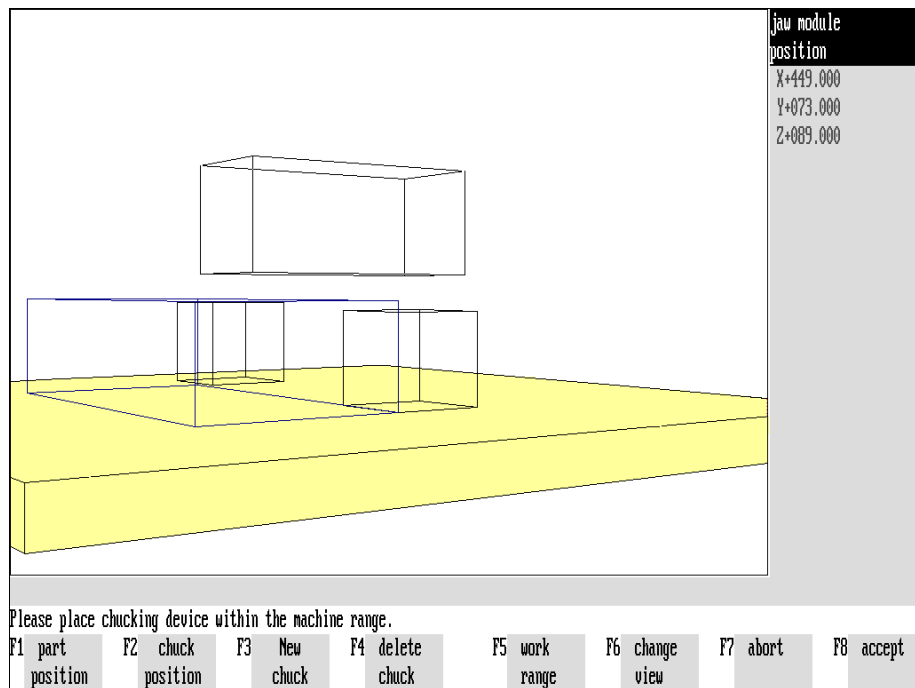
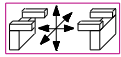


Figure 4.2.3-12:  
CNC Milling,  
Clamping fixture  
definition;  
Clamping elements of  
modular clamping  
displayed in "Office  
perspective".



## Element Position





As mentioned, individual clamping elements can be combined to form a clamping fixture. However, clamping elements placed on top of each other cannot be identified individually in a top view presentation.

In the next work step you clamp the workpart with an already defined clamping element. For this purpose you need to position the clamping element at the desired place of the machine table. Parallel to this, the coordinates of the current position of the center point of the surfaces (related to machine zeropoint) are displayed in the information column:



or





You move the clamping element in X direction (according to G17) with the cursor keys  or .



or





You move the clamping element in Y direction (according to G17) with the  or  **on the numerical keypad** of the PC keyboard.



or




You move the clamping element in Z direction (according to G17) with the cursor keys  or .



and





The clamping element moves with larger steps with the key combination  and [cursor key].



or





**Cancel:** You cancel processing with  or ; the just defined clamping element is then deleted.



or



**Confirm:** Press  or  to confirm the current position of the clamping element. The processing menu for modular clamping is then loaded automatically.




When positioning a clamping element note that an eventual collision message always needs to be confirmed first (with any key) before processing can be continued.

## Note

**With the plane selection G18 or G19, the assignments of the coordinates and of the corresponding cursor keys change accordingly.**

Cf. [Section 4.1.2](#) for the "Work Space" and "Change View" menu functions.

## Position Change



The definition of the clamping element and its clamping position are made in immediate succession without intermediate steps. However, the CNC Simulator also enables the position of a clamping element to be changed later on. This function is called from within the processing menu of the modular clamping with  **"Clamping element position"**.

Determining a new clamping position of a clamping element takes place as described above. However, the desired element needs first to be selected with the cursor keys:



or





Select the desired clamping element with the cursor keys  or . The currently selected element is highlighted in color.



or





**Cancel:** Press  or  to cancel the selection and return to the processing menu of the modular clamping.

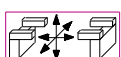


or



**Confirm:** Press  or  to confirm your selection of the clamping element. In the following menu you determine the clamping position of this element as described above.

## Delete Element



In the "Delete Clamping Element" function, the operation sequence:

'Selection of a clamping element => processing'  
is analogous to that of the position change procedure.

Figure 4.2.3-13:  
CNC Milling,  
Schematic of the  
menu sequence for  
defining the clamping  
method.

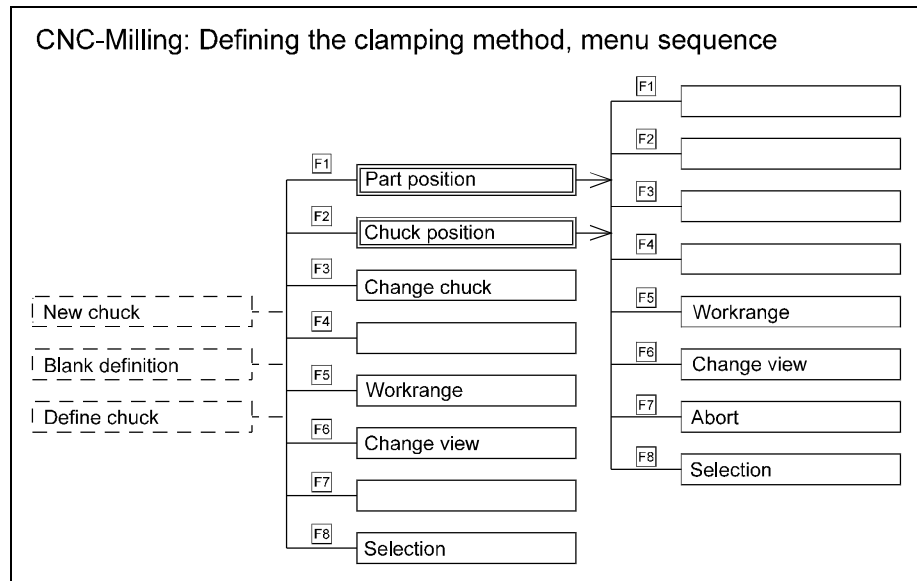
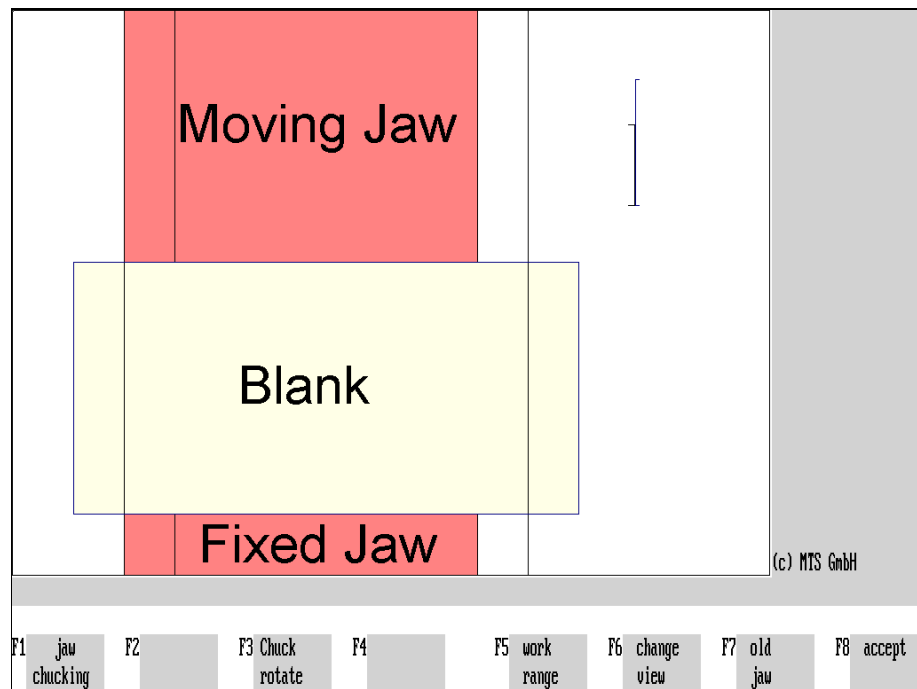
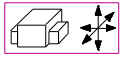


Figure 4.2.3-14:  
CNC Milling,  
Selection menu for  
defining the clamping  
method.



## Defining Clamping



Clamping can be defined by entering the exact position of the workpart or the clamping fixture with the menu item "New clamping" of the "Workpart/clamping fixture definition" menu.

Furthermore, the selection menu for clamping definition is loaded automatically after the blank has been defined and the clamping fixture selected:

F1

**Workpart position:** Press **F1** to position the workpart.

F2

**Clamping fixture position:** Press **F2** to position the clamping fixture. This function cannot be selected for magnetic plate clamping.

F3

**[Change clamping fixture]:** This menu item is available only in the workpart/clamping fixture menu as well as after the blank change. Change the clamping fixture with **F3**.

F5

**Work space:** If you want to display the work space, press **F5**.


F6

**Change view:** To change the perspective view of the workpart, press **F6**.

F8

or



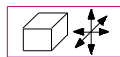
**Confirm:** After the clamping has been defined as desired, press **F8** or  to return to the starting menu "Workpart and clamping fixture definition".

ESC

With **ESC** you cancel processing and return to the starting situation.

### Note

#### Determine Position





Cf. [Section 4.1.2](#) for the "Work space" and "Change view" menu functions.

Provided the selected clamping system allows, the workpart and the clamping fixture can be repositioned with the cursor keys. Parallel to this, the coordinate values of the current position of the left lower corner of the bottom of the blank (referring to the machine zeropoint) are displayed in the information column. Since both work steps are analogous, they are described here together. When positioning the clamping fixture, the workpart comes with:



or



You move the workpart/clamping fixture in X direction (according to G17) with the cursor keys  or .



or





You move the workpart/clamping fixture in Y direction (according to G17) with the keys **9** or **1** **on the numerical keypad** of the PC keyboard.



or




You move the workpart/clamping fixture in Z direction (according to G17) with the cursor keys  or .



and



The workpart/clamping fixture moves with larger steps with the key combination  and [cursor key].

F7

or


ESC

**Cancel:** Press **F7** or **ESC** to cancel the processing and return to the starting situation.

F8

or



**Accept:** Press **F8** or  to confirm the current position of the workpart/clamping fixture. The selection menu for defining the clamping is then loaded again.



Please note when positioning the workpart/clamping fixture that an eventual collision message need to be confirmed first (with any key) before processing can be continued.

### Note

**In the plane selection G18 or G19, the assignments of the coordinates and thus also of the cursor keys change accordingly.**

Figure 4.2.3-15  
CNC Milling;  
Reclamping a  
workpart: vertically; by  
reclamping, it is  
possible to machine  
opposite sides.

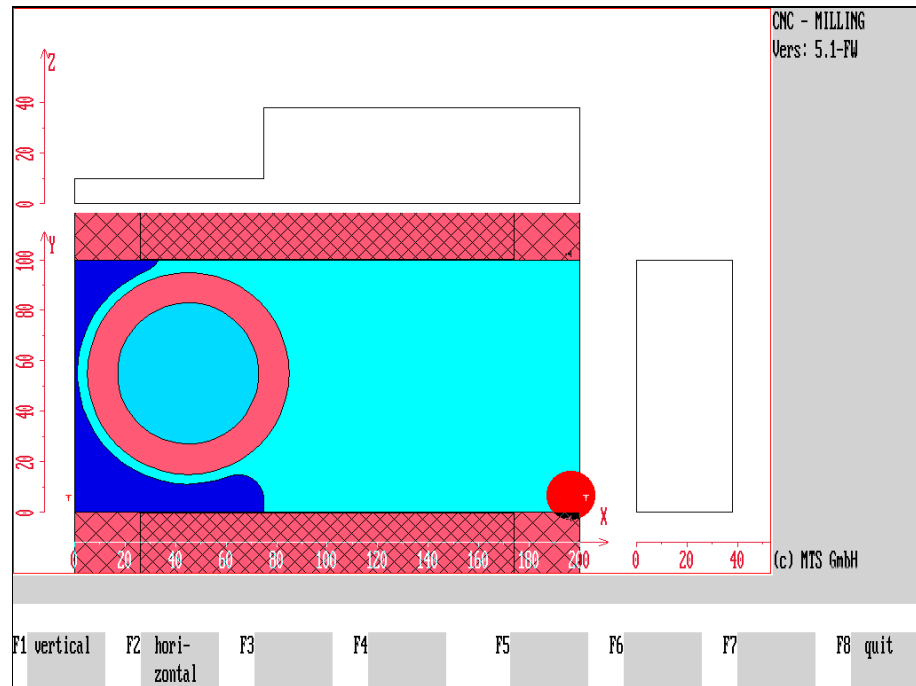
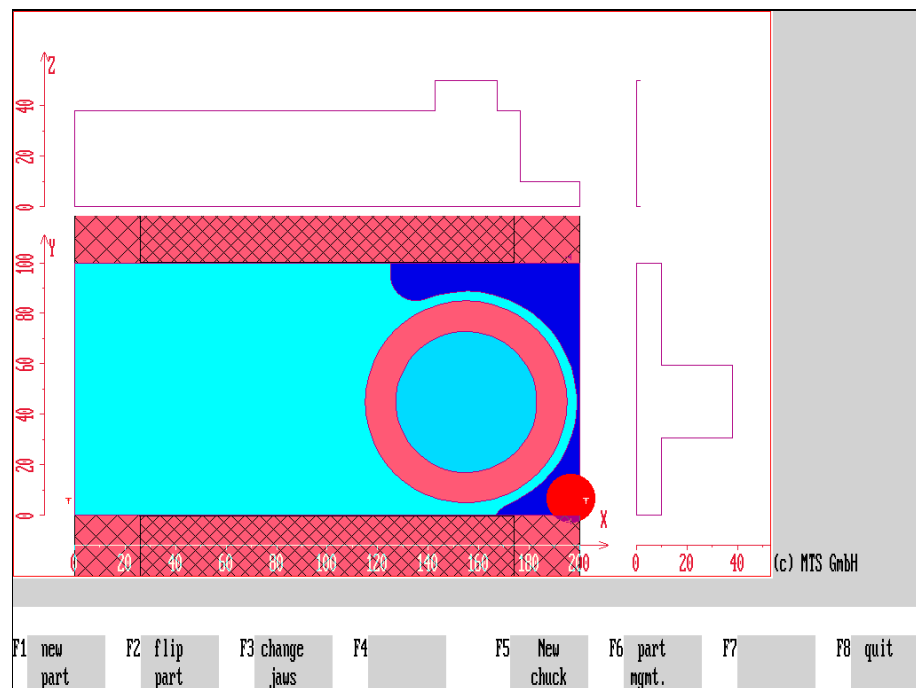


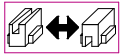
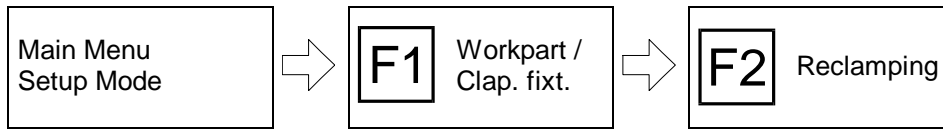
Figure 4.2.3-16:  
CNC Milling,  
Reclamping; if the  
"Reclamping" function  
is performed several  
times in the  
combination: vertically  
and horizontally, thus a  
total of four clamping  
possibilities exist.



## Reclamping the Workpart

To machine both sides of the workpart it can be reclamped as often as required and then machined again.

### Selection:



The "Reclamping" function is called up with **F2** from the menu for "Workpart and clamping fixture definition". In the following menu you define how to rotate the workpart:



**Vertically:** The workpart is turned vertically 180° with **F1** (axis of rotation through the center point of the workpart and parallel to the Y axis). The system then switches automatically back to the "Workpart and clamping fixture definition" menu and the reclamped workpart is shown on the screen.



**Horizontally:** The workpart is turned horizontally 180° with **F2** (axis of rotation through the center point of the workpart and parallel to the X axis). The system then switches automatically back to the "Workpart and clamping fixture definition" menu and the reclamped workpart is shown on the screen.



or



**Return:** Press **F8** or **ESC** to return to the "Workpart and clamping fixture definition" menu without reclamping the workpart.

From the "Workpart and clamping fixture definition" menu, press **F8** to return to the processing level of the Setup Mode.

### Note

If you execute the "Reclamping" function several times consecutively in the combination of vertically and horizontally, then you can specify a total of four clamping alternatives for the workpart.

Figure 4.2.4-1:  
CNC Milling,  
Schematic of the  
menu sequence for  
the Clamping Fixture  
Manager, simplified  
presentation.

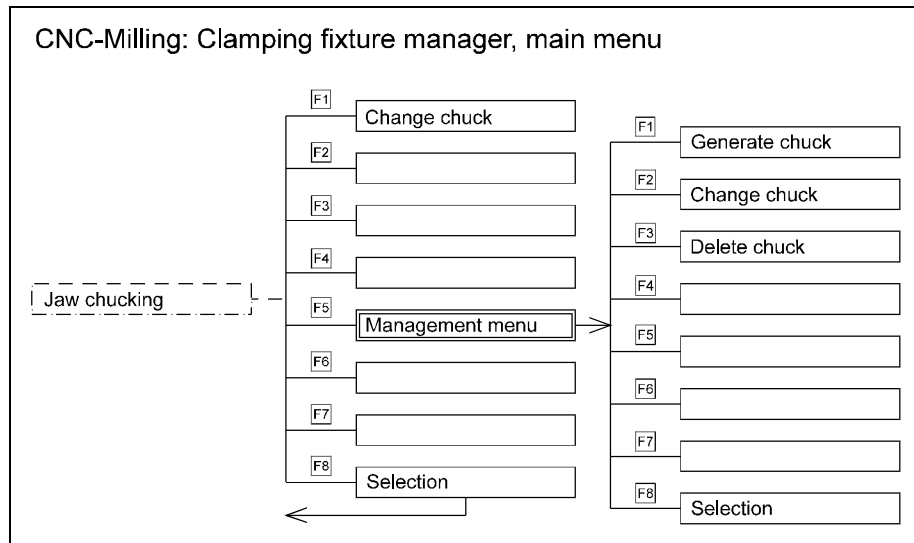
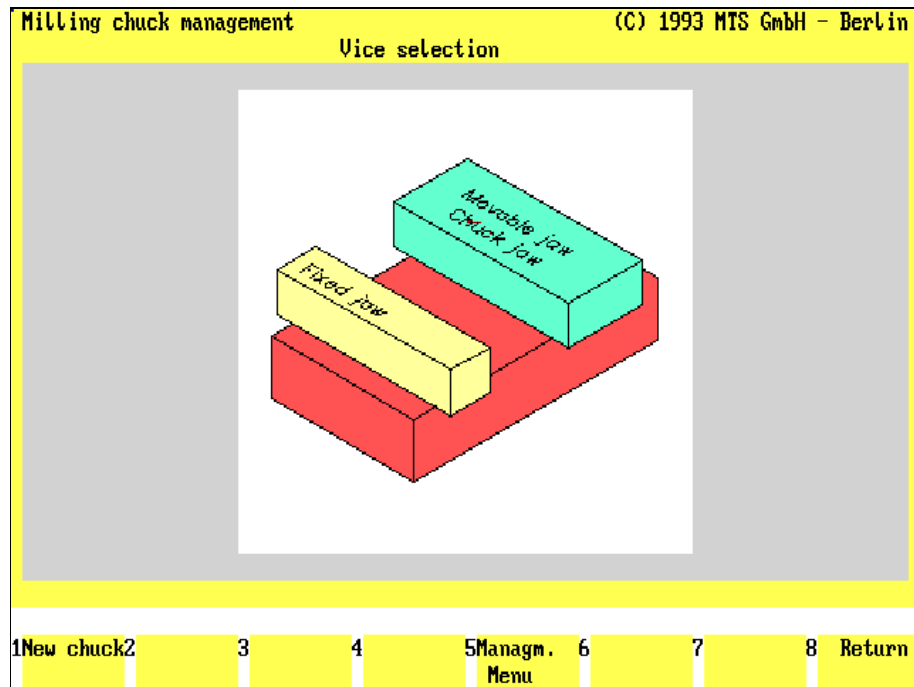
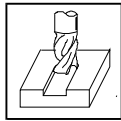


Figure 4.2.4-2:  
CNC Milling,  
Clamping Fixture  
Manager, Main menu.





## 4.2.4 CNC Milling: Clamping Fixture Manager

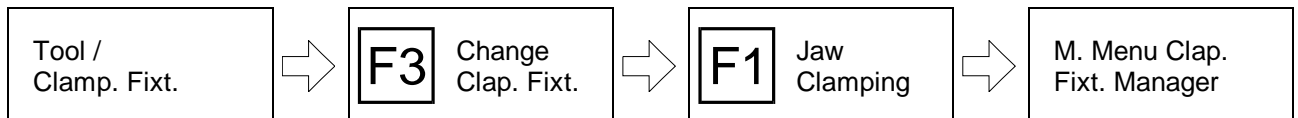


The CNC Milling Simulator provides an separate Clamping Fixture Manager for the "Jaw Clamping" type of clamping for defining, modifying and managing different vices.



**Should you have selected some other type of clamping, the Clamping Fixture Manager is skipped.**

**Selection:**



The Clamping Fixture Manager is called under the functions of "Change clamping fixture" and then "Jaw clamping".

### Clamping Fixture Manager: Main Menu

Screen Display

The layout of the screen display was described above in [Section 4.2.3](#).

The upper part of the screen is reserved for graphics, information and input fields, whereas the labeled function keys in the lower screen margin allow the following processing options:

**F1**

**Change clamping fixture:** You can insert a new clamping element with **F1**.

**F5**

**Management menu:** To generate a new clamping element or to modify or delete an existing one, press **F5**. To avoid unauthorized manipulation of the Clamping Fixture Manager, these functions should be protected with a "password" (cf. configuration instructions).

**F8**

**Return:** Press **F8** to confirm the entered combination of clamping elements and switch to the menu for defining the clamping.

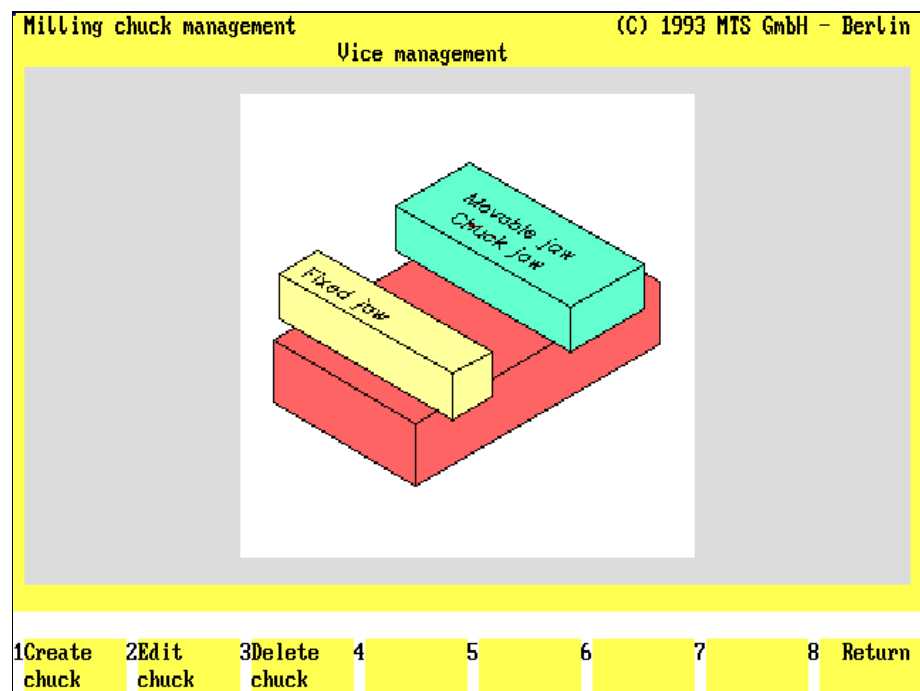
**ESC**

Press **ESC** to cancel processing and return to the starting situation.

**Note**

The Clamping Fixture Managers for the CNC Turning and CNC Milling Simulators are identical in structure and are normally also identical in menu structure and handling. For reasons of clarity, they are therefore described as follows systematically according to the practical handling aspects; to avoid repetition, only the differences are discussed in case of recurring analogous processing steps.

Figure 4.2.4-3:  
CNC Milling,  
Clamping Fixture  
Manager;  
Management Menu



## Clamping Fixture Manager: Management Menu

The management of the clamping elements is called with **F5** proceeding from the main menu of the Clamping Fixture Manager.

### Password

To avoid unauthorized manipulation of the Clamping Fixture Manager, this function can be protected with a "password" (cf. configuration instructions). In this case enter the password and confirm your entry with **↵**. The Clamping Fixture Manager is then loaded:

**F1**

**Create clamping fixture:** To define a new clamping fixture, press **F1**.

**F2**

**Modify clamping fixture:** To modify an existing clamping fixture, press **F2**.

**F3**

**Delete clamping fixture:** To delete a clamping fixture, press **F3**.

**F8**

or

**ESC**

**Return:** Press **F8** or **ESC** to return to the starting situation.

### Note

**Please note that after editing the clamping fixture file, certain entries in existing setup sheets (e.g. data on clamping depth etc.) or existing NC programs may also need to be changed.**

**Therefore edit the clamping fixture file only as far as necessary and then check the setup sheets and NC programs referring to these clamping elements.**

**When in doubt, create a new clamping element!**

Figure 4.2.4-4:  
CNC Milling,  
Clamping Fixture  
Manager;  
Definition of a new  
vice.

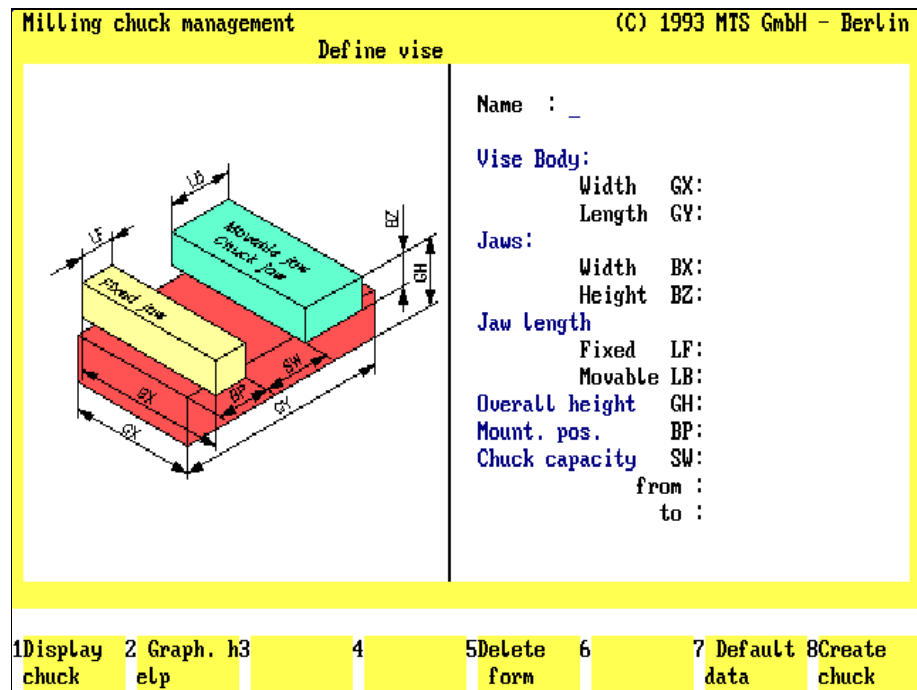
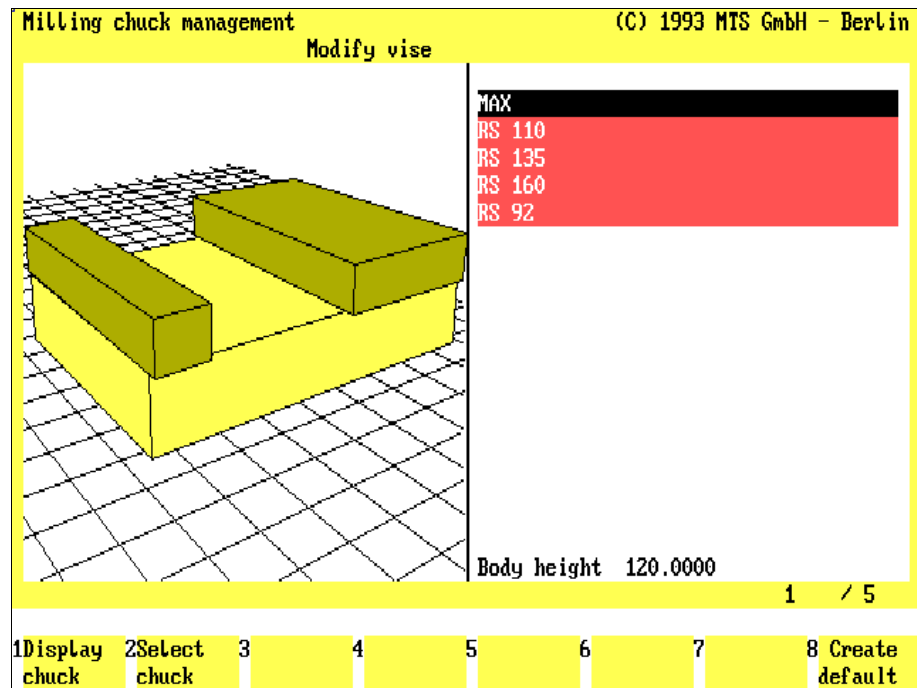


Figure 4.2.4-5:  
CNC Milling,  
Clamping Fixture  
Manager;  
Specify data: select  
the desired vice in  
the selection menu.



## Clamping Fixture Manager: Define Clamping Element

The processing menu for the definition of a new vice is called with **F3** proceeding from the Management menu of the Clamping Fixture Manager. The input menu is then loaded for the input of names and dimensions:

**F1**

**Clamping Fixture graphic:** As soon as all parameters have been specified, you can display a graphic of the clamping element on the screen with **F1**.

**F2**

**Help graphic:** An auxiliary graphic for explanation of the input parameters is displayed with **F2**.

**F5**

**Delete form:** All entries in the input fields are deleted again with **F5**.

**F7**

**Copy data:** If you want to use the geometrical data of an already existing clamping element, press **F7**.

**F8**

**Create clamping fixture:** With **F8** you accept your data to create a new clamping element.

**ESC**

With **ESC** you return to the management menu.

### Procedure

When entering the parameter values etc., the following operating steps are to be considered:



or



You move from input field to input field with , **TAB** or and **TAB**.

[Value Statement or Name]

The values are given as numbers, the decimal point with <, > or <.>.

**F8**

Confirm your entries with **F8** only after all parameters have been entered.



The input menu with empty input fields is then loaded to allow the definition of a further clamping element.

With **ESC** you return to the management menu.

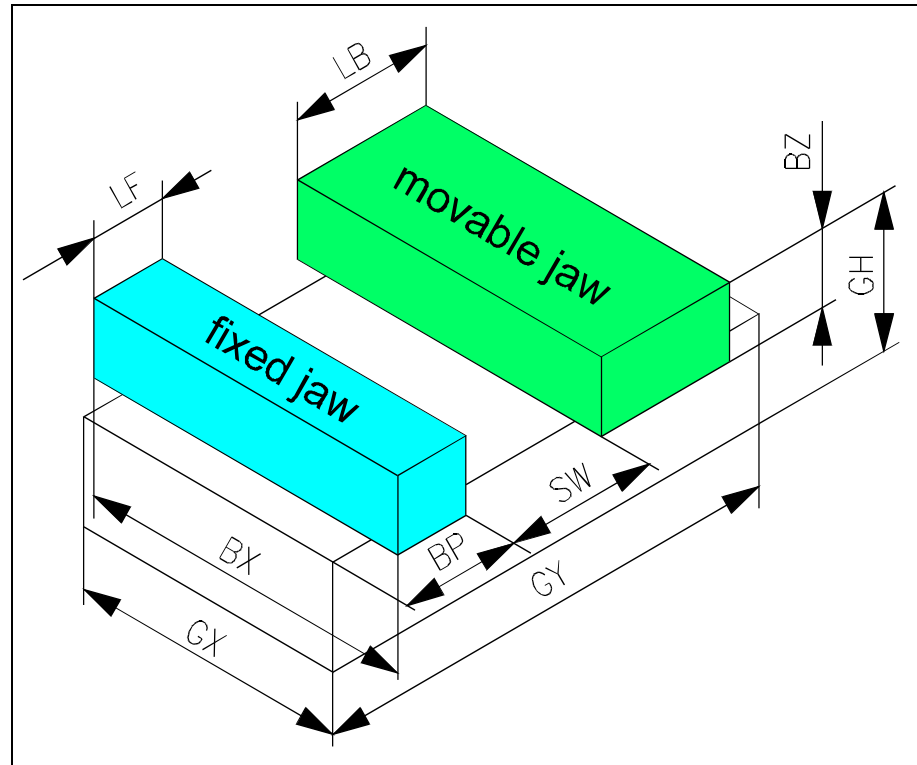
## Copy Data Record

If you want to proceed from an already existing data record for the definition of a new clamping element, you can have this read-in and then modify the parameters as desired:

The "Copy data" function is called with **F7** in the input menu:

The existing clamping elements of the currently selected group are displayed in the upper part of the screen in the selection window (cf. above, Change Clamping Fixture).

Figure 4.2.4-6:  
CNC Milling,  
parameters for  
determining a vice.



F1

F2

F8

**Procedure**

**Show clamping fixture:** With **F1** you call the detailed information for the currently marked clamping element.

**Select clamping fixture:** To select from the complete list of existing clamping elements those meeting certain geometrical criteria, you can have these singled out and so obtain a new list.

**Copy clamping fixture:** With **F8** you confirm the selected clamping element.

For selecting the existing data record to be read-in, please note the following operation steps:

↑

or

↓

Mark the desired clamping element in the selection window with the cursor keys **↑** or **↓** or **PageUp** or **PageDown**.

F8

With **F8** the corresponding data are copied to the input menu. You can then overwrite the data to define a new clamping element.

ESC

With **ESC** you cancel processing and return to the starting situation.

**Vice Input Parameters**

In the Clamping Fixture Manager, a vice consists of the elements: body, fixed and moving clamping jaw (cf. Figure). To determine these enter the following dimensions:

<b>General Entries</b>	Name:	arbitrary entry
	<b>Body:</b>	
<b>Dimensions</b>	Width:	GX (in GX > 0)
	Length:	GY (in GY > 0)
	<b>Jaws (both fixed and moving) :</b>	
	Jaw width:	BX (in BX > 0)
	Jaw height:	BZ (in BZ > 0)
	<b>Jaw length: (differentiated according to fixed and moving):</b>	
<b>Assignment of Spindle Heads</b>	Length of the fixed jaw:	LF (in LF > 0)
	Length of the moving jaw:	LB (in LB > 0)
	<b>Total height: (body and clamping jaw) :</b>	
	Total height:	GH (in GH > 0)
	<b>Mounting position: (fixed clamping jaw related to body) :</b>	
	Mounting position:	BP (in BP · LF)
	<b>Span: (between fixed and moving jaw) :</b>	
	Span:	SW from (= minimum) to (= maximum)





## Clamping Fixture Manager: Modify Clamping Element

The menu to modify an existing clamping element is called with **F2** proceeding from the management menu of the Clamping Fixture Manager.

The procedure takes place in two work steps:

1. First select the clamping element you want to modify, and
2. then change the parameters as necessary:

### Select Element

For the selection, mark the desired clamping element in the list of the existing vices:



or



Select the clamping element with the cursor keys **↑** or **↓** or **Page↑** or **Page↓**.



With **ESC** you cancel processing and return to the starting situation.



**Modify:** With **F8** you confirm the selected clamping element and go on to the input menu to change detailed data.

### Edit Data

The input menu for editing data is analogous to the input menu for the definition of a new clamping element (cf. preceding section): to change entries, first activate the corresponding input field and then overwrite the existing entries:



or



You move from input field to input field with **TAB**, **←** or **←<sub>shift</sub>** and **TAB**.



or



You move the cursor to a certain input position with the cursor keys **→** or **←**.

### [Value Statement or Name]

The values are given as numbers, the decimal point with **<, >** or **<.>** (the parameters are discussed in the previous section).



With **ESC** you cancel processing and return to the starting situation.



**Modify clamping fixture:** With **F8** you confirm your modifications.

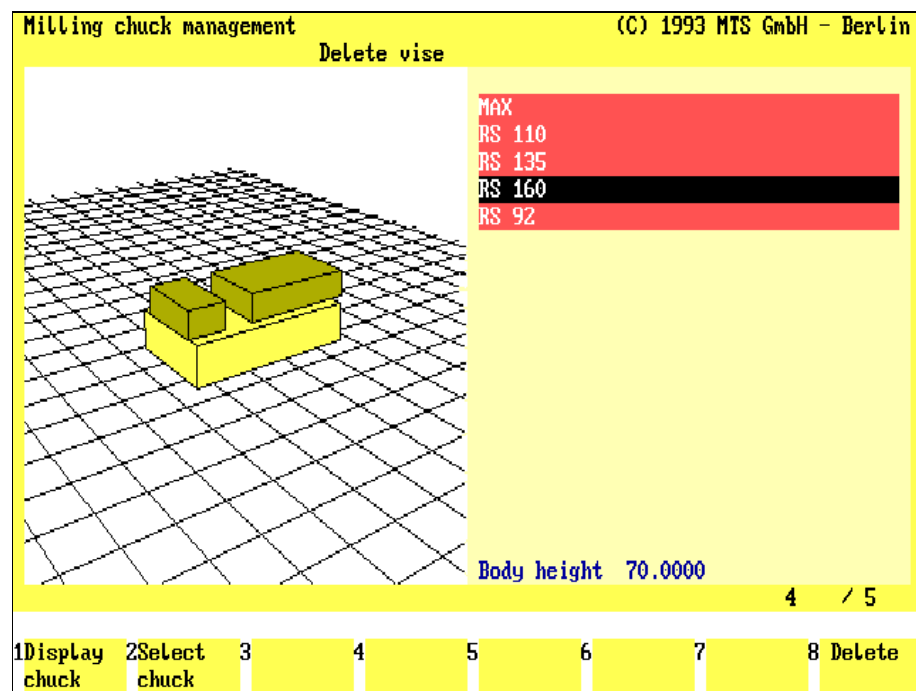
### Note

**Please note that after editing the clamping fixture file, certain entries in existing setup sheets (e.g. data on clamping depth etc.) or existing NC programs may also have to be changed.**

**Therefore edit the clamping fixture file only as far as necessary and check subsequently the setup sheets and NC programs referring to this clamping element.**

**When in doubt, create a new clamping element!**

Figure 4.2.4-9:  
CNC Milling,  
Clamping Fixture  
Manager;  
Delete vice.



## Clamping Fixture Manager: Delete Clamping Element

The menu for deleting an existing clamping element is called with **F3** proceeding from the Management menu of the Clamping Fixture Manager.

Processing consists of two work steps:

- a.) First select the clamping element to be deleted, and
- b.) then confirm the control query to delete this element:

### Select Element

A clamping element is selected - as described before - from the list of existing vices:



or



Mark the desired vice in the selection window with the cursor keys **↑** or **↓** or **Page↓** or **Page↑**.

**ESC**

With **ESC** you cancel processing and return to the starting situation.

**F8**

**Delete:** Confirm your selection with **F8**.

### Control Enquiry

Before the vice is deleted from the clamping fixture file, there is a further control enquiry: **Do you want to delete the clamping element?**

**F7**

**No:** Press **F7** to return to the selection menu.

**F8**

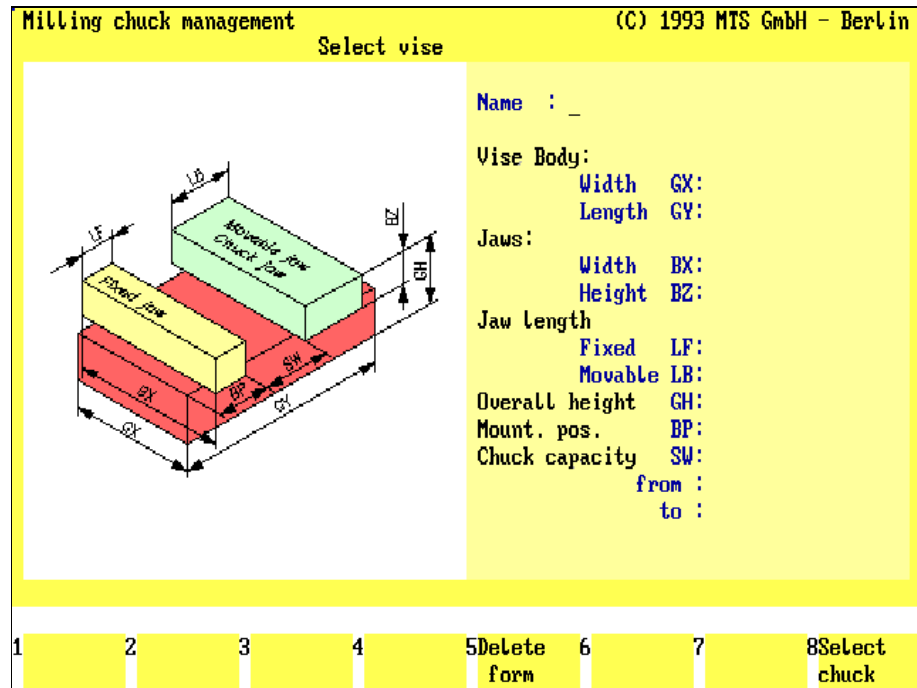
**Yes:** Press **F8** to delete the vice.

### Note

**Please note that after a clamping element has been deleted, corresponding entries in existing setup sheets need to be changed as well.**

**Therefore, edit the clamping fixture file only as far as necessary and check subsequently the setup sheets and NC programs referring to this clamping element.**

Figure 4.2.4-10:  
CNC Milling,  
Clamping Fixture  
Manager;  
Select clamping  
element.



## Clamping Fixture Manager: Select Clamping Elements

With this function it is possible to select from an extensive clamping fixture list the elements with specific geometrical criteria, such as a span or a jaw of a certain size etc.

**F2**

**Select clamping fixture:** This function is called with **F2** in the management menu:

The selection menu is analogous to the input menus (cf. preceding sections):

**F5**


**Delete form:** Press **F5** to delete all entries in the input fields.

**F8**

**Select clamping fixture:** With **F8** you confirm your entries to select the corresponding clamping fixture from the list.

### Procedure

The loaded menu does not include any entries: enter in the corresponding input fields **only those dimensions** which are to serve as selection criteria:

 or **TAB**

You move from input field to input field with , **TAB** or  and **TAB**.

[Value Statement or Name]

The values are entered as numbers, the decimal point with <, > or <.>. Further input options have been discussed above (cf. CNC Turning, Clamping Fixture Manager).

**ESC**

With **ESC** you cancel current processing and return to the management menu.

**F8**

Confirm your entries with **F8** only after the input of the selection criteria has been completed.

You then automatically return to the preceding processing situation, the new list of the vices now only contains those meeting the selection criteria.



To obtain a complete list of all existing clamping elements again, leave the input fields empty, this means do not enter any data in the input fields.

Figure 4.3-1:  
CNC Turning,  
Schematic of the tool  
and zeropoint  
definition.

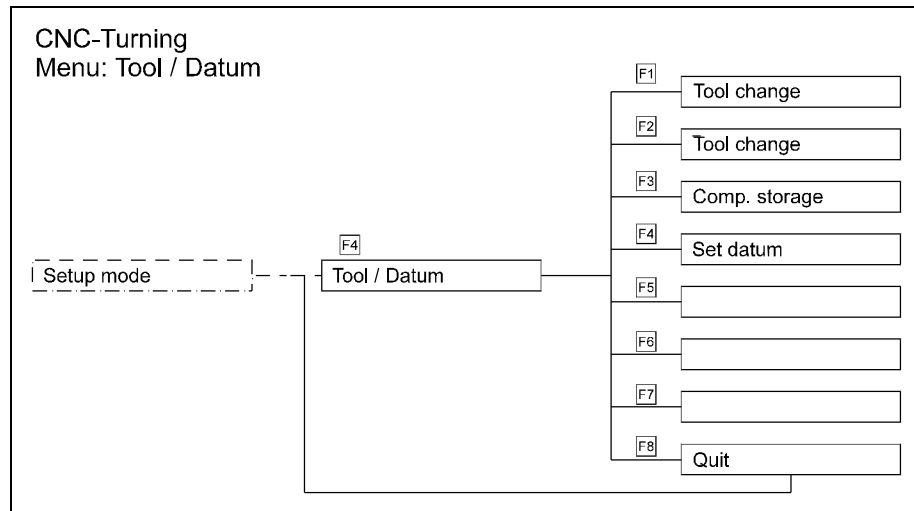


Figure 4.3-2:  
CNC Turning,  
Setup Mode;  
"Tool and zeropoint  
definition" menu.

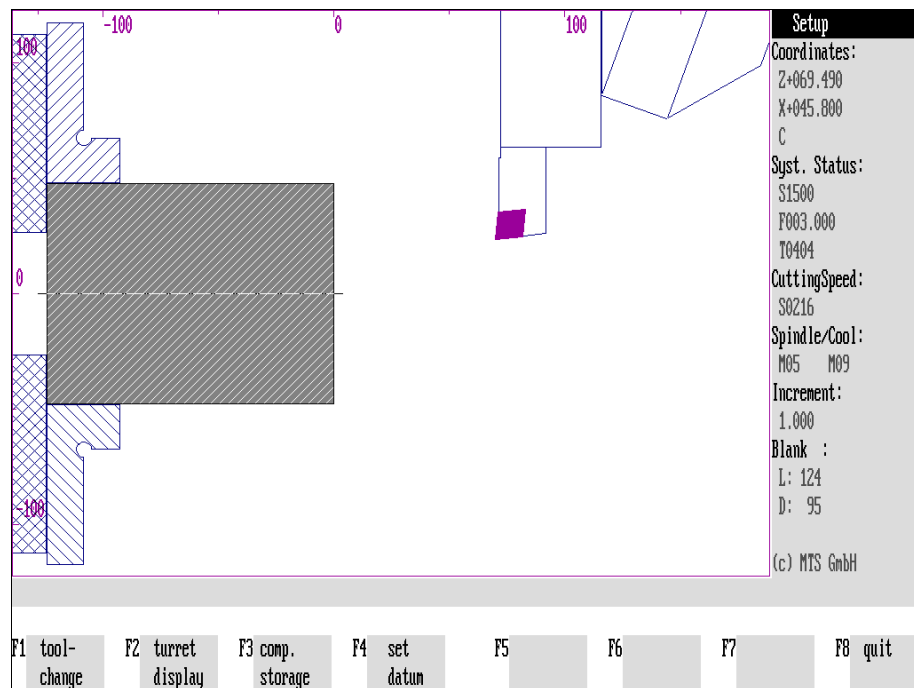
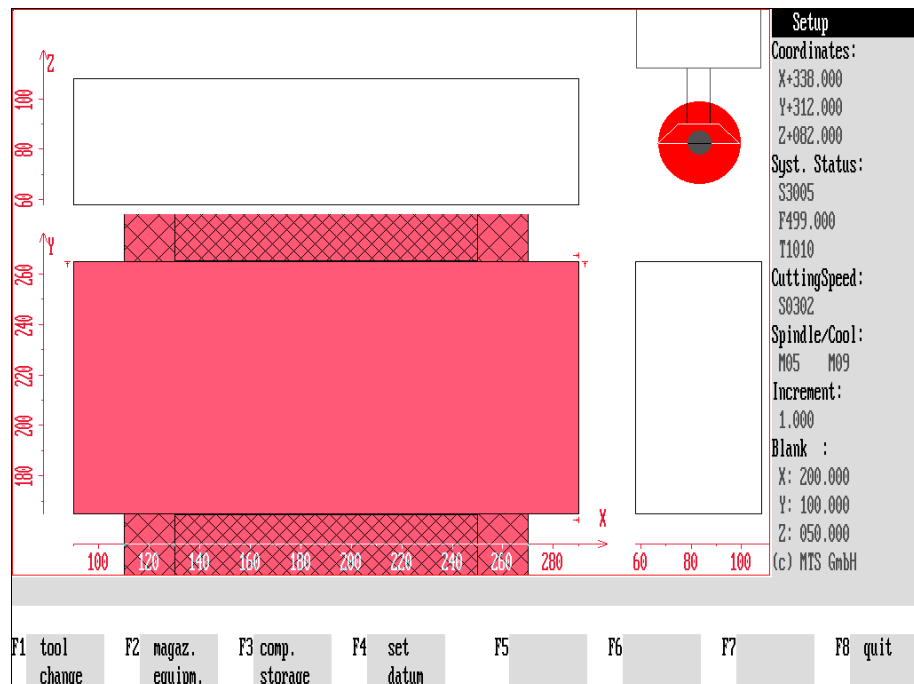


Figure 4.3-3:  
CNC Milling,  
Setup Mode;  
"Tool and Zeropoint  
Definition" menu.



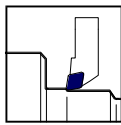
## 4.3 Tooling and Tool Change

For turning and as well as for milling, the system modules for the tools, tool carriers and tool changing systems have been considerably extended, to allow the CNC Simulator be set up, to a great extent, like a machine tool.

The most important features are:

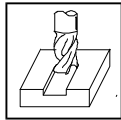
- a separate **tool manager**, for defining and managing the tools, reversible tips, tool holders and tool adapters [according to DIN],
- configuration of different **tool carriers**,
- a maximum of **16 turret positions** or **99 magazine pockets**,
- **compensation value registers**, to determine and assign tool compensations with and without finishing allowances,
- determining different **workpart zeropoints** which can be activated as required.

### CNC Turning



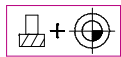
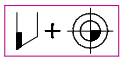
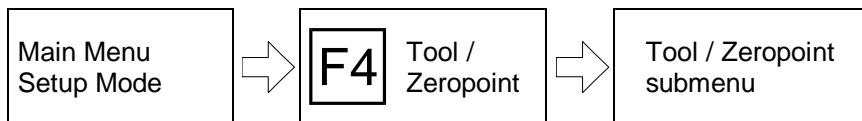
The lathe can be equipped with six different types of turrets (cf. Configuration Instructions) to insert the standardized tool holders in. Each turret has a maximum of 16 tool places.

### CNC Milling



The spindle head, tool adapter and tools [according to DIN] are defined in the configuration or in the Tool Manager. The automatic tool changing system with a magazine of maximum 99 tools guarantees continuous production.

### Selection:



The tool change and equipping the turret or tool magazine takes place in the "Tool/Zeropoint" submenu. It is called up with [F4] within the "Tool/Zeropoint" function proceeding from the main menu of the Setup Mode.

The boxes in the bottom screen margin indicate the available operation steps:

**F1**

**Tool change**: The currently active tool is changed with this function.

**F2**

**Turret/Magazine configuration**: To change the assignment of tools to turret positions or magazine pockets (configuration), press [F2].

**F3**

**Compensation storage**: Load the compensation value register for the tools with [F3].

**F4**

**Set zeropoint**: Should the workpart zeropoint be set manually (e.g. by touching) press [F4].

**F8**

**Return**: [F8] returns you to the main processing level of the Setup Mode.



As a rule, the modules: tool (reversible tip), tool holder and tool adapter are changed completely in the new turret/magazine configuration. To allow further tool combinations, the individual Tool Manager modules can be reassigned.

Figure 4.3.1-1:  
CNC Milling,  
Schematic of the  
menu sequence for  
tool change.

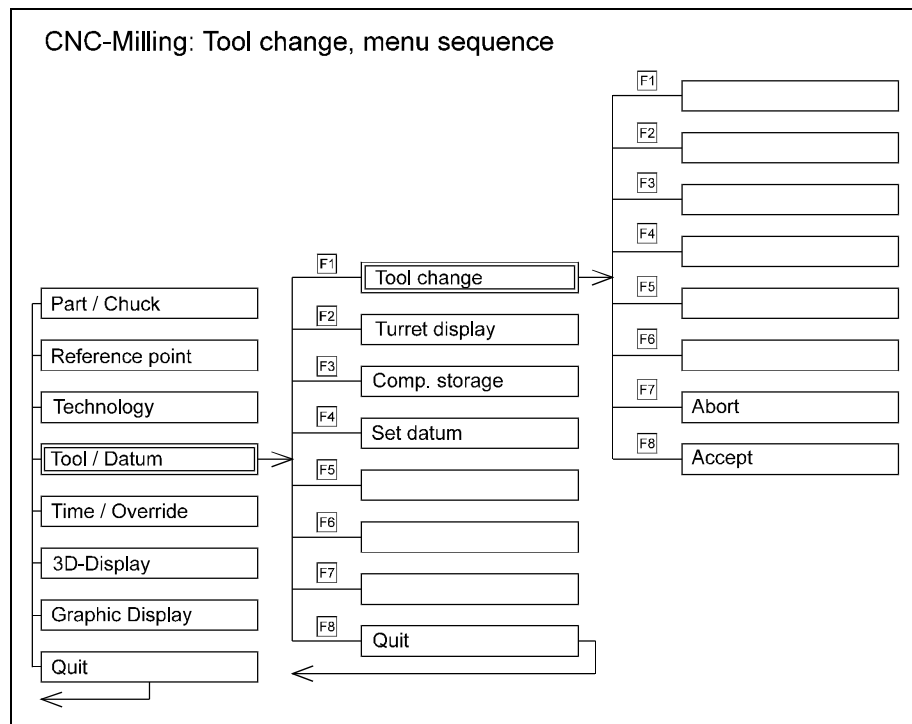
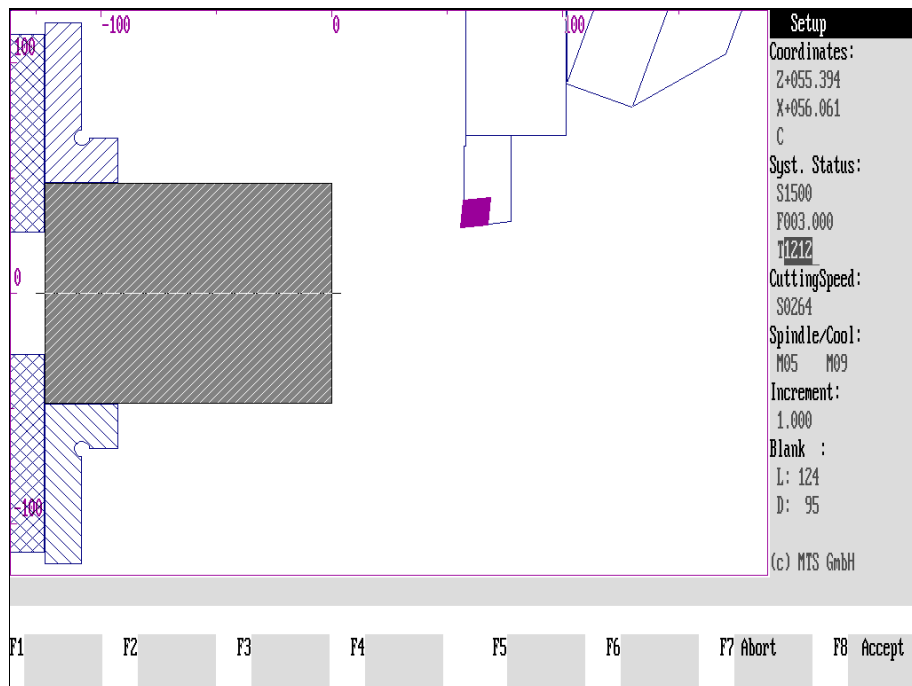


Figure 4.3.1-2:  
CNC Turning,  
Setup Mode;  
Manual tool change.



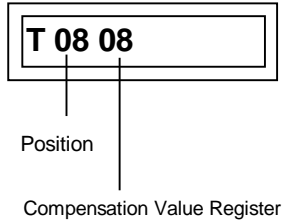


### 4.3.1 Tool Change

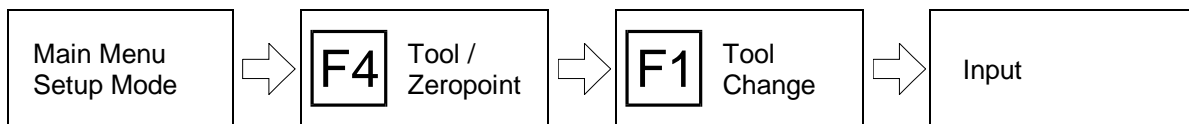
To allow the simulation of a continuous production sequence, the program-controlled tool change takes place fully automatically and is added as down time to the total time of the program run.

In an NC program, the tool change is programmed with the machine command "T", followed by two-digit number indicating the tool location in the turret/magazine as well as further two-digits number indicating the corresponding compensation value register. After this, in the Setup Mode as well as in the Automatic Mode, the current working tool and compensation value register are displayed in the **information column**, under the entry "T".

Naturally, the current working tool and the assigned compensation value register can be changed in the Setup Mode for machining with manual control:



#### Selection:



The manual tool change is called up with **F1** proceeding from the tool and zeropoint definition menu.

#### Procedure

After you have called up the "Tool change" function, the previous entries under "T", in the information column, are highlighted in color and released for modification:

[Entry of the Digits for the Tool Location and for the Compensation Value Register]

Enter the tool location and the number of the corresponding compensation value register **in each case as a two-digit number**: e.g. "0808", "1515" or "0517" (if the compensation value register is assigned to "17").



or



For turning, you can also switch on the turret head from one position to the next with the cursor keys **↑** or **↓**.



or



**Cancel:** Cancel processing with **F7** or **ESC**.



or



**Confirm:** Then confirm your entries with **F8** or **↵**. The tool change takes place at once and you return to the starting situation.

"Switching" the turret to the selected working position is quickly displayed with the intermediate positions of the tool.

#### Note

Whereas in milling the spindle head goes to the tool change point during the tool change, it is possible in turning to swing a specific turret station into the working position within the entire work space. Therefore, pay attention to sufficient distance between the turret and the tool/clamping element, to avoid an collision during tool change. In such a case, a **collision message** is displayed.

#### Cross Reference

Tool change is also possible in the menu for the technology functions (cf. below, [Section 4.5](#)).



When using the CNC keyboard or graphic tablet, the tool change can be activated directly.

Figure 4.3.2-1:  
CNC Turning,  
Setup Mode;  
Schematic of the  
menu sequence for  
processing the Turret  
Configuration.

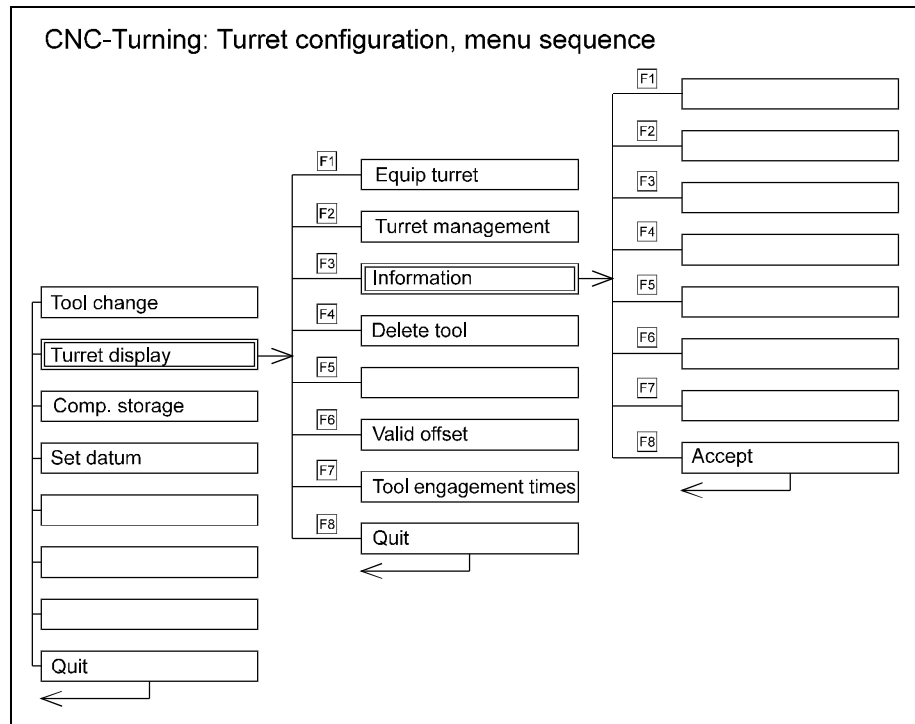
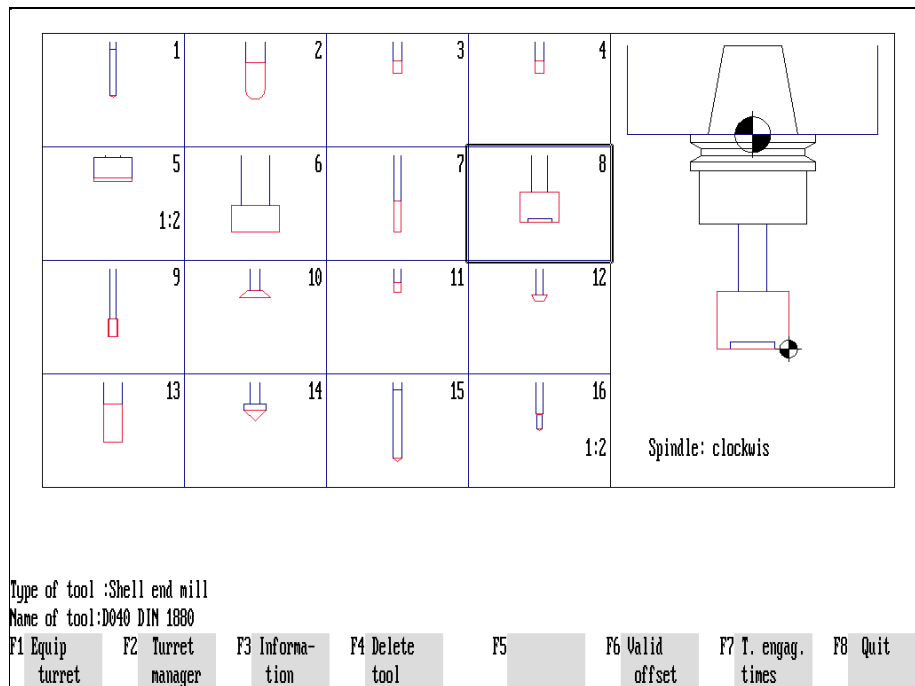


Figure 4.3.2-2:  
CNC Milling,  
Setup Mode;  
Processing menu for  
Magazine  
Configuration.

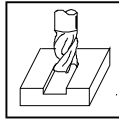


## 4.3.2 Turret/Magazine Configuration

As already stated, there are max. **16 positions** for turrets available and in the tool magazine of the CNC Simulator a maximum of **99 tool pockets with continuous numbers**. This quantity can be limited in the Simulator configuration. In the configuration you also select the type of turret to be used currently, and you can specify certain geometrical and technical data for this.

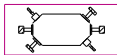
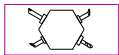
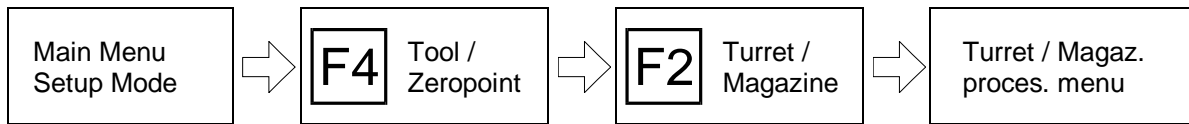
To reduce the setup time, the relevant turret/magazine configuration is recorded in the setup sheet or status file. After each program start or on reading this status file, the corresponding set of tools is loaded automatically.

### CNC Milling



The configuration of the milling Simulator allows various controls of the tool change: with the T command, the magazine is firstl either set to the position for the tool change and this is then executed with the command M06, or both operations are effected with the T command.

### Selection:



The Turret/Magazine Configuration is called with **F2** proceeding from the tool and zeropoint definition menu.

In turning, the tool changing point is approached as well.

### Screen Display

The menu of the turret or magazine configuration shows the positions/pockets with the inserted tools. In addition, the turret station currently in the working position is marked by a second frame. Correspondingly, the active tool is displayed enlarged and the relevant information is shown (cf. Figure).

The following functions are available for processing:



or



For removing or inserting the tool, the position needs to be determined first. To mark the desired location shift the moving frame field with the cursor keys **→** or **←**.



**Tooling:** To insert a tool (from the tool manager) in the marked location, call up the tool selection with **F1**.



**Management:** If you want to define a new tool or modify or delete an existing one, press **F2** to activate the tool manager.



**Information:** You obtain detailed information on the tool geometry, the tool holder or the tool adapter with **F3**.



**Remove tool:** You remove the tool from the currently selected turret/magazine location with **F4**.



**Valid compensation values:** The compensation values of each inserted tool are entered in the register with the same number with **F6**.



**Engagement times:** You display the engagement times of the tools with **F7**.



or



**Return:** With **F8** or **↵** you conclude the processing and return to the starting situation.

### Cross Reference

The tool manager is discussed in [Chapter 6](#).

Figure 4.3.2-3:  
CNC Turning,  
information on a  
postaxial internal tool.

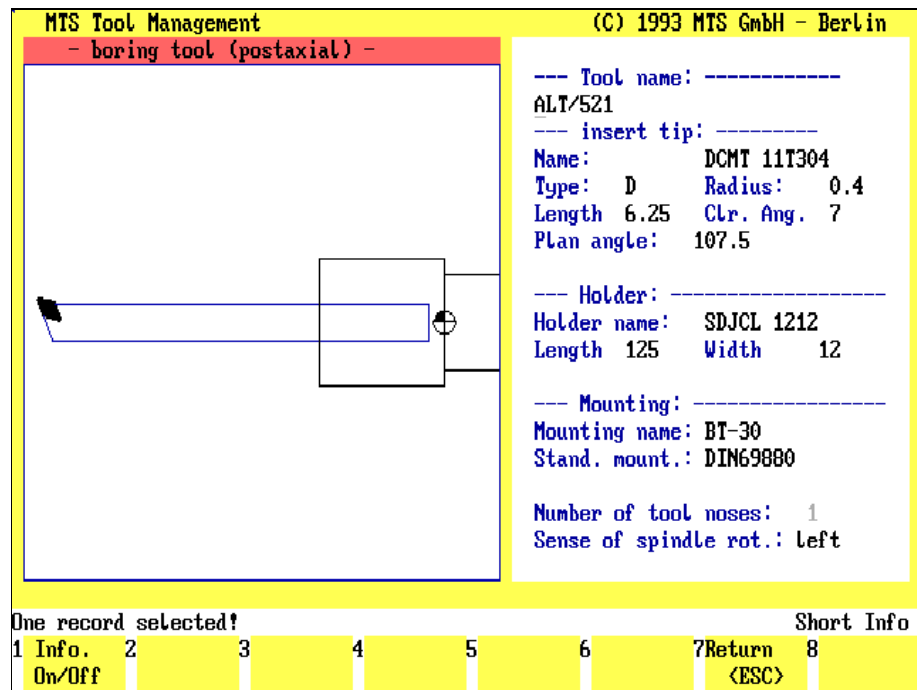
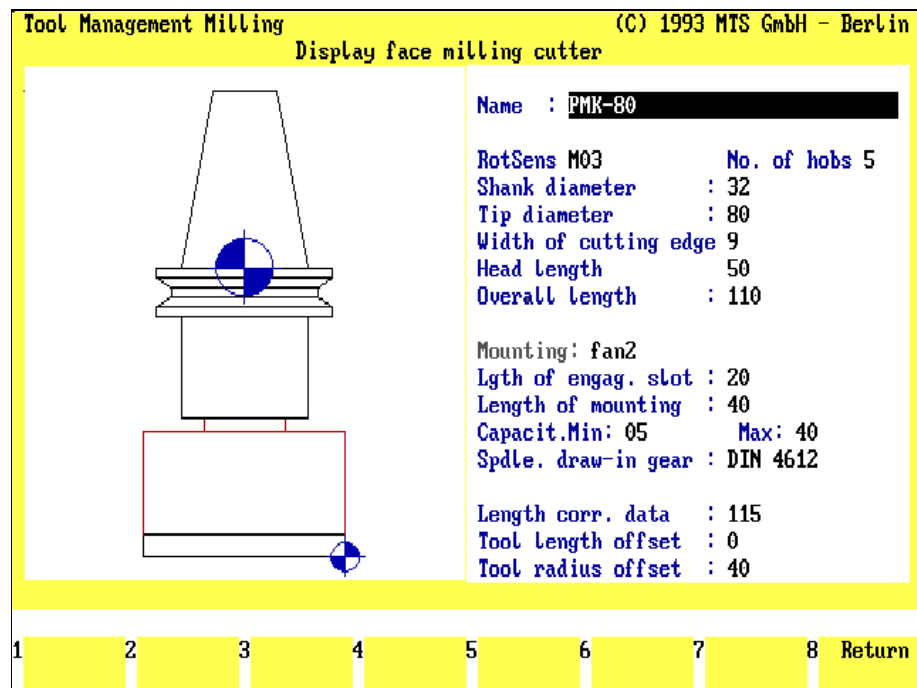
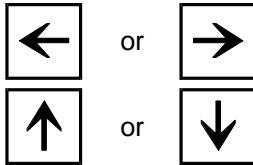


Figure 4.3.2-4:  
CNC Milling,  
information on a face  
end mill.



## Information

If you want to call up detailed information on the tool geometry, the reversible tip, the tool holder and the tool adapter of a specific tool in the turret/magazine, first mark the corresponding position/pocket and then select the "Information" function:



You mark a tool station by setting the movable frame into the corresponding field with the cursor keys.

At the same time, the inserted tool is displayed in detail.

**F3**

**Information:** You call up detailed information on this tool with **F3**:

Tool Information

The input menu of the tool manager is then loaded showing the name of the tool, the geometrical data etc. on the right-hand side. The tool, tool holder and adapter are illustrated graphically on the left side.

**F1**

**Info on/off:** You get information on each input field in the information line with **F1**.

**F7** or **F8**

**Return:** **F7** [CNC Turning] or **F8** [CNC Milling] returns you to the processing of the turret/magazine configuration.

## Valid Compensation Values

The system control initially refers to the **tool reference point** for all tool coordinate data. This point is a fixed reference point on the tool carrier.

To be able to calculate the correct target point for the tool, the distance between the real tool tip and the tool reference point has to be taken into account. The differences of these values are called **tool compensation values**. They are stored in a **compensation value register** and assigned to each tool.

For a good control of compensation values register, the numbers of the tool positions should agree with those of the assigned compensation value register.

The compensation value registers do not need to be defined manually; modern CNC control systems allow you to use default tool data. Similar to this, the turret/magazine processing offers the option of automatic transfer of the compensation values of the inserted tools into the corresponding registers:

**F6**

**Valid compensation values:** With **F6**, the compensation values of **all** inserted tools are transferred into the registers with the number corresponding to the station.

At the same time, you are returned to the tool and zeropoint definition menu.

Figure 4.3.2-5:  
CNC Turning,  
Display output of the  
tool machining times.

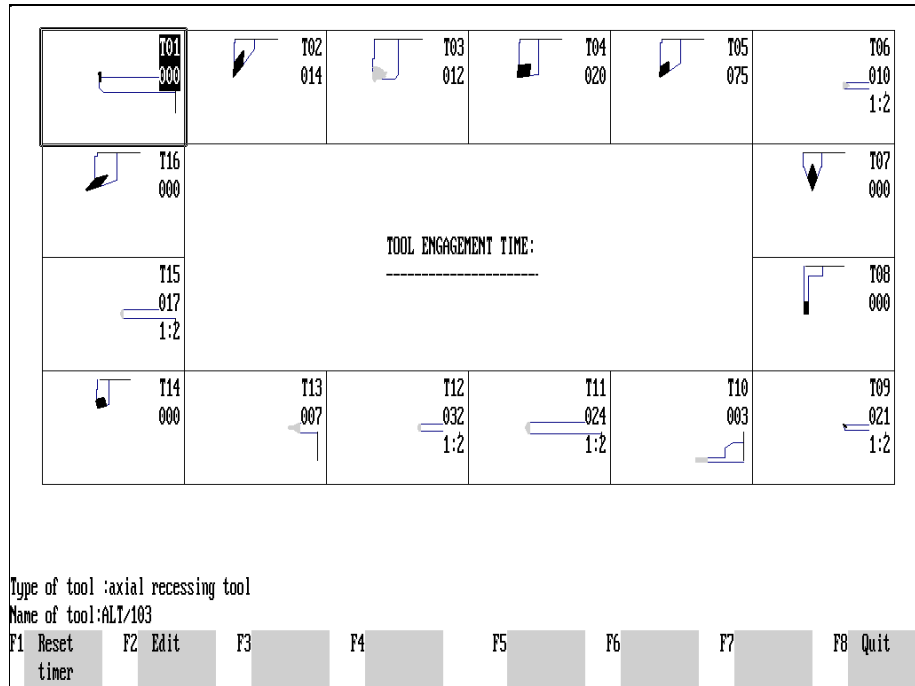
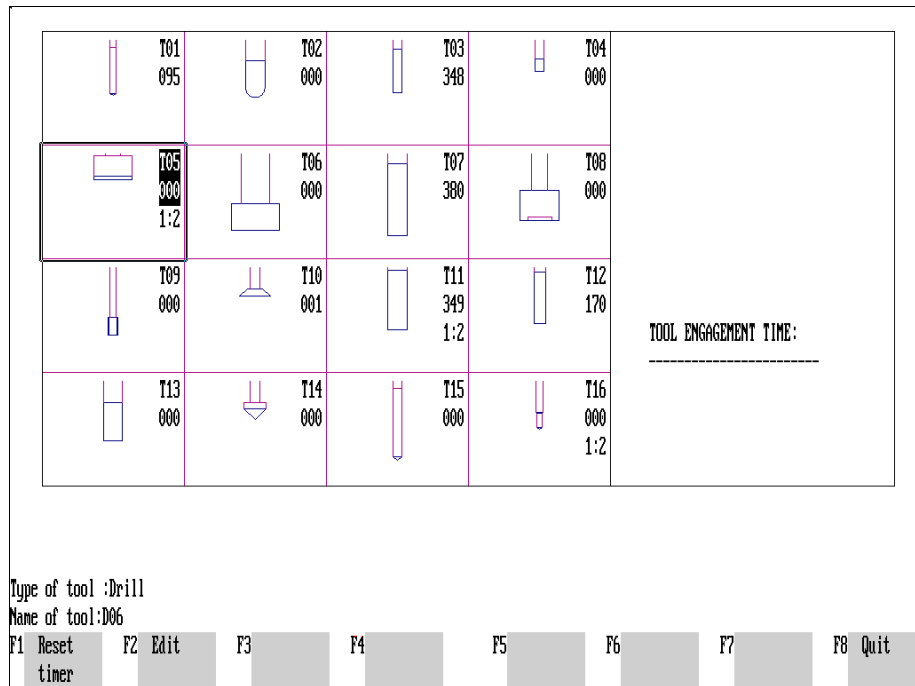


Figure 4.3.2-6:  
CNC Milling,  
Display output of the  
tool machining times.



## Tool Engagement Times

To support work preparation, it is possible to approximate the tool operation times. An overview of the operation times of each tool (feed movement in seconds calculated from the time of switching on) inserted in the turret or magazine is given in the submenu for the turret/magazine configuration. These times can be taken for costing calculations. Further additional operation time to be calculated can be determined with reference to the theoretical service times required. The times are set back to zero on tool change or as a result of a manual entry.

**F7**

**Operation times:** **F7** displays the operation times.

The determined operation times of the tools (in seconds) are then shown in the turret/magazine configuration display and you can choose between the following functions:

**F1**

**Zeroing:** Press **F1** to set all given times to zero.

**F2**

**Editing:** Press **F2** to call up the editing function.

**F8**

**Return:** Press **F8** to return to the turret/magazine configuration processing menu.

Editing

If you want to change the given operation times manually, please proceed as follows:

**←** or **→**

**↑** or **↓**

Mark the desired tool location with the cursor keys by setting the moving frame into the corresponding field.

**F2**

Then select the editing function with **F2**. The engagement time data are then released for manual change.

[Time Data]

Now enter the desired time in seconds.

**↵**

You accept the changed times with **↵**. You can then change any further time statement or return with **F8** to process the Turret/Magazine Configuration.

Figure 4.3.3-1:  
CNC Turning,  
Schematic of the  
menu sequence for  
Turret Tooling.

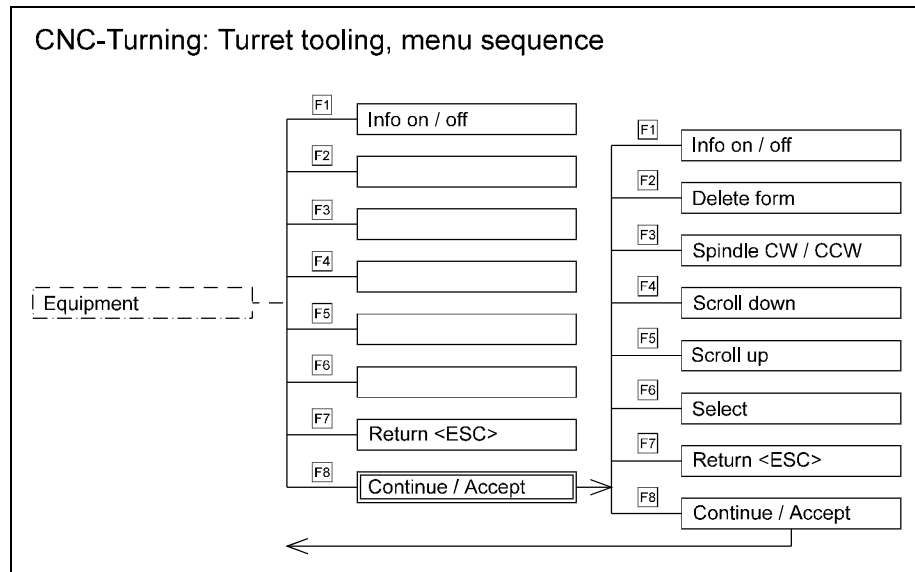
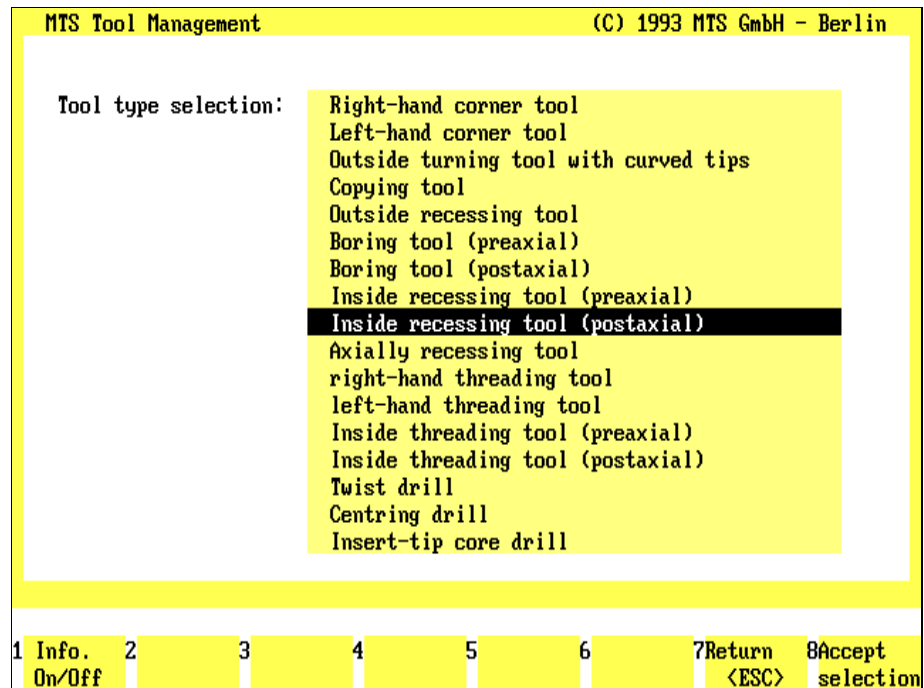
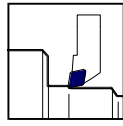


Figure 4.3.3-2:  
CNC Turning,  
Setup Mode;  
Turret Tooling,  
Selection of the tool  
type.





### 4.3.3 CNC Turning: Turret Tooling



A maximum of 16 tools can be inserted in each turret head depending on the configuration. The tool manager with the available turning tools including the tool adapter and the tool holder stored in it is used for tooling.

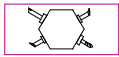
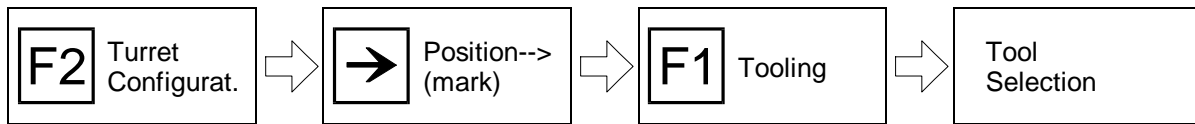


For the sake of clarity, handling the Tool Manager is described at this place only as far as necessary for tooling the turret.

Tooling a turret position consists of three work steps:

- first mark the location in which the tool should be inserted,
- then specify the desired tool type and
- choose the required tool from the available tools of this type:

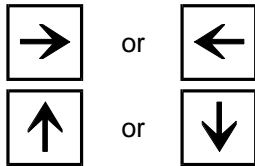
#### Selection:



The turret configuration is called with **F2** proceeding from the tool and zeropoint definition menu.

#### Mark Position

The turret assignment menu shows the turret head positions with the inserted tools. In addition, the active position is highlighted with a second frame.



You mark a tool station by setting the moving frame into the corresponding field with the cursor keys. Parallel to this, the data of the inserted tool are displayed.

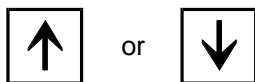


**Tooling:** To insert an other tool in the marked location, call up the Tool Selection with **F1**.

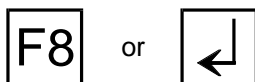
The management menu of the tool types is then loaded for selection of the desired tool (cf. Figure):

#### Select Tool Type

The tool types are listed in a window in this menu. Analogously to the marking of a tool location, the field activated for further processing is highlighted in color:



Mark a tool type with the cursor keys **↑** or **↓**.

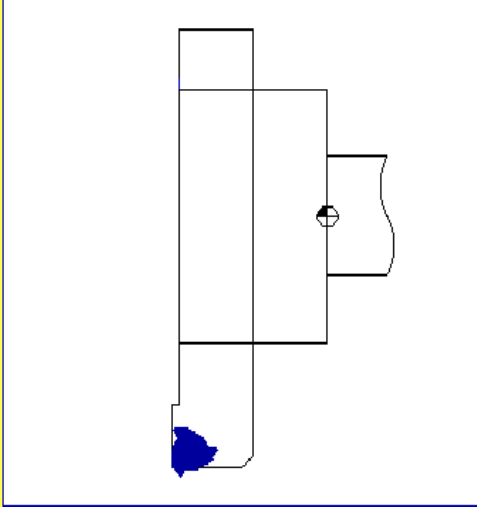


**Confirm selection:** You confirm your selection with **F8** or **←**. The Tool Selection menu is then loaded:

Figure 4.3.3-3:  
CNC Turning,  
Tool Manager, Display  
of a tool, here a left  
cutting external  
threading tool.

MTS Tool Management
(C) 1993 MTS GmbH - Berlin

- Left-hand external threading tool



--- Tool name: -----  
ALT/050-1

--- insert tip: -----  
Name: ISO/60/3.5  
Size 12. P. Angle: 60  
Lead 3.5 Radius: .505

--- Holder: -----  
Holder name:  
Length 150 Width: 25

--- Mounting: -----  
Mounting name: BT-40  
Stand. mount.: DIN69880

Sense of spindle rot.: right

Name of the mounted tool (holder and mounting)

One record selected!

1	Info.	2	3	4	5	6	7Return	8	Short Info
	On/Off							<ESC>	

### Tool Selection

For tool selection, the data of each tool is displayed on the right side of the screen while the tool, tool holder and tool adapter are illustrated graphically on the left side.

The further processing options are given in the bottom screen margin:

**F1**

**Info on/off:** You obtain information on each input field in the information line with **F1**.

**F2**

**Delete form:** To delete all data in the input fields, press **F2**.

**F3**

**Spindle cl.wise/cnt.cl.wise:** Specify the spindle rotation direction clockwise or counterclockwise with **F3**.

**F4**

or

**F5**

**Browsing forwards/back:** You "browse" in the tool file with **F4** or **F5**, i.e. all existing tools (of the selected tool type) are displayed consecutively.

**F6**

**Select:** With this function you select specific tools from an extensive tool list in order to have a better overview (as a reduced list). The data entered in the data fields serve as selection criteria. If all the fields are left empty, the entire tool list is read in again.

**F7**

or


**ESC**

**Return **ESC**:** The selection is interrupted with **F7** or **ESC** and you return to the tool type management menu.

**F8**

or



**Further/Accept:** With **F8** or , you accept the currently displayed tool in the turret position marked at the beginning.

Simultaneously you are returned automatically to processing the turret configuration.

### Procedure

Provided the tool name or certain geometrical data of the desired tool are known, you can select this tool from the tool file using the "Select" function and then confirm it in the turret position marked at the beginning:

**TAB**

You activate the input fields one by one with **TAB**.

[Value Statement]

Enter the data you know of the desired tool in the relevant input fields.

**F6**

With **F6** you select from the tool list the tools meeting the criteria you have specified.

**F4**

or


**F5**

**Browse forwards/back:** Should several tools meet the specified selection criteria, select the desired tool with **F4** or **F5**.

**F8**

or



**Proceed/Confirm:** With **F8** or , you confirm the currently displayed tool in the turret position marked at the beginning.



If you do not know any concrete data on the desired tool, first "search/select" the tool file without data. The data of the first tool (of this tool type) are then displayed and you can have the desired tool displayed and selected with the aid of the "Browse" function.

Figure 4.3.4-1:  
CNC Milling,  
Schematic of the  
menu sequence for  
magazine tooling.

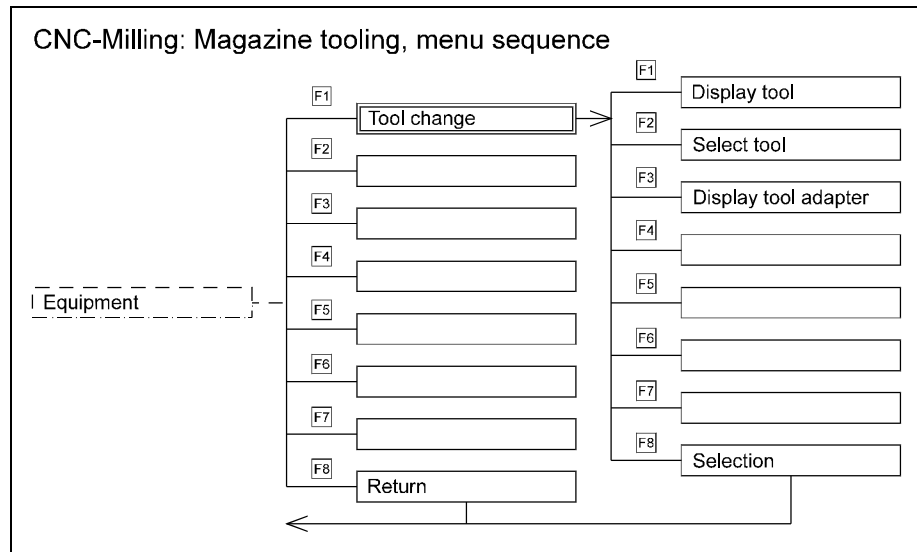
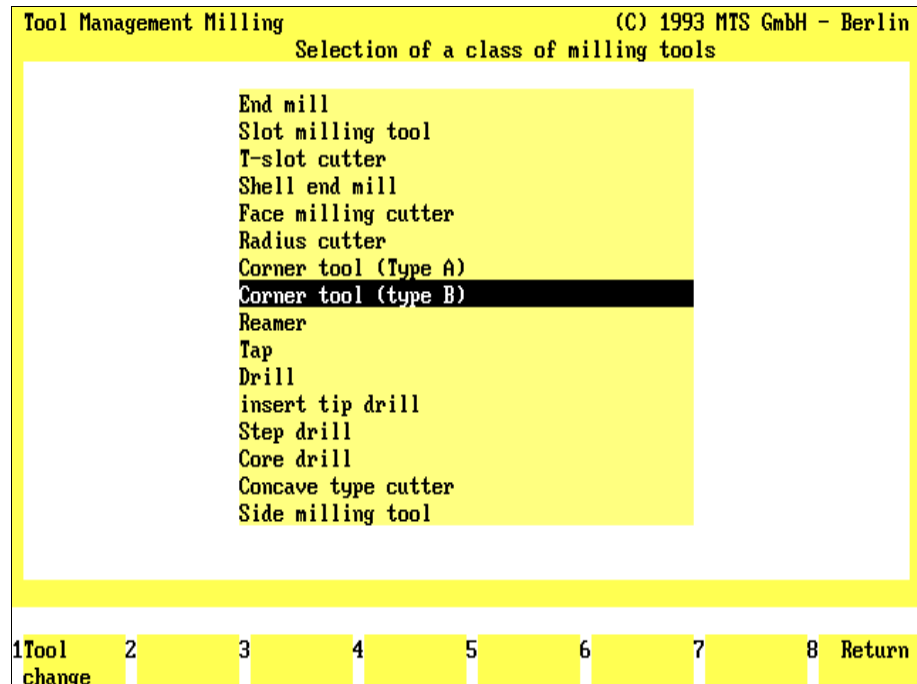
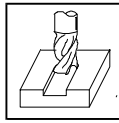


Figure 4.3.4-2:  
CNC Milling,  
Setup Mode;  
Magazine tooling,  
Selection of the tool  
type.



### 4.3.4 CNC Milling: Magazine Tooling



A maximum of 99 tools can be inserted in the magazine, depending on the configuration. The Tool Manager with the available milling tools including the tool adapter and the tool holder stored in it is used for tooling.

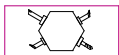
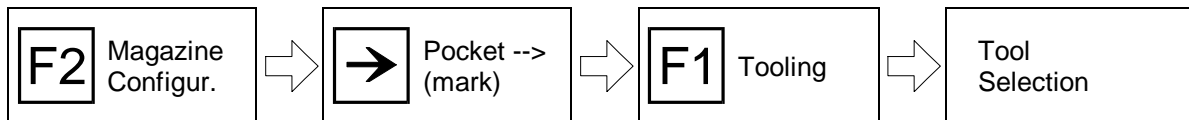


For the sake of clarity, handling of the Tool Manager is described here only as far as it is needed to tool the magazine.

Tooling a magazine position consists of three work steps:

- first mark the pocket in which the tool should be inserted,
- then enter the desired tool type and
- select the required tool from the list of existing tools of this tool type:

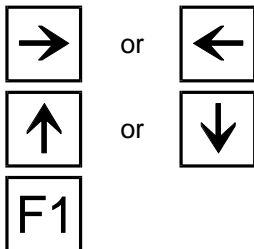
#### Selection:



The magazine configuration is called up with **F2** proceeding from the tool and zeropoint definition menu.

#### Mark Pocket

The magazine configuration menu shows the pockets with the inserted tools. In addition, the position activated for further processing is highlighted with a second frame.



You mark a tool pocket by setting the movable frame into the corresponding field with one of the cursor keys.

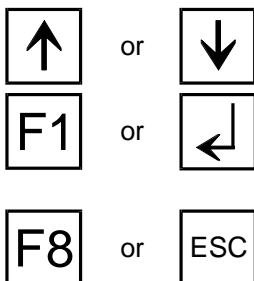
Parallel to this, the data of the inserted tool is displayed.

**Tooling:** To insert another tool in the marked position, select Tool Selection with **F1**.

Then the Management menu of the tool types is loaded for selection of the desired tool (cf. Figure):

#### Select Tool Type

The tool types are listed in a window in the selection menu. Analogous to position marking, the field that is activated for further processing is emphasized in color:



Mark a certain tool type with the cursor keys **↑** or **↓**.

**Change tool:** Then confirm your selection with **F1** or **↵**. The Tool Selection menu is then loaded.

**Return:** With **F8** or **ESC** you cancel the tool type selection.

Figure 4.3.4-3:  
CNC Milling,  
Tool Manager, display  
of a tool list, here:  
angular cutters (Type  
A).

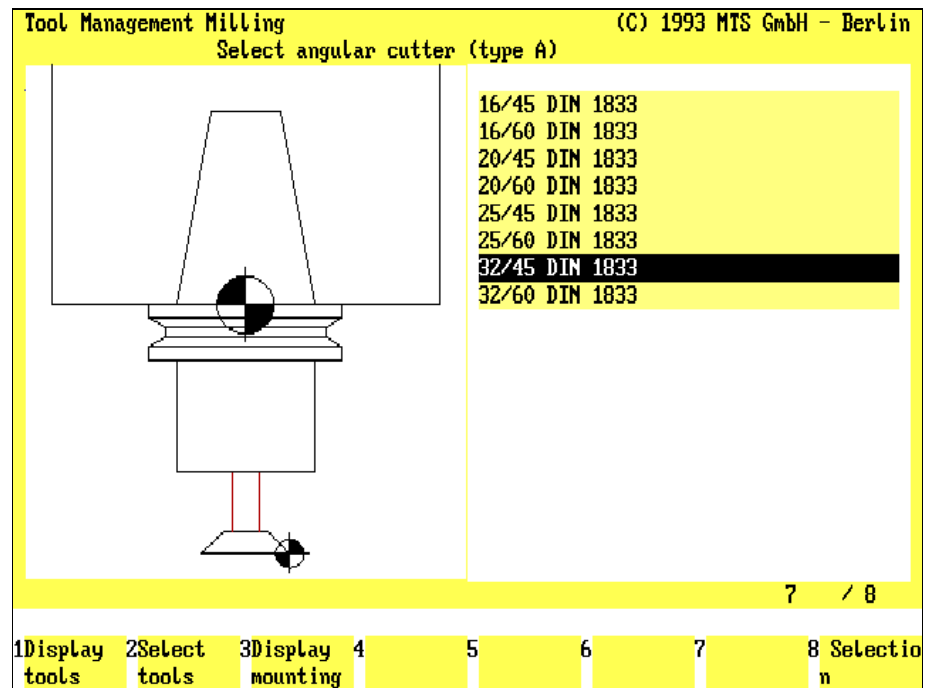
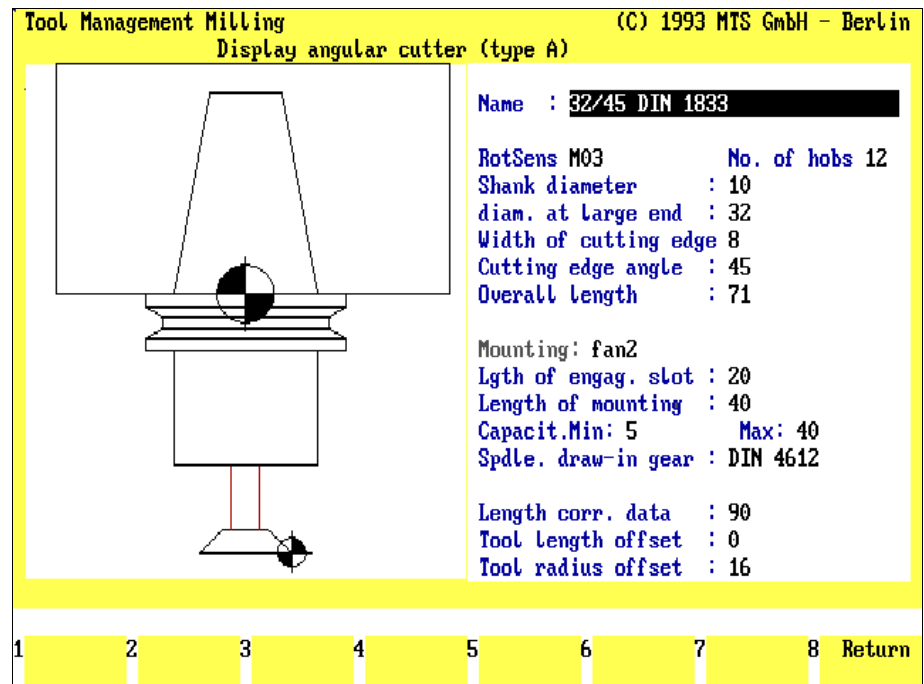


Figure 4.3.4-4:  
CNC Milling,  
Tool Manager,  
information on an  
individual tool, here:  
angular cutter (Type  
A).



Tool Selection
----------------

For tool selection there are two menus of analogous structure:

- 1.) either all available tools are listed on the right half of the screen and the currently selected tool is highlighted graphically or
- 2.) you display one by one the specific tool data of each tool and select then the tool required.

In both menus the left-hand-side screen window shows a graphical illustration of the current tool (including tool holder and adapter).

The further processing options are stated in the lower screen margin:

F1
----

**Show/list tools:** With 

F1
----

 you switch between the tool list and the specific information of a tool.

F2
----

**Select tools:** With this function you select tools from an extensive tool list (cf. CNC Turning above).

F3
----

**Show tool adapter:** Apart from displaying the tool as such, the data of the corresponding tool adapter can be displayed separately as well.

F8
----

**Select:** With 

F8
----

 you confirm the currently displayed tool in the magazine position marked at the beginning.

ESC
-----

With 

ESC
-----

 Tool Selection is canceled and you return to the tool type selection menu.

### Procedure

For the selection of the tool, the following operation steps are to be taken:

↑
---

or

↓
---

Mark the required tool in the tool list with the cursor keys 

↑
---

 or 

↓
---

 or 

PageUp
--------

 or 

PageDown
----------

.

F5
----

or

F6
----

**Next entry/ previous entry:** During the display of the detailed information, display the next or previous tool with 

F5
----

 or 

F6
----

 or 

△/shift
---------

 and 

↓
---

 or 

△/shift
---------

 and 

↑
---

.

F8
----

**Select:** With 

F8
----

, the tool is inserted in the previously defined magazine position. At the same time you switch automatically back to the processing of the magazine configuration.

### Search List

With this function you select from the extensive tool list of all tools those meeting specified geometrical criteria. Thus you reduce the current tool list and have a better overview of the tool selection.

F2
----

**Select tools:** In the selection menus this function is called with 

F2
----

:

The Selection menu and the Information menu are of analogous structure - with the difference that there are no initial entries in the data fields. The possible entries are discussed above (cf. CNC Turning, Clamping Fixture Manager).

F3
----

**Tool adapter selection:** You can select a specific tool adapter with 

F3
----

.

F5
----

**Delete form:** All entries in the input fields are deleted again with 

F5
----

.

F6
----

**Switch rotation direction:** With 

F6
----

 specify the rotation direction either as "M03" or "M04".

F8
----

**Select tools:** Press 

F8
----

 to confirm your data in order to select the corresponding tools from the list.

Figure 4.3.4-5:  
CNC Milling,  
Tool Manager,  
display output of the  
selection criteria.

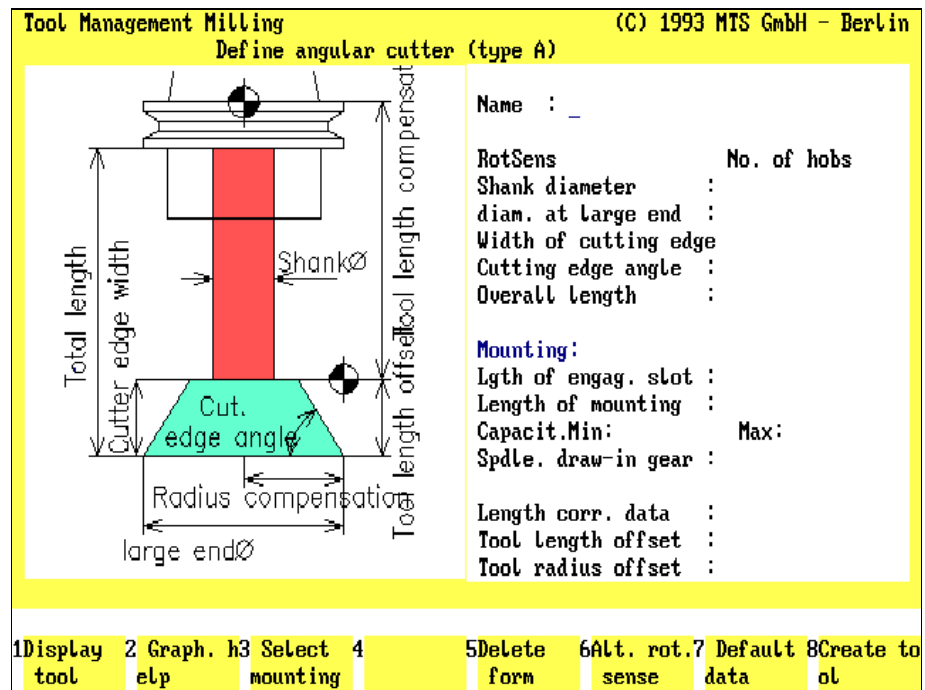
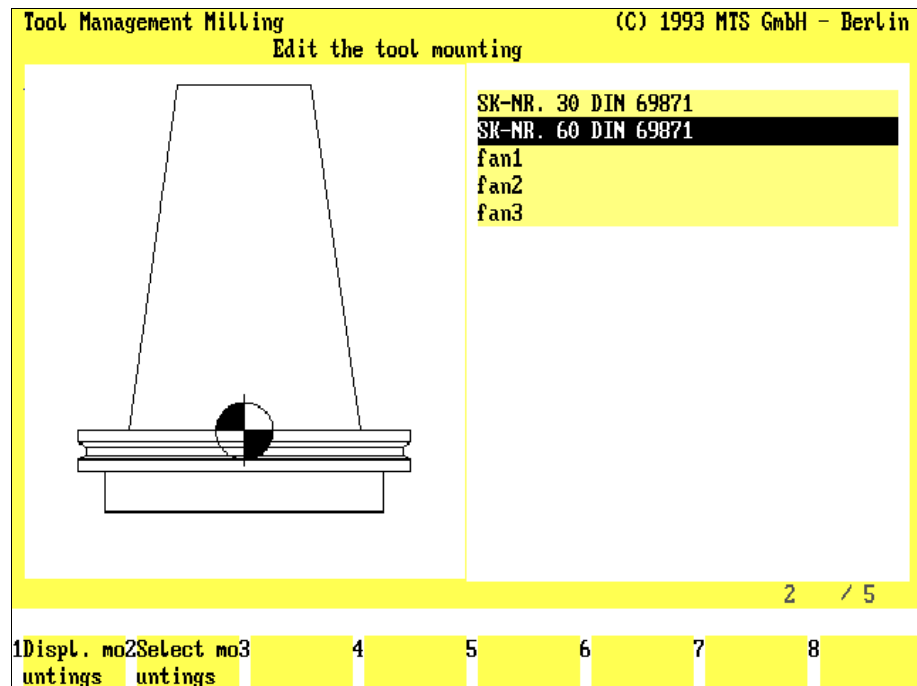


Figure 4.3.4-6:  
CNC Milling,  
Tool Manager,  
display of a tool  
adapter.





**Procedure**

For "Search and Selection" in the tool file, the input menu is first loaded without any input data in it. You then enter **only the selection criteria** meeting the desired tool requirement in the corresponding input fields:

**TAB**

You move from input field to input field with the **TAB** key.

**[Value Statement or Name]**

The values are entered as numbers, the decimal point as <,> or <.> (cf. above CNC Turning, Clamping Fixture Manager).

**ESC**

With **ESC** you cancel current processing and return to the selection menu.

**F8**

**Select tools:** Confirm your entries with **F8** only after all desired selection criteria have been entered.

You then return automatically to the preceding processing situation. The list of tools now contains only those tools meeting the selection criteria.



To have the complete list of all existing tools displayed again, perform the "Select tools" function without any entries in the input fields.

**Show Tool Adapter**

Apart from displaying the data of a single tool, the data of the associated tool adapter can be displayed separately as well. And it is further possible to select the tool adapter separately when selecting the tool file. The functions for displaying and selecting the tool adapter are analogous to those described above.

**Cross Reference**

All other functions of the Tool Manager are discussed in the [next chapter](#).

Figure 4.3.5-1:  
Calculating the theoretical cutting edge tip for a turning and a milling tool.

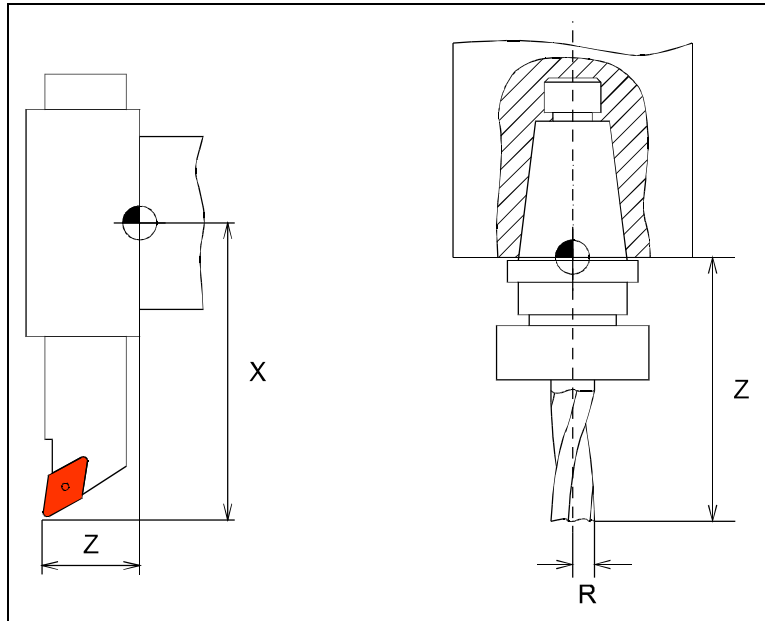


Figure 4.3.5-2:  
CNC Turning,  
Compensation values of  
a tool for corner work  
(LH) and for an external  
recessing tool (RH).

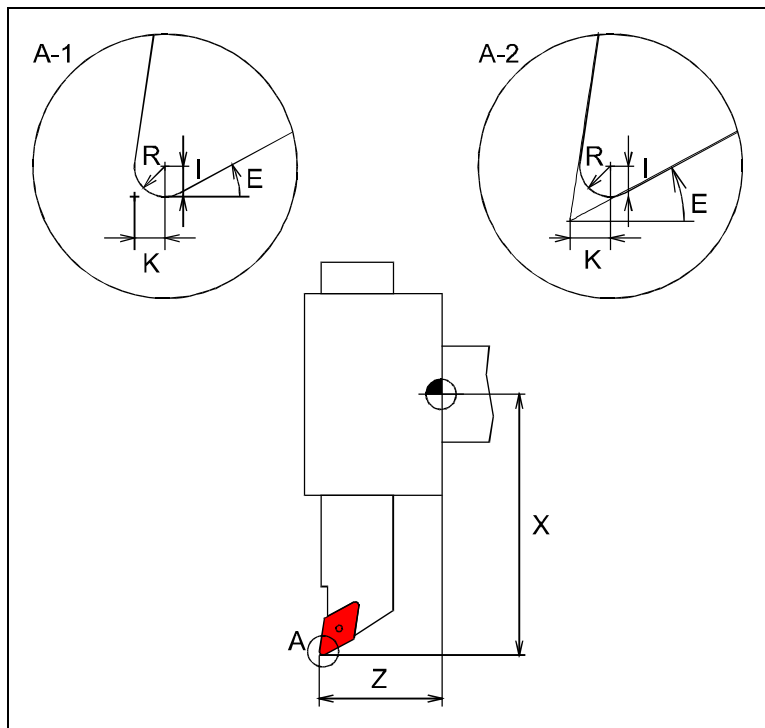
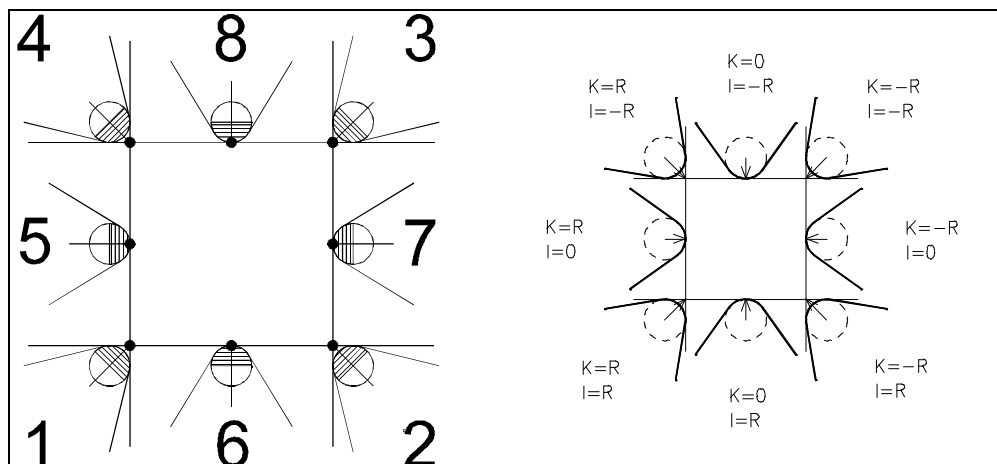


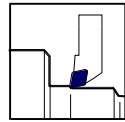
Figure 4.3.5-3:  
CNC Turning,  
back rake angle and  
infeed angle: in the left  
example the back rake  
angle is larger than the  
infeed angle, so that  
the contour can be  
produced without  
collision.



### 4.3.5 Compensation Value Register

The control system initially refers to the **tool reference point** for all tool data coordinates. This point is a fixed reference point on the tool carrier, which, as a rule, is located on the stop face of the adapter (cf. also CNC Turning and CNC Milling Programming Instructions).

#### CNC Turning



To calculate the target points for the **tool tip** (= theoretical tool nose), the distances between the tool tip and the tool reference point need to be defined for each tool (without tool nose compensation (TNC), cf. Programming Instructions).

The calculated differences of these values are stored in a **register** as X and Z compensation values and are used for calculating the tool movements (see Figure 4.3.5-1).

#### TNC

The above mentioned tool nose represents only the "theoretical tool tip", since the cutting edge/reversible tip is radiused at its "tip". The real cutting point is a few tenth of a millimeter offset of this point depending on the machining position. This difference causes a small radius in the inner corner contours when machining parallel to the axis. For rising and falling contours as well as for circles, however, considerable contour deviations may occur.

To eliminate these deviations, an equidistant to the contour is calculated as a tool path with the so-called "**tool nose compensation**", whereby the cutting edge radius **and** the position of the cutting edge tip (Cutting edge compensation vector, cf. Programming Instructions) are taken into account. Therefore, in addition to the indicated X and Z compensation values, the radius of the tool tip is stored under "R" and either the working quadrant under „Q“ or the cutting edge correction vector as additional register values under "I" and "K".

#### Working Quadrant

Finally, the angle of the TNC vector is relevant for the calculation of the cutting tip

as well. Either the „working directions“ are entered with the values of „I“ and „K“, as prefix or they can be more easily defined as „**working quadrants**“.

**If the working quadrant Q is entered the exact compensation value of the vector is calculated automatically. If Q is not entered the values of I and K are considered correspondingly.**

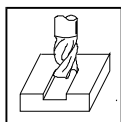
#### Cutting Edge Angle

For **external and internal turning tool**, the **rear cutting edge angle** (back rake angle) is also given under "E" to enable the eventual correction of the machining of "falling contours"(cf. Fig. 4.3.5-2/3 as well as programming instructions, G81 and G82).

#### Cutting Edge Width

And finally, for **recessing tools** the maximum **cutting edge width** (cf. Fig. 4.3.5-2) is given under "G" to enable optimized calculation of the cut segmentation of the cycles.

#### CNC Milling



The correct path movement for a **cutting tool** or a **drill** is computed under consideration of the distance between the real tool tip and the tool reference point. If the milling cutter radius compensation or the pocket cycle are switched on the radius of the milling cutter tool is used in the calculation as well.



All tools of the tool files are premeasured. Their compensation values can be confirmed automatically when tooling the turret/magazine (cf. above).

To be able to memorize the tool assignments to the turret/magazine, the numbers of the tool location should agree with that of the compensation value register.

Figure 4.3.5-4:  
CNC Turning,  
Schematic of  
the menu sequence  
for determining a  
compensation value  
register.

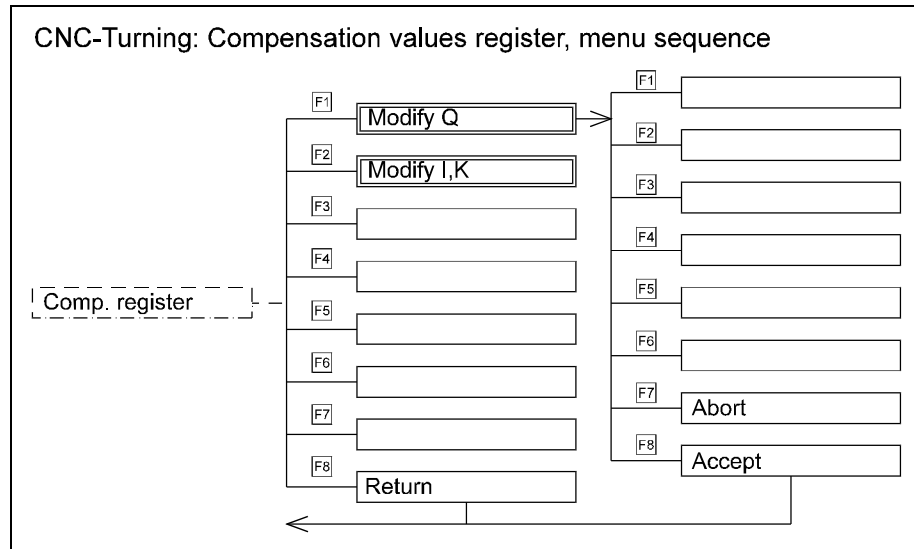


Figure 4.3.5-5:  
CNC Turning,  
Setup Mode;  
Compensation value  
register.

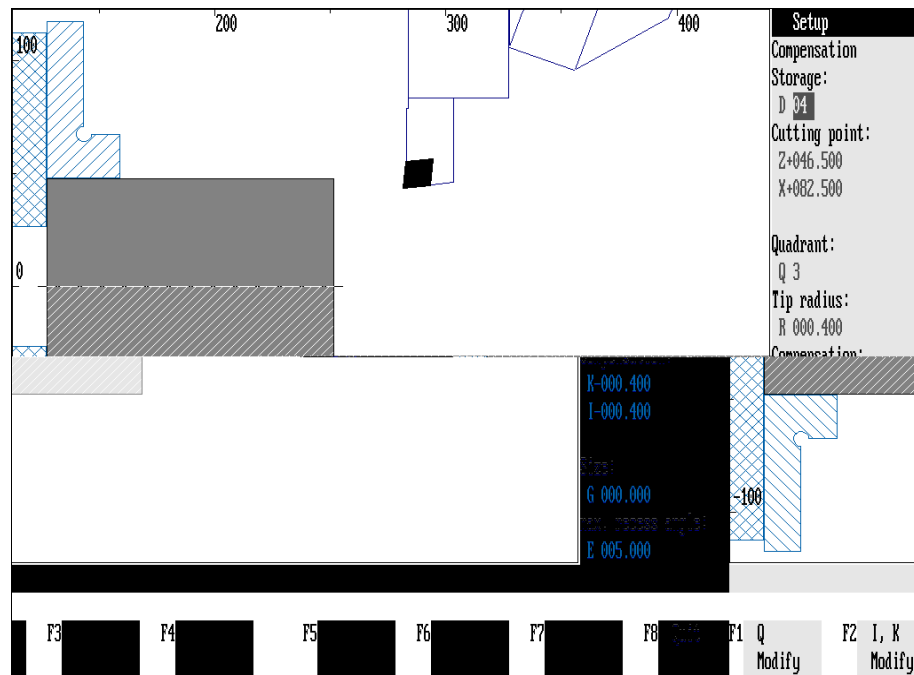
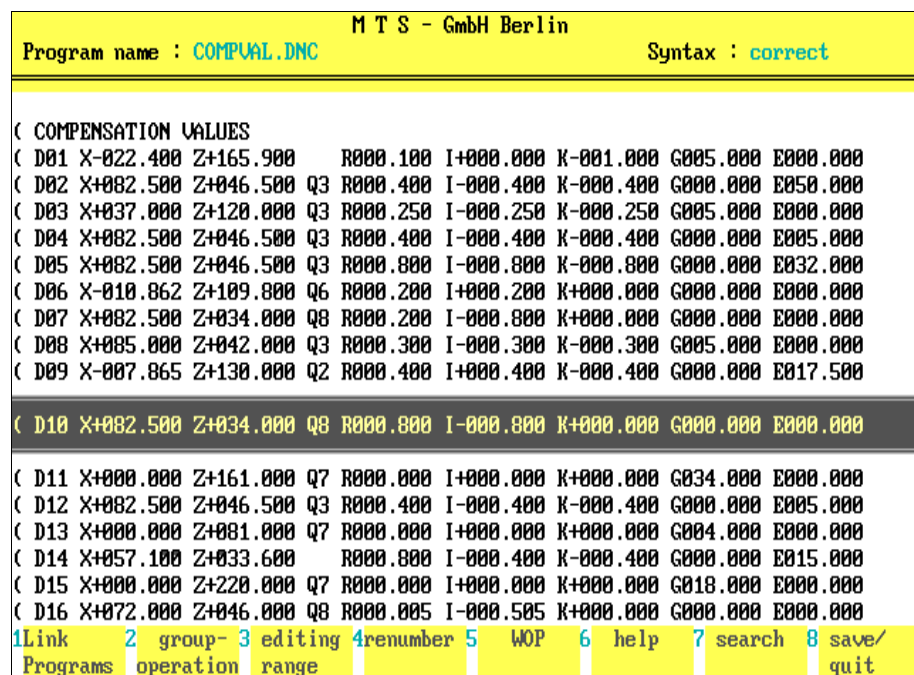
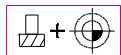
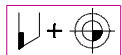
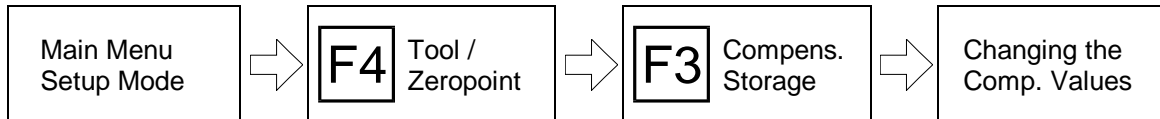


Figure 4.3.5-6:  
CNC Turning,  
Compensation value  
data in a Setup Sheet



**99 compensation value registers** in total are available. They can be defined or edited. In this way it is possible, for instance, to generate finishing allowances through slight changes of the compensation values, without making changes in the NC program (cf. below, Fig. 4.3.5-9).

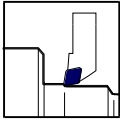
### Selection:



Processing the compensation value register is called with **F3** proceeding from the "Tool and Zeropoint Definition" menu.

When using the CNC keyboard or the graphic tablet, processing of the compensation value registers can be activated directly.

The currently loaded compensation value register is then opened and the **register numbers** and the **compensation values for the current working tool** are displayed in the information column:

 Designation	Parameter
Compensation storage	<b>D</b>
Cutting edge point	<b>X</b> and <b>Z</b> Coordinates of the theoretical cutting edge tip related to the tool reference point
Working quadrant	<b>Q</b> Position of the cutting edge tip, results from the data <b>I</b> and <b>K</b>
Cutting edge radius	<b>R</b> for boring tools: R=000.000
Size	<b>G</b> max. width of a recessing tool or drill diameter; for all other tools G=000.000
Infeed angle	<b>E</b> rear cutting edge angle of the external and internal turning tools; for all other tools E=000.000
Compensation	<b>I</b> and <b>K</b> Statement of the cutting edge correction vector <sup>1</sup> ; for boring tools =000.000
Cutting edge radius	<b>R</b> Radius of the cutting edge tip; for drills R=000.000

<sup>1</sup> The vector is defined with X and Y in the Tool Manager (cf. Chapter 5)



**To get the contour-parallel finishing allowance, increase the cutter radius in the compensation value register and start machining with the cutter radius compensation switched on.**

Figure 4.3.5-7:  
CNC Milling;  
Compensation values

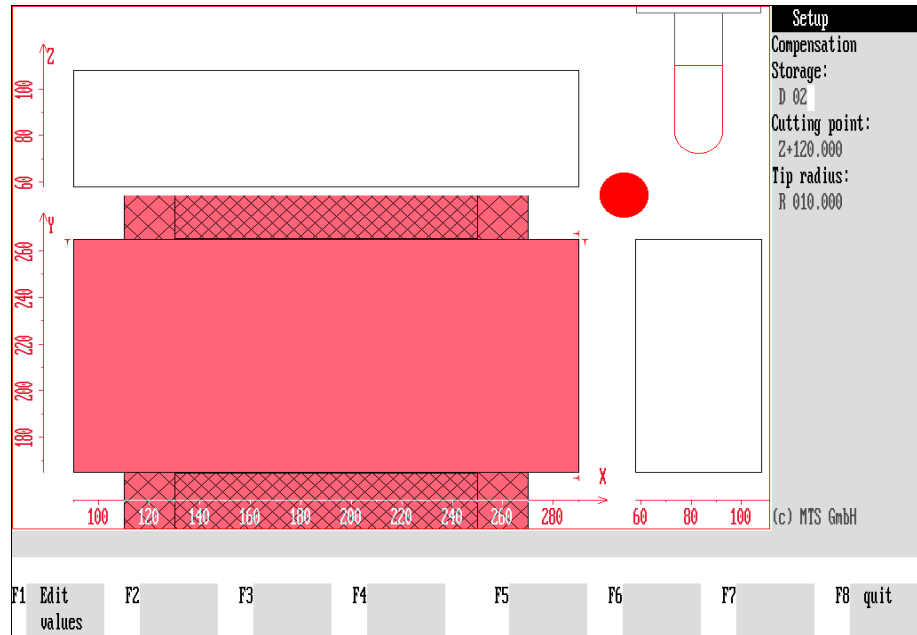
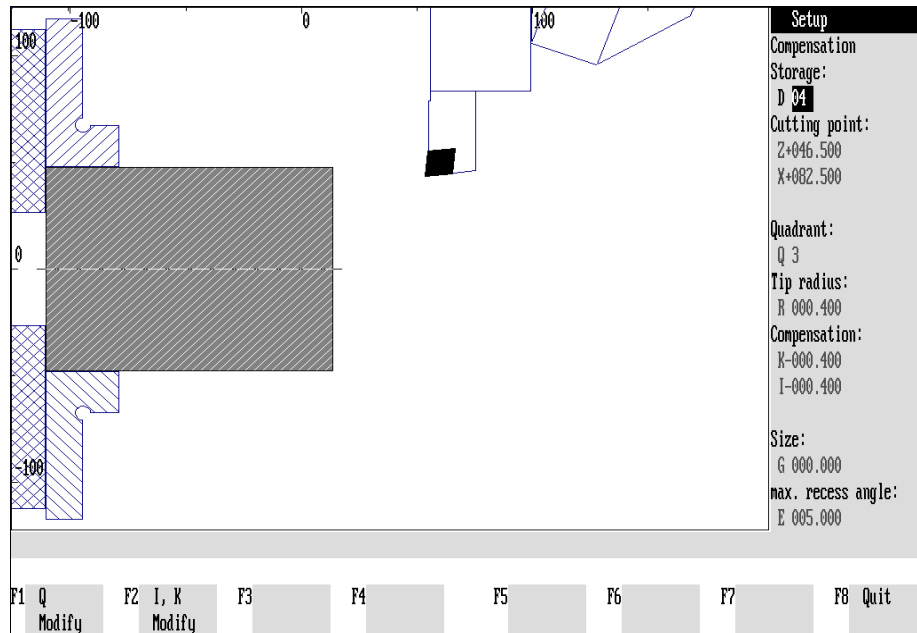
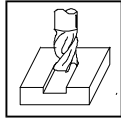


Figure 4.3.5-8:  
CNC Turning,  
Setup Mode;  
Entry of the compensation values





In contrast to turning, the compensation value registers of milling consists of two parameters only:

Designation	Parameter
Compensation storage	<b>D</b>
Cutting edge point	Coordinate of the cutting edge tip referring to the tool reference point
Cutting edge radius	<b>R</b> Radius of the milling/boring tool

### Procedure

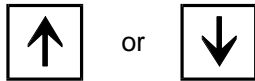
Processing the compensation value register consists of two working steps:

- first select the desired register, and then
- change the compensation values as required:

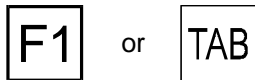
#### Select Register

After the register is called, the cursor is first located under the register number.

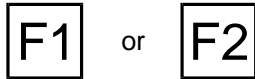
[Register Number]



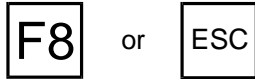
Enter the number of the register or select the register with the cursor keys or .



**Edit values:** Then switch with or or to the editing function.



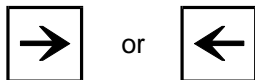
**Change Q / change I, K:** For turning you have the additional option of editing working quadrants or the values of I and K of the cutter vector.



**Return:** You switch back to the starting situation with or .

#### Edit Values

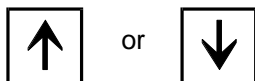
To change the compensation values, the currently activated input field in each case is highlighted in color:



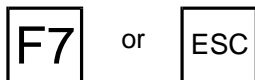
You move the cursor forwards and backwards in the input fields with the cursor keys or .

[Value Statement]

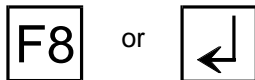
You change the compensation values by overwriting the data.



You move from input field to input field with the cursor keys or .



**Cancel:** With or you cancel processing.



**Accept:** With or you confirm the changes. Simultaneously you return to the starting situation.

### Note

If a reference tool exists, the compensation values of another tool can be first calculated with the help of the coordinate values by "touching" them and then storing the values in a register. In this way the CNC Simulator allows the compensation values to be checked and so it is possible to demonstrate tool geometry during CNC training.

### Cross Reference

Please pay attention to the necessary entries for preaxial or postaxial machining in the CNC Turning configuration (cf. Configuration Instructions).

Figure 4.4-1:  
CNC Turning;  
Schematic  
representation of the  
work space.

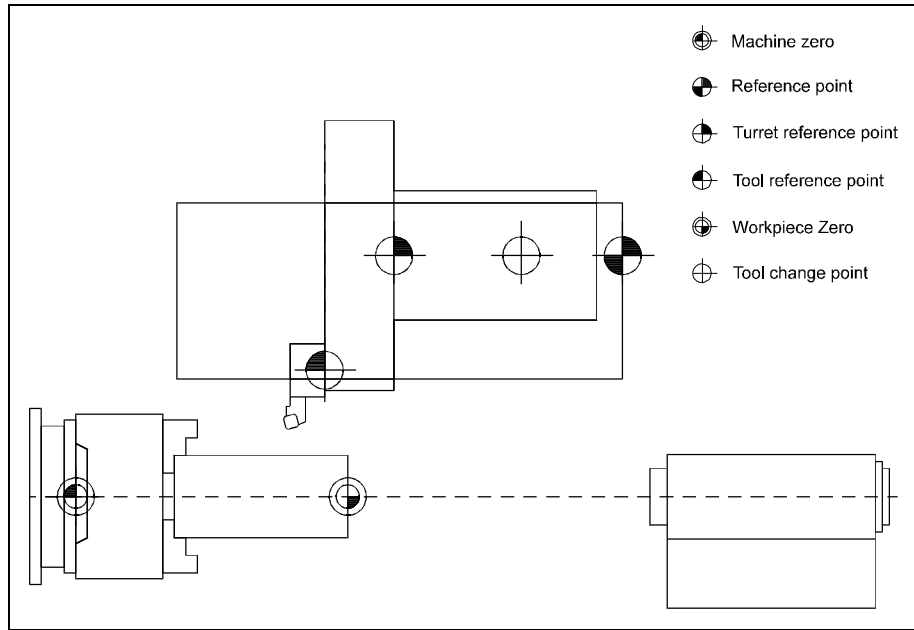
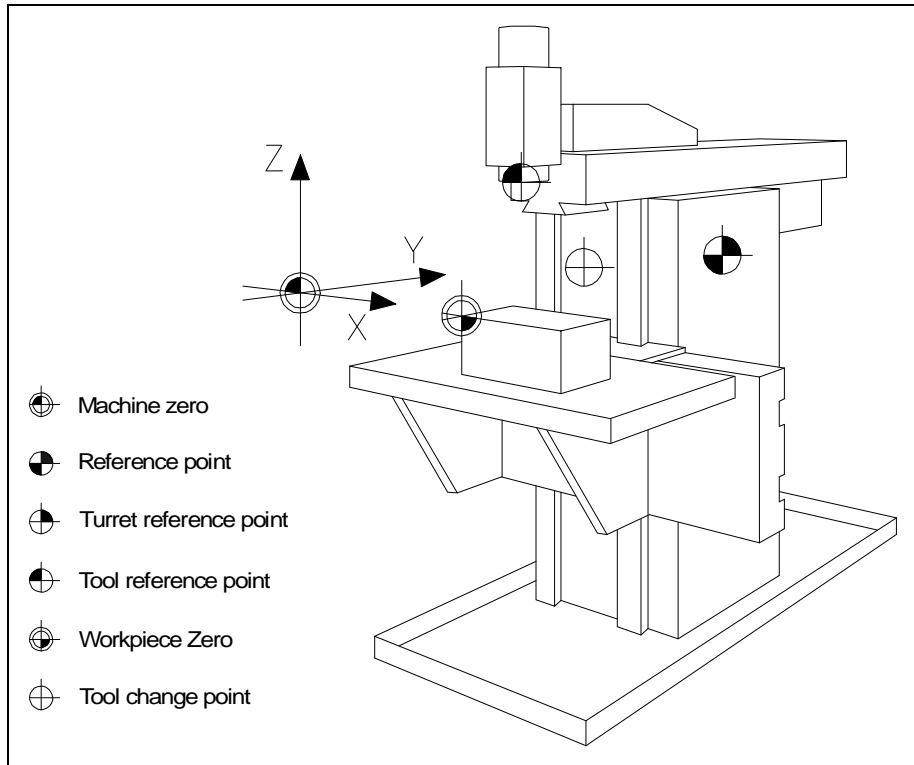


Figure 4.4-2:  
Example of the zero and  
reference points of a  
milling machine





## 4.4 Zero and Reference Points

The most important reference and zeropoints of an NC machine tool are:

- the machine zero point (also called datum),
- the machine reference zero point (also called home position),
- the workpart zero point,
- the tool change point and
- the tool reference point.

### Machine Zeropoint

The "starting point" of the coordinates system of a machine tool is determined by the machine zeropoint, which, in turn, has been defined in the construction design by the manufacturer by means of the position measuring system and control adaptation. In the same way, the machine zero point of the CNC Simulator is predetermined and cannot be changed.

### Reference Point

The reference point serves for calibration and control of incremental position measuring systems. It is located within the travel range of the turret slide/spindle head (i.e. spatially diagonally opposed to the machine zeropoint). After the NC machine or the CNC Simulator are switched on, you need to go to the reference points first, in order to adjust the position measuring system in relation to the reference point.

In absolute position measuring systems, it is not necessary to go to the reference point, since here the position of the feed units in relation to the machine zeropoint can be read in at each place of the scale.

### Workpart Zeropoint

The workpart zeropoint designates the starting point of the cordiantes system of the workpart. It is free definable, however, it is helpful to define it analogously to the dimension reference point of the tool drawing.

With the new version 5.2 of the CNC Simulator, four different zeropoints can be stored simultaneously under the G commands "G54", "G55", "G56" and "G57" or "G58" (cf. Programming Instructions).

### Tool Change Point

In turning, the tool carrier goes to the the tool change point if the turret is newly tooled.

In milling, the spindle head always moves to the tool change point in order to insert a new working tool.

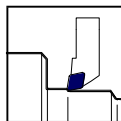
The position of the tool change point is determined in the **configuration** (cf. Configuration Instructions).

### Tool Reference Point

The tool reference point is located on the tool carrier and is identical with the tool setting point when inserting the tool holder.

Since the system control refers for all data of the tool coordinates to the tool reference point, the position of the tool tip or cutting edge in relation to the tool reference point needs to be calculated using the compensation values (cf. above).

### Turret Reference Point



The turret reference point is located on the tool carrier and functions as a reference point for the configuration of the travel range (cf. Configuration Instructions).

Figure 4.4.1-1:  
CNC Turning,  
Setup Mode;  
Going to the reference  
point.

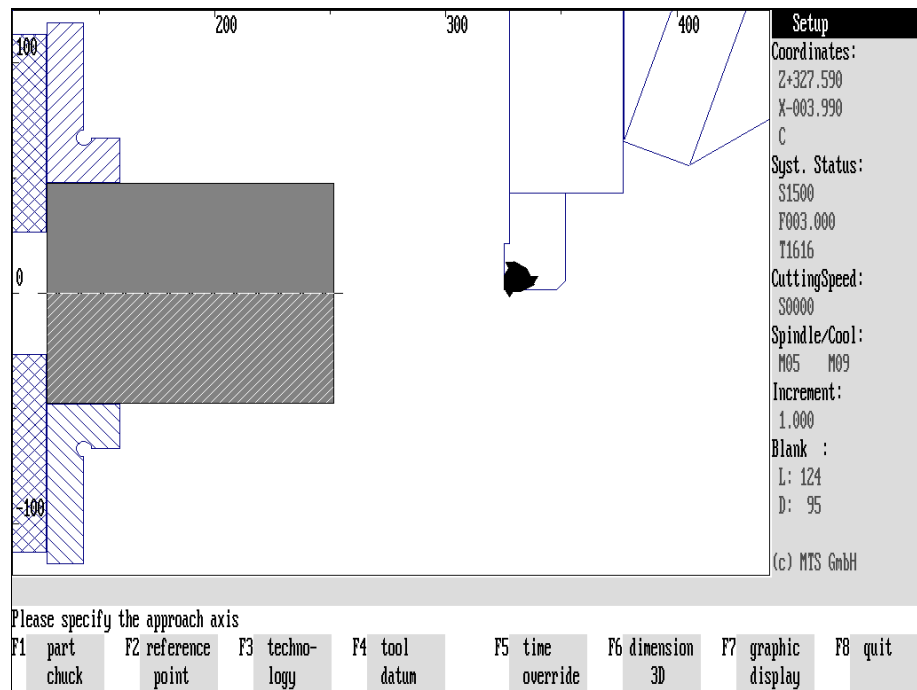
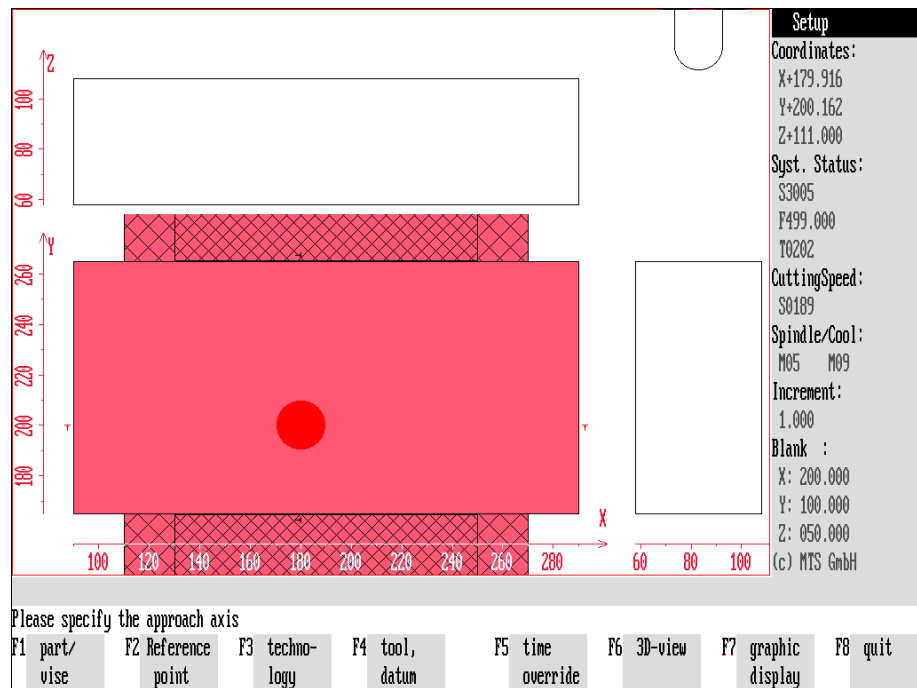


Figure 4.4.1-2:  
CNC Milling,  
Setup Mode going to  
the reference point.



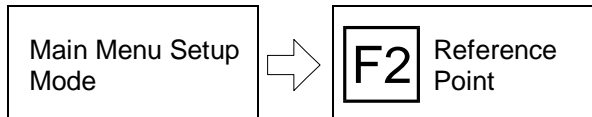
## 4.4.1 Reference Point

Similar to the NC machine, you need to go to the reference point after starting the CNC Simulator in order to adjust the position measuring systems of the axes in relation to the reference point or the machine zeropoint.

### Note

If you do not go to the reference point, NC programs cannot be executed in Automatic Mode and the workpart zeropoint cannot be determined. When going to a reference point specified in the configuration make sure to avoid a collision.

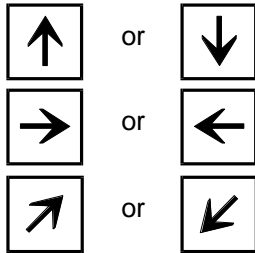
### Selection:



The function "Go to reference point" is called with **F2** proceeding from the main menu of the Setup Mode.

### Procedure

For the "Go to reference point" function no menu changes are required!  
After the call, the following message is displayed in the dialogue line:  
**"Please specify approach axis".**



The tool carrier moves to the reference point immediately after a cursor key is pressed. Please note the assignments of the approach axes:

Key	CNC Turning	CNC Milling
↑ or ↓	X axis	Z axis
→ or ←	Z axis	X axis
↗ or ↘	not applicable	Y axis

<sup>1</sup> to G17

<sup>2</sup> numerical keypad

After the reference point has been reached, you remain in the Setup Mode in the machining plane.

Figure 4.4.2-1:  
CNC Turning,  
Schematic of the  
menu sequence for  
definition of a  
zeropoint

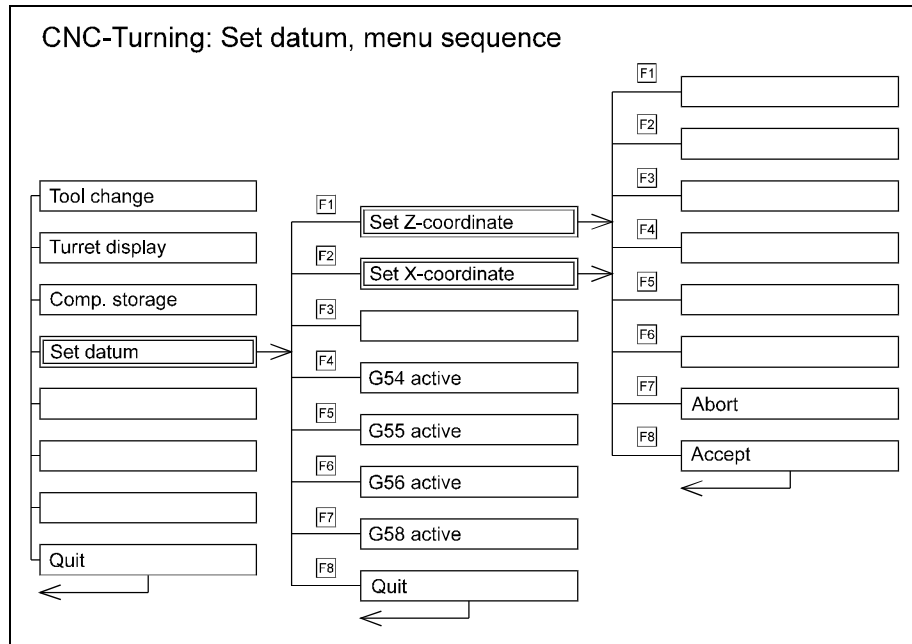


Figure 4.4.2-2:  
CNC Turning,  
Setup Mode;  
Menu for determining  
the workpart zeropoint.

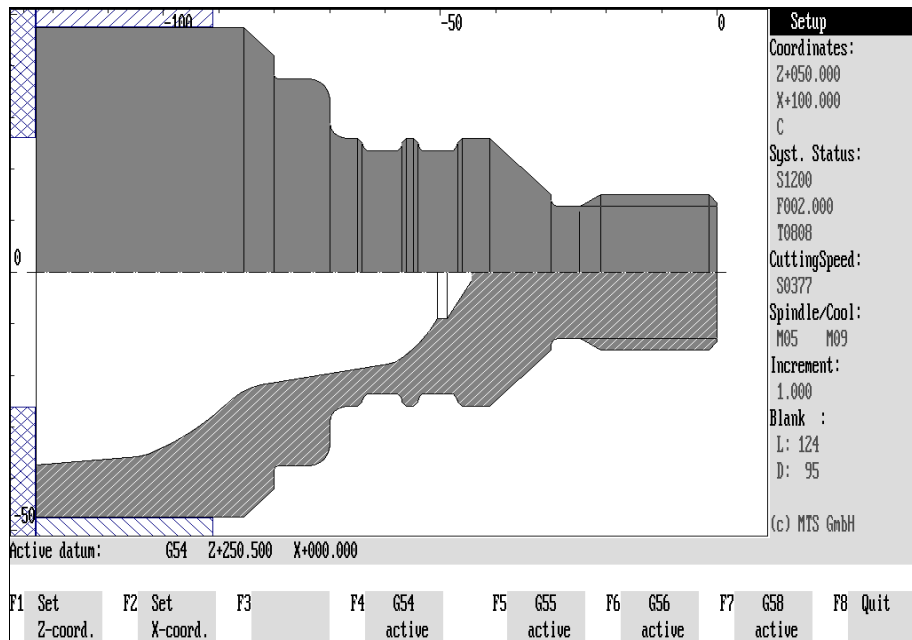
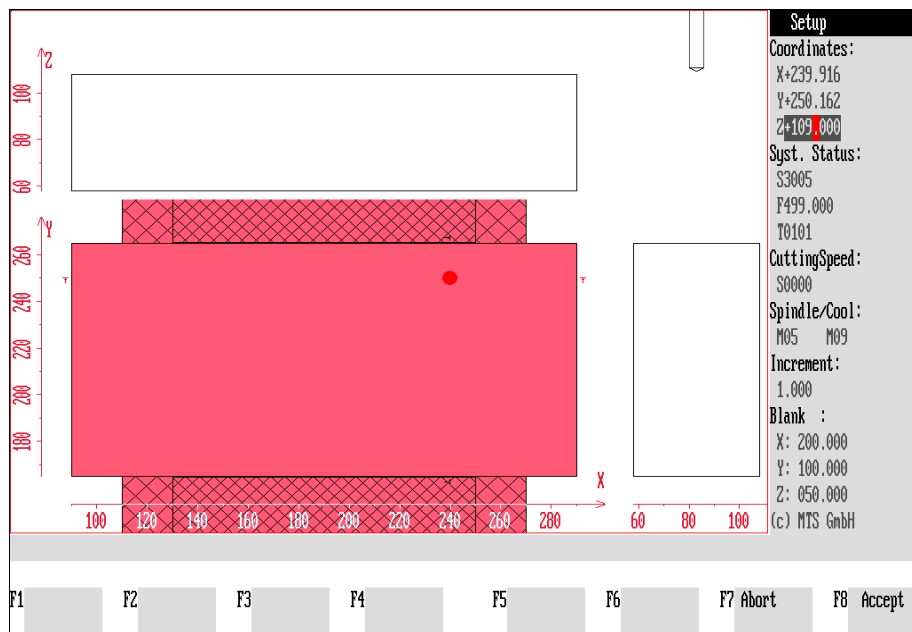


Figure 4.4.2-3:  
CNC Milling,  
Setup Mode;  
Manual entry for  
determining a new  
workpart zeropoint.

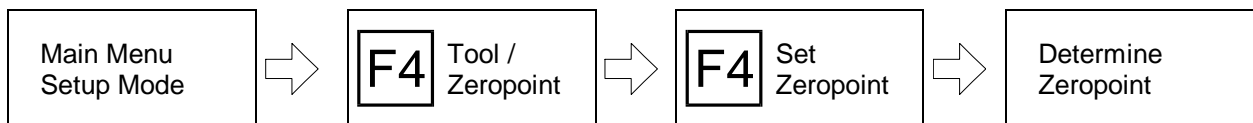


## 4.4.2 Workpart Zeropoint (Coordinate Change)

The workpart zeropoint can be reset anywhere in the work space area by "zeroing" the coordinates or by entering a new value for a statement. In manual mode, this is normally done by touching the workpart, in which case new coordinate values are assigned to the tool cutting edge.

Furthermore, the MTS programming code allows four different workpart zeropoints to be programmed and activated with the corresponding G command. Since these G commands are modally valid (after the entry they are stored until overwritten), they can be used for the manual mode as well. If you determine a new workpart zeropoint manually, first enter the G command under which this zeropoint should be saved.

### Selection:

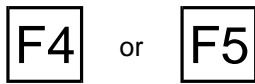


The "Set zeropoint" function is called with **F4** proceeding from the "Tool and zeropoint definition" menu.

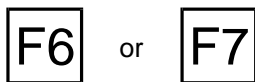
### Procedure

When processing the workpart zeropoint, you need to differentiate two work steps from each other:

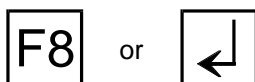
1. you activate a stored zeropoint after a G command;
2. you change this zeropoint by touching and/or entry of the coordinate values:




**G54/G55/G56/[G57/G58] active:** Activate one of the corresponding zeropoints with the function keys **F4** to **F7**. The corresponding coordinate values are



displayed simultaneously in the information line next to the message: "**Active zeropoint**".



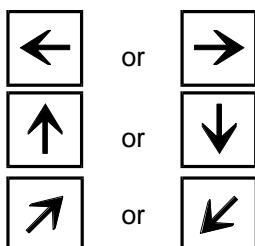
**Return:** With **F8** or  you conclude definition of the workpart zeropoint and return to the starting situation.



When in doubt, use as a guideline for the coordinates system the position of the current workpart zeropoint. In milling, you can display the axes using the "Graphic display" menu.

### Manual Change of a Zeropoint

The manual change of the workpart zeropoint is made by "zeroing" the coordinates or a certain value statement. The changes are stored under the zeropoint which was active last (cf. above).



If you want to determine the workpart zeropoint by **touching**, first move the tool tip carefully to one edge of the workpart (or reverse) until the collision message

**"Collision: Please switch on spindle"**

is displayed.

Figure 4.4.2-4:  
CNC Turning,  
Workpart zeropoint;  
Example of activating  
a stored zero with G54

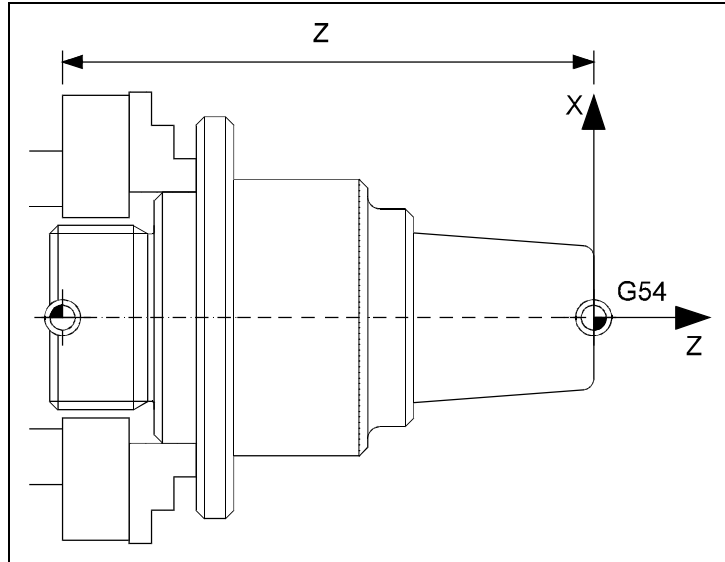


Figure 4.4.2-5:  
CNC Turning,  
Workpart zeropoint;  
Example of activating  
a stored zero with G55

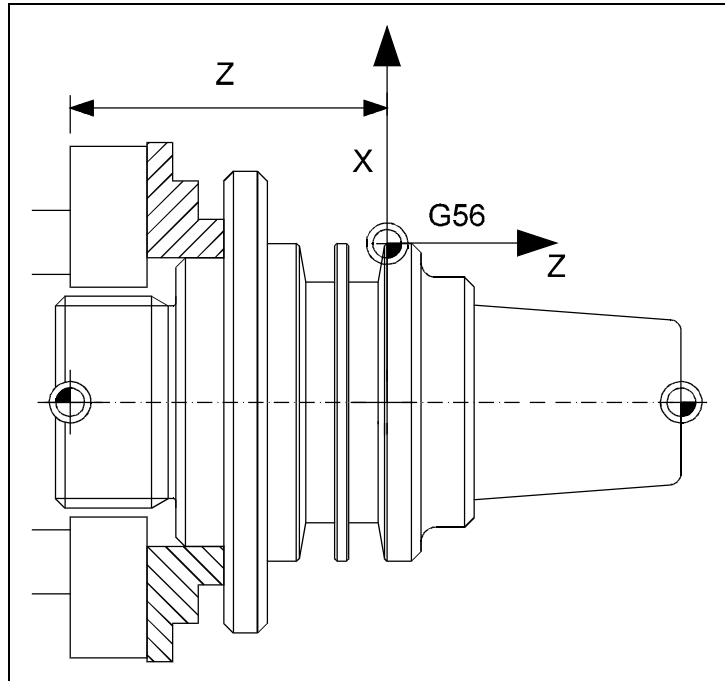
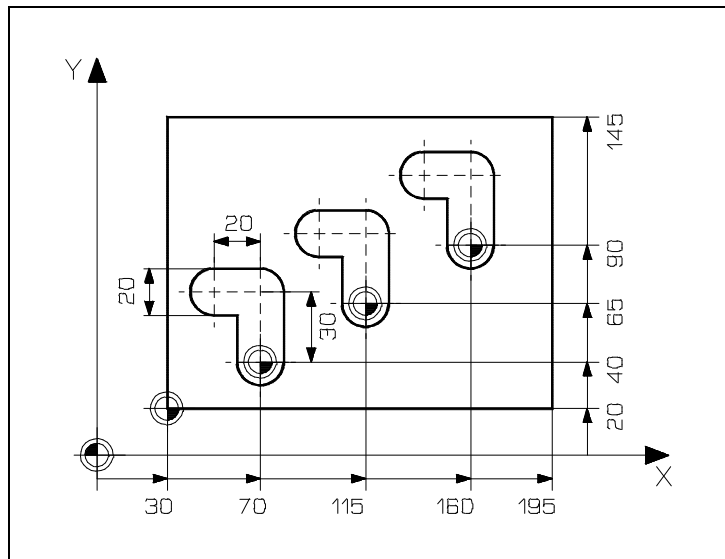


Figure 4.4.2-6:  
CNC Milling,  
Workpart zeropoint;  
Example of activating  
a stored zero with  
G54, G55, G56 and  
G57.



F1

or

F2

F3

or

**Set X/Y/Z coordinates:** Depending on the coordinates to be defined, press the relevant function key (after touching the workpart).


The corresponding coordinate field in the information column is then released for entry:

[Entry of a Coordinate Value]

Now enter the desired coordinate value (the existing values are overwritten), for zero setting therefore "000.000" etc.

**In milling, the cutter radius needs to be taken into account as well for the X and Y coordinates (according to G17).**





After entering the values, confirm with .

F8

**Accept:** To save the entry, then confirm it with .

F7

**Cancel:** With  you cancel the entry of a coordinate value.

You then return to the "Define zeropoint" menu and you can either edit a further coordinate statement or change the values in an other G command. You return to the starting situation with .

#### Note

When "touching" the workpart, you proceed simply as follows: move in rapid traverse motion (see below) up to a safety distance from the workpart, then you switch to the smallest incremental value (cf. following chapter) and travel now in feed motion with the tool tip towards the workpart edge (or reverse) until the collision message is displayed.



When using the CNC keyboard or graphic tablet, the entry options of the coordinate statement are activated immediately after pressing a "zeroing key".

Figure 4.5-1:  
CNC Turning,  
Schematic of the  
menu sequence  
"Machine and  
technology functions"

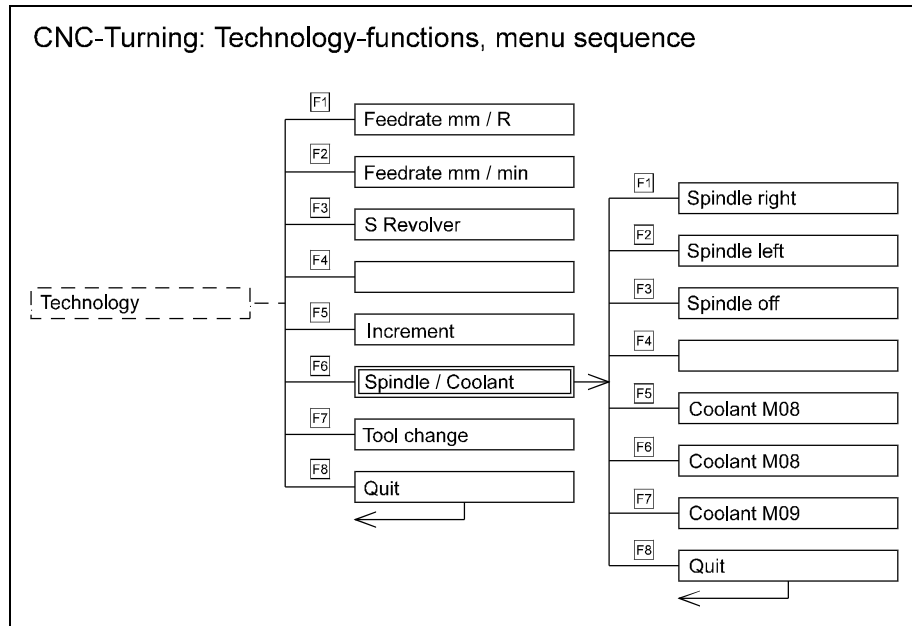


Figure 4.5-2:  
CNC Turning,  
Setup Mode;  
"Machine and  
technology functions"  
menu.

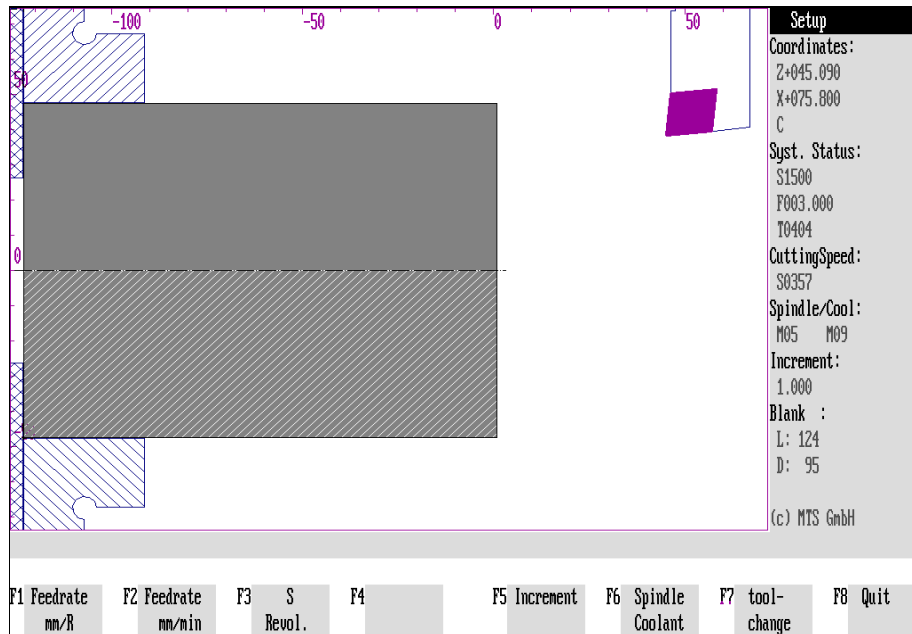
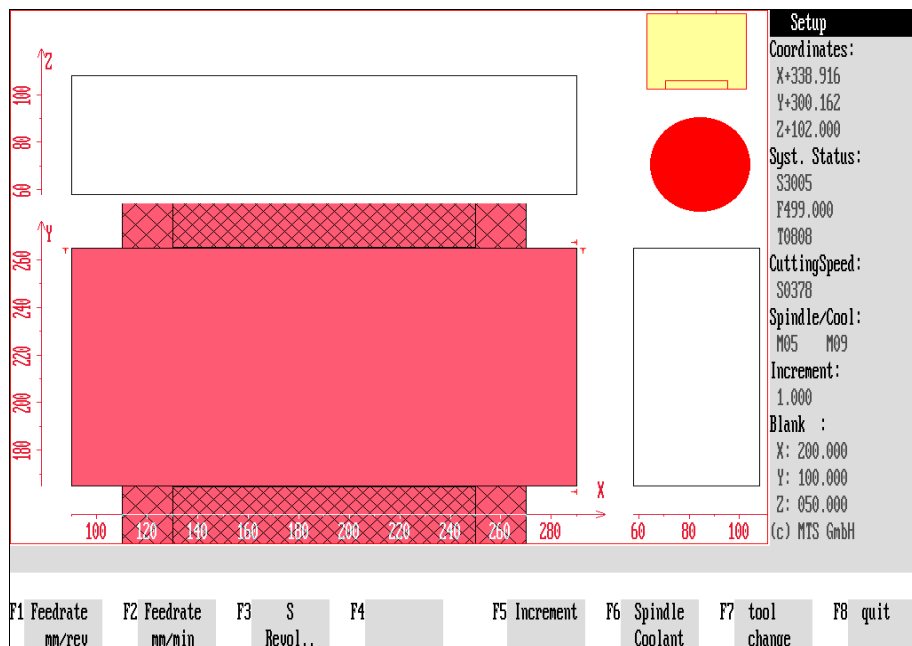


Figure 4.5-3:  
CNC Milling,  
Setup Mode;  
"Machine and  
technology functions"  
menu.





## 4.5 Machine and Technology Functions

The CNC Simulator provides the customary machine functions also in the Setup Mode. In addition to the tool change already mentioned, these are:

- Enter **feedrate**,
- Enter **speed**,
- Switch **spindle** on/off, determine sense of rotation,
- Switch **coolant** on/off.
- Determine **increment**.

### Note

Within the technical performance entries of the machine configuration of the CNC Simulator, presettings can be made for the maximum and minimum speed, the maximum and minimum feedrate and the rapid traverse speed. The currently selected configuration file is decisive for machining (cf. also Configuration Instructions).

TECHNO

The manual machine controls are set in two submenus. These are called up with **F3** in the "Technology" function proceeding from the main menu of the Setup Mode.

**F1**

**Feedrate mm/r:** The feedrate in mm/r is set with this function.

**F2**

**Feedrate mm/min.:** The feedrate in mm/min. is set with this function.

**F3**

**S speed:** The spindle speed in rpm is set with this function.

**F5**

**Increment:** The "increment value" defines the step length (in mm) of the tool carrier (or the machine table) when pressing a direction key once. You select the increment value with **F6**.

**F6**

**Spindle/coolant:** You can switch the spindle and the coolant pumps on or off according to the machine commands in an other submenu. You call up this submenu with **F6**.

**F7**

**Tool change:** To change the tool in working position (by turning the turret to the stated station or changing in a tool from the magazine), press **F7**. The tool change is described in Section 4.3.1.

**F8**

**Return:** **F8** returns you again to the main processing level of the Setup Mode.

The second submenu allows you the following additional operation possibilities:

**F1**

or

**F2**

**Spindle right/left/off:** With these functions, the spindle is switched on or off according to the M commands M03/M04/M05.

**F3**

**F5**

or

**F6**

**Coolant M08/M07/M09:** With these functions, the coolant pump is switched on or off corresponding to the M commands.

**F7**

**F8**

**Return:** **F8** returns you to the starting situation



Since the handling of the above machine functions is not difficult, the operation is given below as a summary.

Figure 4.5-4:  
CNC Turning,  
Setup Mode;  
"Spindle and coolant  
control" menu.

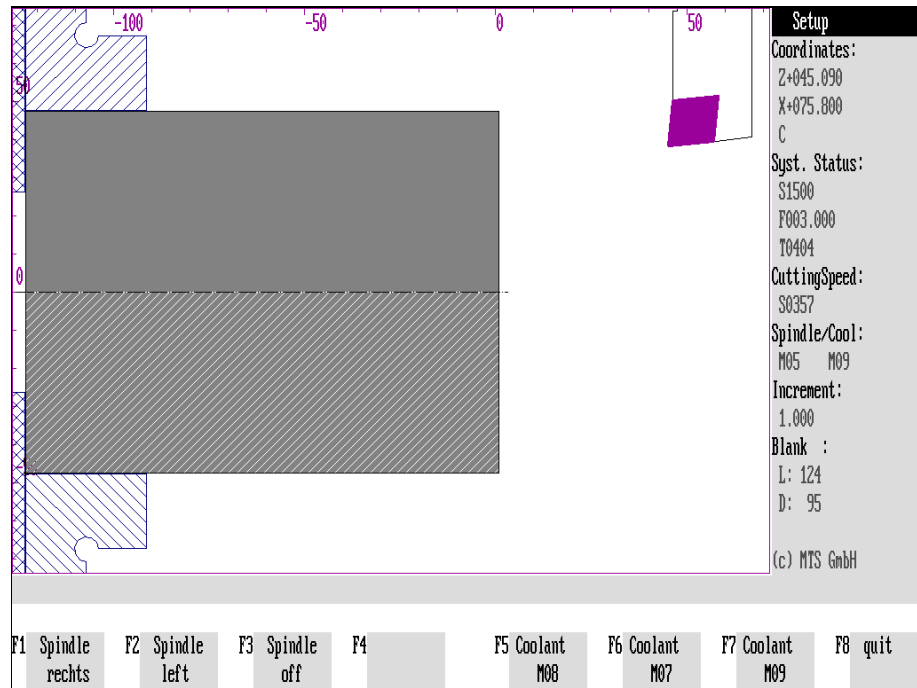
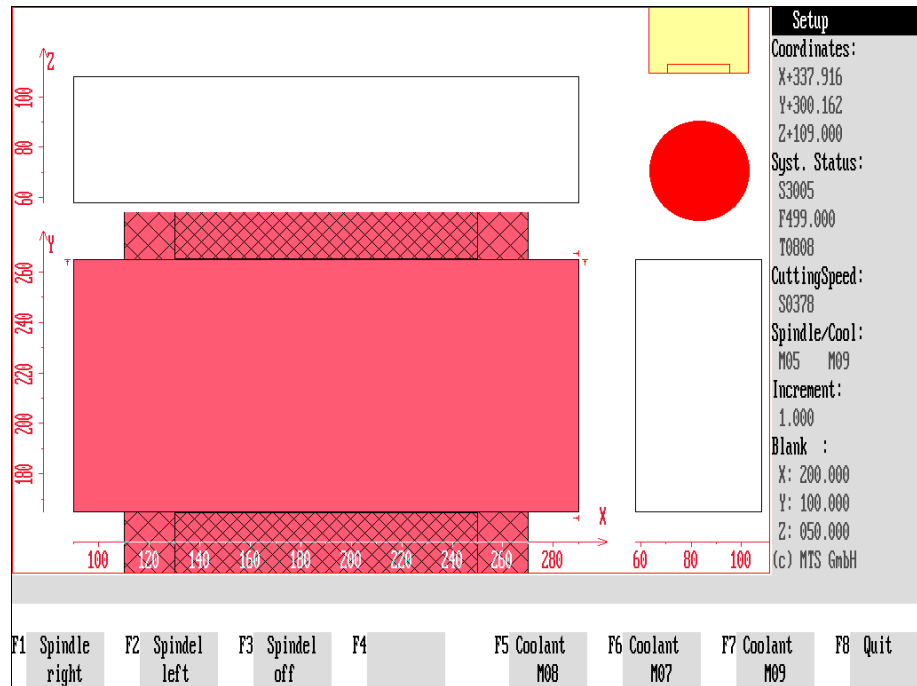
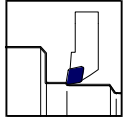


Figure 4.5-5:  
CNC Milling,  
Setup Mode;  
"Spindle and coolant  
control" menu.



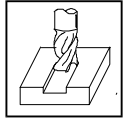
## Speed and Feedrate

### CNC Turning



Provided no performance restrictions have been made in the configuration, the **minimum possible speed** of the Simulator for Turning is **10 rpm** and the **maximum 9,000 rpm**; and the **minimum possible feedrate 0.001 mm/r** and the **maximum 50 mm/r**.

### CNC Milling

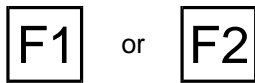


According to the standard configuration of the milling Simulator, the following minimum and maximum values also apply:

**Speed: 10 rpm - 6,000 rpm** and  
**Feedrate: 0.001 mm/min. - 999.9 mm/min. .**

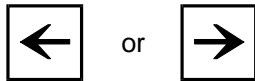
### Procedure

To determine the speed and feedrate, select the relevant function and enter the desired values in the information column:



or

**Feedrate (mm/r or mm/min.) / S speed:** Activating these functions releases the corresponding input field in the information column:

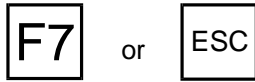


or

The cursor can be set to a certain field with the cursor keys.

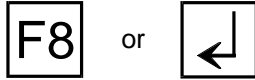
[Switching Data]

Enter the desired speed or feedrate.



or

**Cancel:** With or you cancel processing.



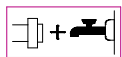
or

**Accept:** With or you confirm your entry. You then return to the starting situation.

### Note

If the stated value is outside of the range determined in the configuration, **the entry is ignored after the confirmation**, and the input field is released again for entries.

## Spindle and Coolant



Analogous to the switching commands of the NC programming, the spindle and the coolant pump can be switched on and off manually. The current machine status is displayed in the information column under "**Spindle**" and "**Coolant**". Here the following applies:

- M03 : Spindle switched on clockwise,
- M04 : Spindle switched on counterclockwise,
- M05 : Spindle switched off,
- M07 : Coolant pump, stage 1 switched on,
- M08 : Coolant pump, stage 2 switched on,
- M09 : Coolant pump switched off.

### Procedure

The functions in the menu for spindle and coolant control are simple switch on/off functions to set up/off the machine tool.



If the CNC keyboard or graphic tablet is used, the machine controls are executed directly on the processing level of the Setup Mode.

Figure 4.5-6:  
CNC Turning,  
Schematic of the  
menu sequence for  
choosing the  
increment values.

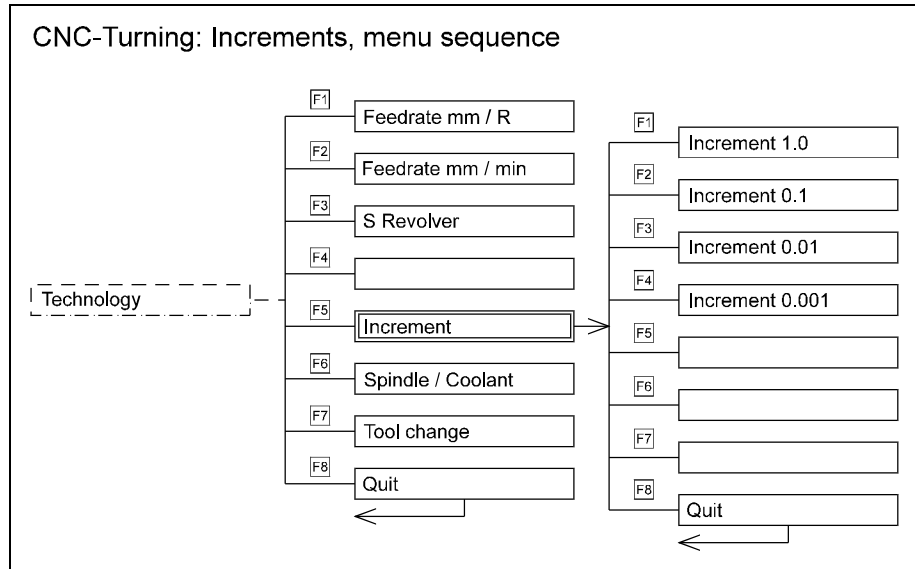


Figure 4.5-7:  
CNC Turning,  
Setup Mode;  
Increment values.

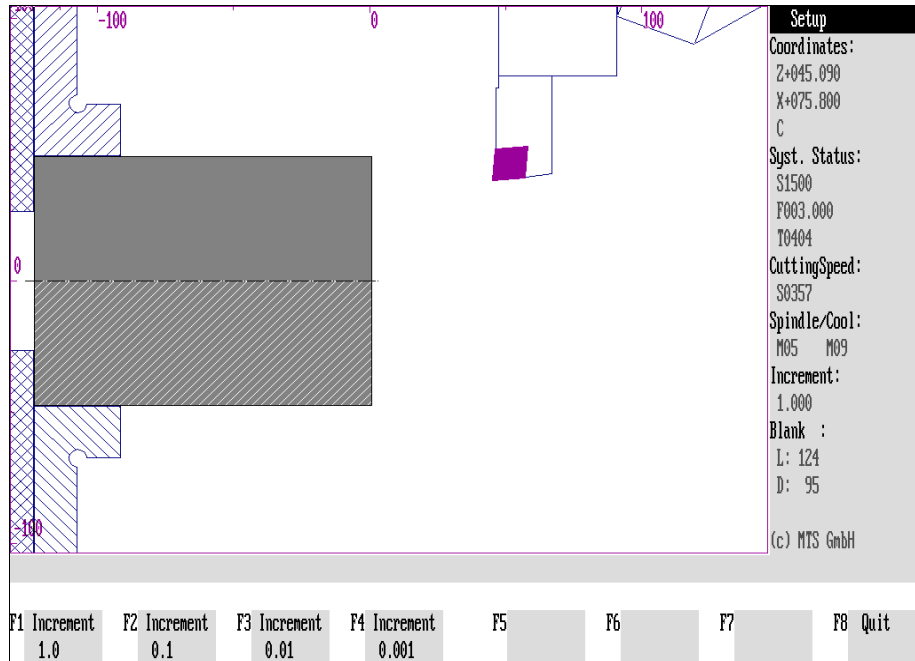
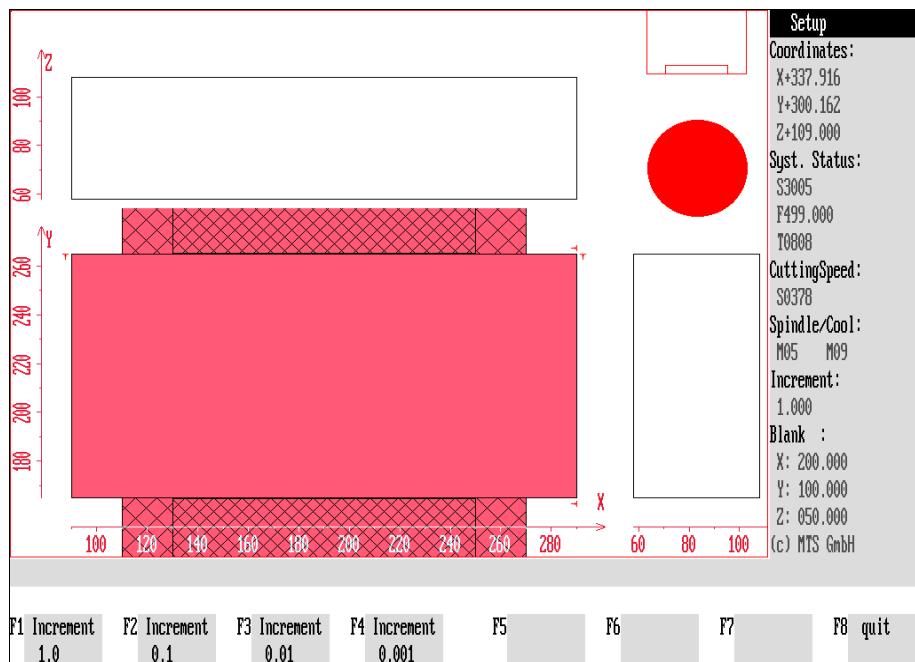


Figure 4.5-8:  
CNC Milling,  
Setup Mode;  
Increment values.



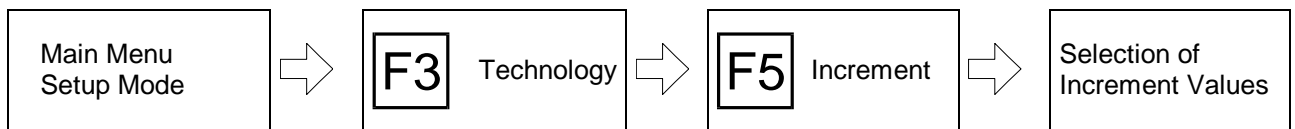
## Increment Values

"Increment value" determines the step length (in mm) of the tool carrier (or the machine table) when pressing the direction key once. The increment therefore defines the **movement unit** for machining in manual mode.

The currently active increment value in each case is displayed under "Increment" in the information column. At the same time you can read the increment value of the tool carrier in motion or of the machine table in feed motion in the corresponding change of the coordinate values.

The CNC Simulator offers the choice between four increment values: 1.0 mm, 0.1 mm, 0.01 mm or 0.001 mm:

### Selection:

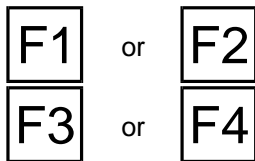


The option to switch on the increment value is called up with **F5** proceeding from the menu of the technology functions.

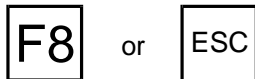
### Procedure

Then select the desired increment with one of the function keys:

You then return automatically to the starting situation.



**Increment 1.0/0.1/0.01/0.001:** You choose the desired increment with the function keys **F1** to **F4**.



**Return:** You cancel processing with **F8** or **ESC**.



When the CNC keyboard or graphic tablet is used, the change of the increment value is performed directly on the processing level of the Setup Mode.

Figure 4.6-1:  
CNC Turning,  
Schematic of the  
"Override/Times"  
menu

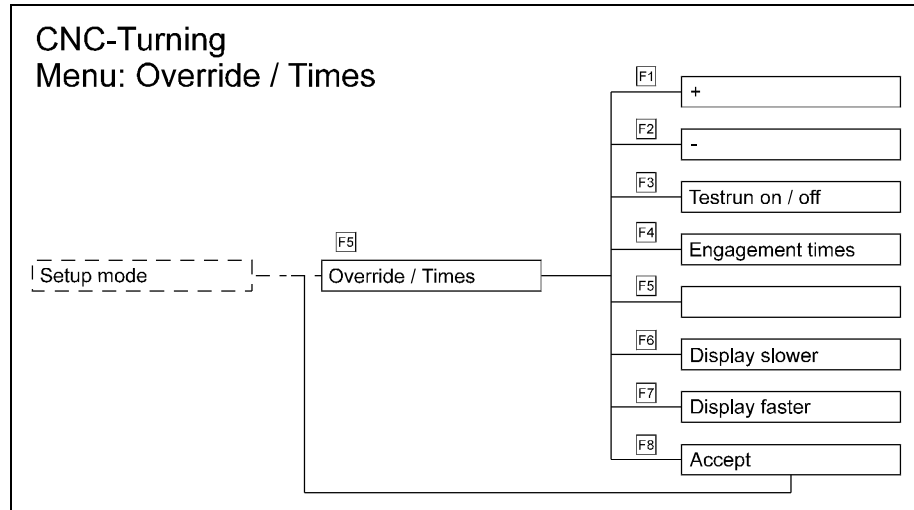


Figure 4.6-2:  
CNC Turning,  
Setup Mode;  
"Override/Times"  
menu.

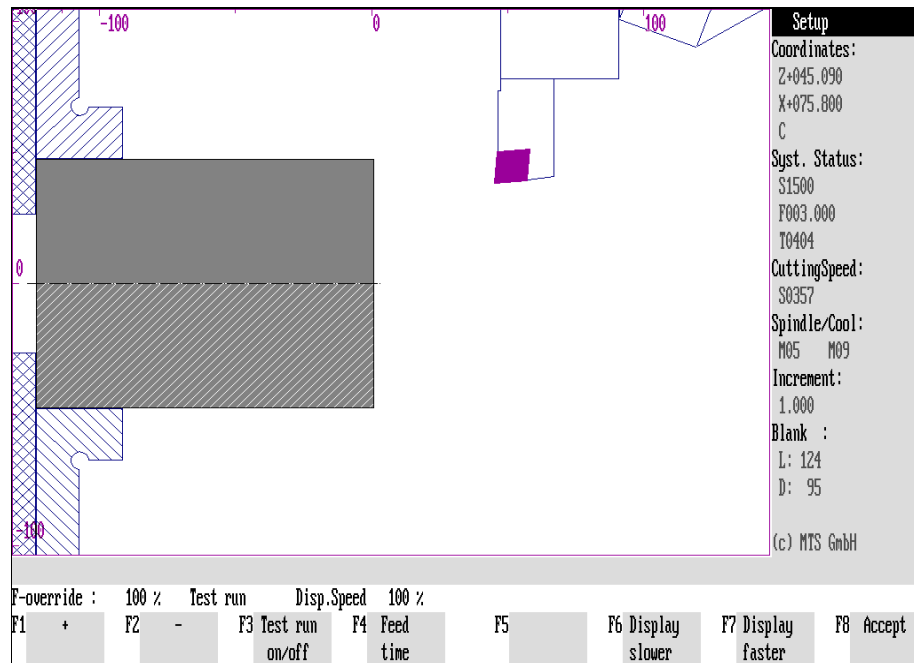
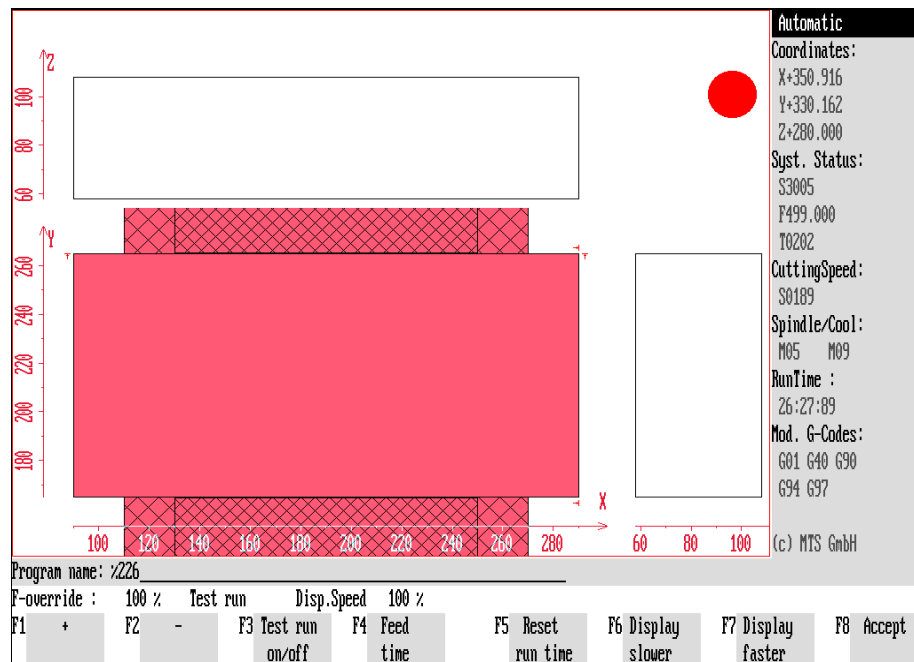


Figure 4.6-3:  
CNC Milling,  
Setup Mode;  
"Override/Times"  
menu.



## 4.6 Override/Times

The additional function "Override/Times" is available both in the Setup Mode and in the Automatic Mode. Feedrate and time settings of the CNC Simulator can be modified with it. Furthermore, the additional function also allows "access" to the established engagement times of the tools.

### Override

With "Override", the feedrate can be increased or reduced. This function is used frequently to check the movements of the tool during the test run. The override of the CNC Simulator can be set from 10% to 150%.

### Test Run/Slow Motion

The workpart is machined at an override of 100% in "real time", i.e. its presentation in the CNC simulation takes the same time as the actual workpart production with a machine tool.

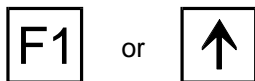
To speed up the simulation of complex machining operations - without changing the technology data of an NC program - a so-called "test run" also exists. In this mode, the workpart machining is run at maximum speed. However, this depends upon the computing power of your personal computer. The "slow motion" setting also permits gradual setting of the presented machining speed. This can be used, for instance, to obtain a good workpart machining flow for a PC of low computing capacity.

### Note

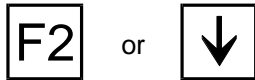
The time calculation which is shown in the information column remains unchanged by the settings for the "test run" and "slow motion"; and corresponds to the actual machining time - however, considering the set override.



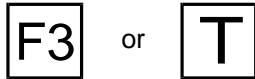
The override or the test run are set directly in the "Override/Times" submenu. Call it up with **F5** proceeding from the main menu of the Setup Mode. The current override (in percent) is then displayed in the dialogue line:



or



or



or



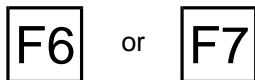
**+ / -:** To change the override, the desired percentage value (10% - 150%) is set with **F1** or **F2** or the cursor keys **↑** or **←** (decreasing) or **→** or **↓** (increasing).

**Test run on/off:** You switch the test run on or off with **F3** or the **T** key. After switching on, the message "Test run" appears in the dialogue line next to the percentage for the override, it disappears when switching off again.

**Engagement times:** This function permits a direct "access" to the established engagement times of the tools. Since the corresponding submenus have been already discussed when dealing with the turret/magazine (cf. above), they are not repeated here.



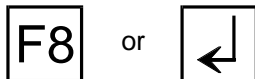
**Zero running time:** You set the calculated machining time ("running time") back to zero with **F5**.



or



**Slow motion slower/faster:** The displayed machining speed can be set between 10% and 1000% with this function.



or



**Accept:** With **F8** you confirm your settings and return to the main processing level of the Setup Mode.



With **ESC** you cancel processing without taking over new set values.

Figure 4.7-1:  
CNC Turning,  
Setup Mode;  
Machining with manual  
control.

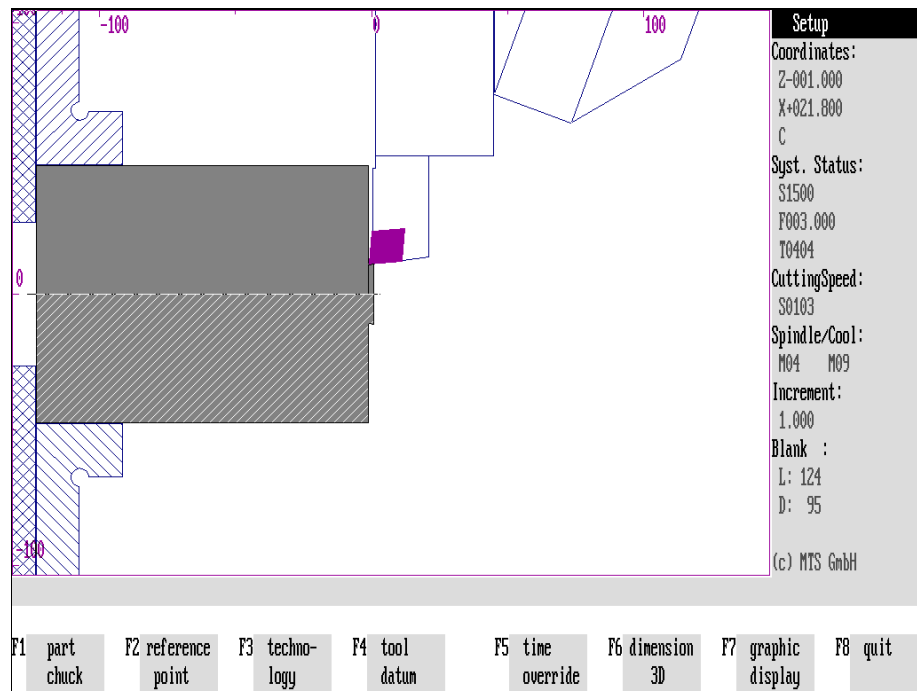
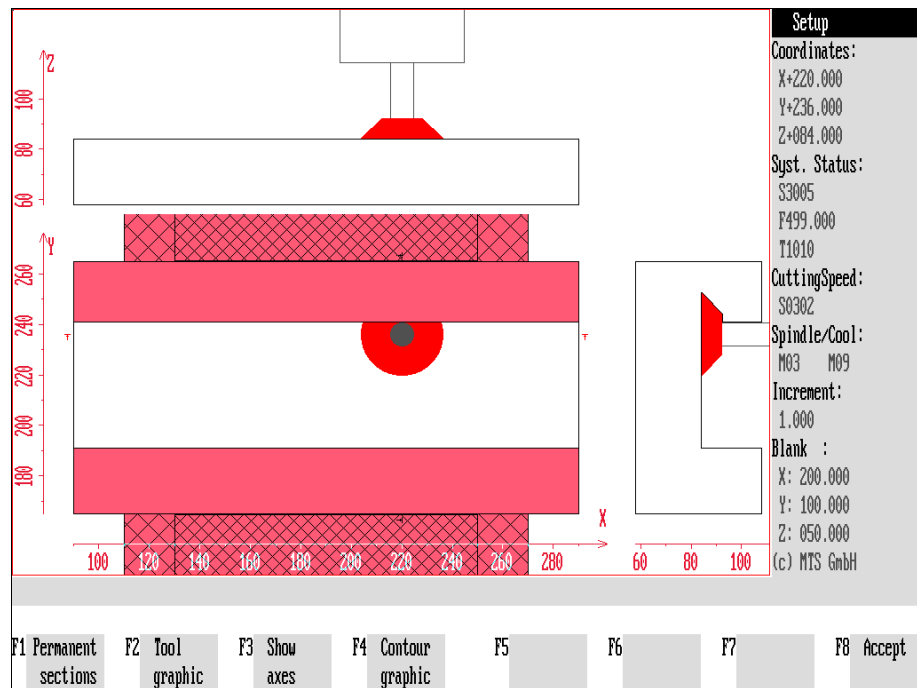


Figure 4.7-2:  
CNC Milling,  
Setup Mode;  
Machining with manual  
control.

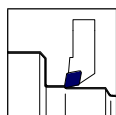




## 4.7 Machining with Manual Control

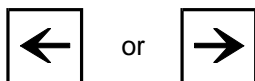
Machining with manual control is possible at any time after calling the Setup Mode with **F3** - provided no other function is active and not completed yet. After switching on the CNC Simulator the reference point should be approached first and the compensation values of the tools should then be entered.

### 4.7.1 CNC Turning



After you have set up the CNC Turning Simulator and entered the desired technology data, the blank or the workpart can be machined with manual control. Analogous to the NC commands, it is possible to move the tool slide in **feed motion** and in **rapid traverse motion**.

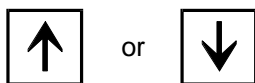
#### Feed Motion Parallel to the Z Axis



The tool slide is moved in Z direction with the cursor keys **←** or **→**. If the spindle is switched on, the workpart can be machined and the cutting is shown on the screen.

**Collisions** with the tool holder etc. are indicated by the corresponding collision or error messages.

#### Feed Motion parallel to the X Axis



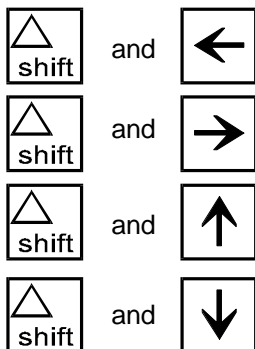
The tool slide is moved in X direction with the cursor keys **↑** or **↓**. If the spindle is switched on, the workpart can be machined and the cutting is shown on the screen.

**Collision** with the tool holder etc. is indicated by respective collision or error messages.

#### Note

In feed movements, feedrate and spindle speed are taken into account. In addition, the movement of the tool slide is coupled with the selected increment value (cf. above). In many cases, faulty operations result from not having switched on the spindle or from having determined wrong direction for spindle rotation, from cutting edge collisions or from choosing the feedrate too low.

#### Rapid Traverse in Z and X Direction

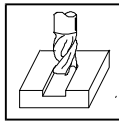


Moving in rapid traverse is effected analogous to the feed movement. You press the **shift** key and simultaneously the corresponding cursor key for the desired movement direction on the PC keyboard.

The rapid traverse speed is determined in the configuration and can be between 1000 and 10000 mm/min.

**Collisions** and other errors are displayed in rapid traverse as well. In this case, of course, workpart machining is not possible .

## 4.7.2 CNC Milling



After you have set up the CNC Milling Simulator and entered the desired technology data, the blank or the workpart can be machined with manual control. Analogously to the NC commands in **feed motion** and in **rapid traverse motion** the movements of the spindle head or of the machine table are possible here as well.





For the sake of clarity, we discuss the movement of the spindle head only, in the following.

### Feed Movement parallel to the Z Axis (G17)



or



The spindle head is moved in the Z direction with the cursor keys  or . If the spindle is switched on, the workpart can be machined and the cutting process is shown on the screen.



**Collisions** with the tool holder etc. are indicated with collision or error messages.

### Feed Movement parallel to the X Axis (G17)



or



The spindle head is moved in the X direction with the cursor keys  or . If the spindle is switched on, the workpart can be machined and the cutting process is shown on the screen.



**Collisions** with the tool holder etc. are indicated with collision or error messages.

### Feed Movement parallel to the Y Axis (G17)



or



The spindle head is moved in the Y direction with the  or  keys **on the numerical keypad** of the PC keyboard. If the spindle is switched on, the workpart can be machined and the cutting process is shown on the screen.

Here **collisions** with the tool holder etc. are shown by collision or error messages.

#### Note

In feed movements, the feedrate and the spindle speed are taken into account. The movements of the spindle head and of the machine table are coupled with the selected increment value (cf. above).


In many cases, faulty operations result from not having switched on the spindle or from having determined wrong direction for spindle rotation, from cutting edge collisions or from choosing the feedrate too low.

### Rapid Traverse in Z, X and Y Direction



and



Movement in rapid traverse is analogous to the feed movement. For this you press the  key on the PC keyboard and simultaneously the desired cursor key or key on the numerical keypad of the PC keyboard for the desired direction.



and



The rapid traverse speed is defined between 1000 and 10000 mm/min in the configuration.



and

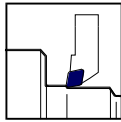


In rapid traverse, **collisions** and other errors are indicated as well. Naturally enough, workpart machining is not possible here.

## 5. Tool Management

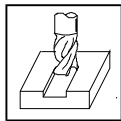
Version 5 of the CNC Simulator has a tool management function with a novel structural concept and an extended functional application field. The program provides almost all DIN tool types and tools as standard options, and allows all common tools to be defined (parameterized tool geometry). Naturally, the tool management includes options for editing the available tool files, i.e. modification of existing tools and deletion of those no longer required.

### CNC Turning



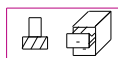
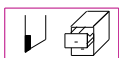
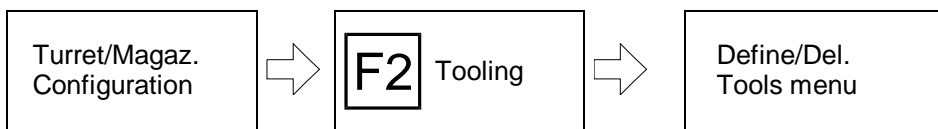
Turning tools are defined [according to DIN] and entered into the tool file with their data for geometry and for fastening of reversible tips, including the data on infeed and relief angle, toolholder and tool holding fixtures (adapters).

### CNC Milling



The same applies for the CNC Milling Simulator. Milling tools available in its standard data files are: face end mills, shank end mills, slot drills, T-slot, shell end, angular, radial, concave and side cutters. Additionally, various drill types are available as well, such as tap drills, reversible tip and stepped drills, as well as reamers and countersinks.

### Selection:



The starting point is the Turret/Magazine Configuration menu (see also [chapter 11](#)):

**Define/Delete Tools:** To access the **Define/Delete Tools menu**, continue with the **F2** function key.

### Password

To prevent unauthorized access to the Define/Delete Tools menu, it can be protected by a password. If you have specified a password during program configuration, enter this password after selecting Define/Delete Tools, and confirm your entry by pressing **↵**. Subsequently, the Define/Delete Tools menu is loaded.

### Note

If you wish to change the turret or magazine configuration only, you can use **F1** to access the Tooling function for selecting a tool from the Define/Delete Tools menu and placing it in a previously marked turret/magazine location (see also [chapter 4](#)).



**The only function of the Define/Delete Tools menu is to define of new tools, toolholders and tool adapters, and the modification or deletion of existing ones.**

**Please note that modifications of a tool file may necessitate changes in existing setup sheets or NC programs. It is therefore recommended that only necessary tool changes are made, and that subsequently the setup sheets and NC programs containing data of the changed tool are checked. In case of doubt, (rather than changing an existing one), generate a new tool!**

Figure 5.1-1:  
CNC Turning,  
Define/Delete Tools  
menu schematic

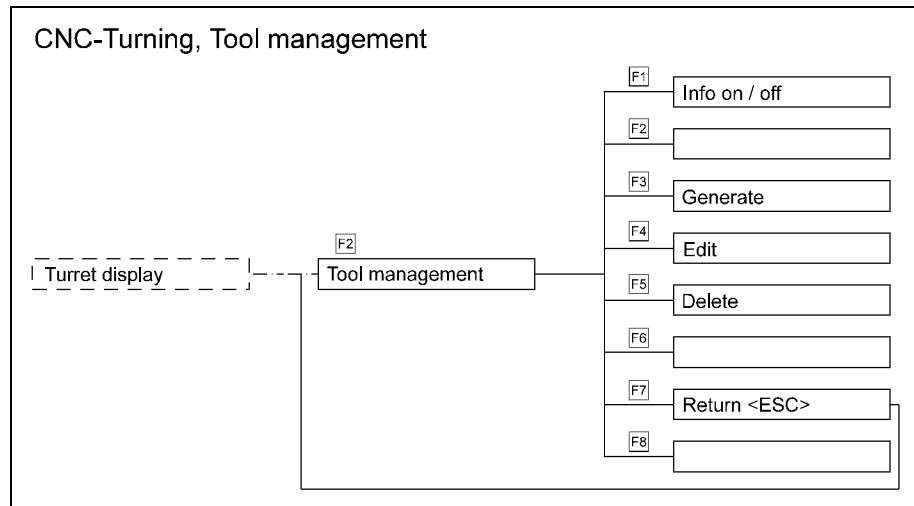
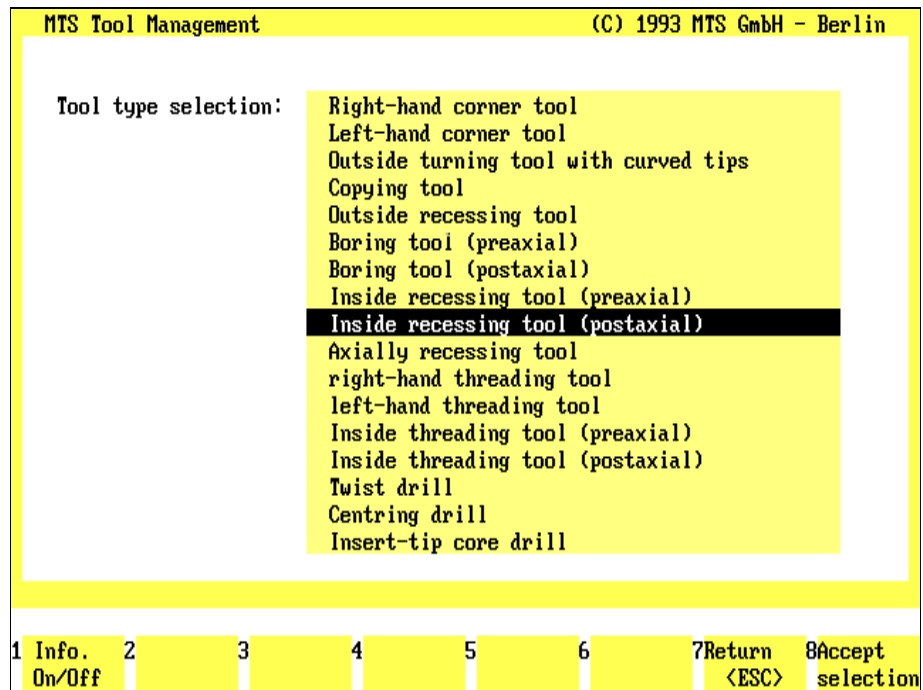


Figure 5.1-2:  
CNC Turning,  
Setup Mode;  
Define/Delete Tools  
menu.



## 5.1 CNC Turning: Define/Delete Tools Menu

The Define/Delete Tools menu is a separate program component of the CNC Simulator for the generation, definition, management, and/or modification of tools, (reversible tips), toolholders and tool adapters [e.g. according to DIN]. Accordingly, you can generate a tool file containing all the tools available for your lathe. The program includes 283 tools as standard defaults:

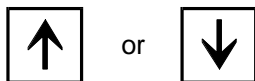
Number	CNC Turning Tool Types
26	Corner work tools, RH cutting
32	Corner work tools, LH cutting
20	External tools (circular bits)
12	Copying tools
16	External recessing tools
30	Internal corner tools (preaxial)
30	Internal corner tools (postaxial)
10	Internal recessing tools (preaxial)
10	Internal recessing tools (postaxial)
8	Axial recessing tools
8	External thread cutters, RH cutting
8	External thread cutters, LH cutting
6	Internal thread cutters (preaxial)
6	Internal thread cutters (postaxial)
40	Twist drills
12	Centering drills
9	Reversible tip drills

To support logical work progress, the Tool Management functions are accessed **immediately following the Turret/Magazine Configuration** menu. The process includes three dialogue-guided menu types:

- 1.) Define/Delete Tools main menu,
- 2.) Selection menu for procedural steps,
- 3.) Data entry menus for defining tool geometry.

### Main Menu

#### Procedure



or

Use the or cursor keys to select a specific tool type.

**F1**

**Info On/Off:** Use this function to toggle information on the current procedure on or off.

**F3**

**Create:** Select **F3** to define a new tool of the currently selected tool type.

**F4**

**Modify:** Select **F4** to modify an existing tool of the currently selected tool type.

**F5**

**Delete:** Select **F5** to delete an existing tool of the currently selected tool type.

**F7**

**Return:** Use **F7** or **ESC** to cancel the current operation and to return to the Turret/Magazine Configuration menu.

Figure 5.1.1-1:  
CNC Turning,  
Define/Delete Tools;  
Defining reversible tips  
for an LH cutting  
corner work tool.

MTS Tool Management		(C) 1993 MTS GmbH - Berlin	
- preaxial Boring tool -			
Tool name: ALT/521			
Insert tip: Name: DCMT 11T304			
Type:	D	Plan angle:	
Cutting edge radi	0.4	Side rake angle	107.5
Length of cut. ed	6.25	End angle	
2nd radius:	0.0		
Clearance angle:	7		
-- available types of tips: -----			
S - 90	Square	T - 60	Triangular
C - 80	Rhombic	c - 100	Rhombic
D - 55	Rhombic	U - 35	Rhombic
E - 75	Rhombic	e - 105	Rhombic
W - 80	Trigon		
			Generate
1 Info.	2help	3	4
On/Off	diagram		
		5Read in	6
		7Return	8Accept
		<ESC>	<ENTER>

Figure 5.1.1-2:  
CNC Turning,  
Define/Delete Tools;  
defining tool data for a  
reversible tip drill.

MTS Tool Management		(C) 1993 MTS GmbH - Berlin	
-- Insert-tip boring tool --			
Tool name: ALT/557			
Drill data: Name: R-34-UCMX-06			
Diameter:	34	Shank diam.:	32
Length:	148	Max. drill. dpth	76
Radial		Radius of cutting	0.2
adjustment:	2.8	Sense of rotation	left
Enter drill diameter			
			Generate
1 Info.	2help	3Spindle	4
On/Off	diagram	CW/CCW	
		5Read in	6
		7Return	8Accept
		<ESC>	<ENTER>

### 5.1.1 Creating a New Tool

Depending on the tool type, the definition of a new tool requires two to four working steps:

- 1.) To start with, the geometrical shape of the reversible tip, tool nose or drill is specified,
- 2.) secondly, define the working tool nose (except thread cutting tools and drills),
- 3.) then add the appropriate toolholder (exception: drills), and
- 4.) the tool adapter.



A separate data entry menu is available for each operating step.

#### Data Entry: Tool with Reversible Tip or Drill

After accessing the Create function by selecting **F3** in the Define/Delete Tools menu, the Data Entry menu for defining the cutting edge / reversible tip or drill is loaded immediately:

##### Data Entry Menu

Similar to the Define/Delete Tools main menu, the Data Entry menu consists of two sections: in the upper screen section you enter the tool description and the desired geometrical values. In this area, you can also display additional information on the entries made.

The function keys at the bottom of the screen indicate further steps for defining or editing tool data:

**F1**

**Info On/Off:** With this function you toggle on and off clarifying information on the current operation.

**F2**

**Show Help Graphic:** Use **F2** to display a help graphic clarifying the geometrical parameters required for the entry.

You can then return to the Data Entry menu by pressing either **ESC** or **F7**.

**F3**

**[Spindle Rotation RH/LH]:** When defining a drill, you can specify the rotation direction (right-hand/left-hand) of a drill by pressing **F3**.

**F5**

**Copy Data:** If you wish to define a tool by using and modifying the data of an existing tool, you can copy its data. Press **F5** to select the tool (see below).

**F7**

or **ESC**

**Return:** Use **F7** or **ESC** to cancel the operation, and to return to the main Define/Delete Tools menu.

**F8**

or

**Accept:** Use **F8** or to confirm the currently entered values, and to continue with the following menu for selecting toolholder or adapter, etc.



Before adding the tools which you actually need in your production process by entering them into the tool files of your CNC Simulator to have them at your disposal during the simulation, it is recommended that you first study and practice the application of the Define/Delete Tools menu.

Figure 5.1.1-3:  
CNC Turning,  
Define/Delete Tools;  
Help graphic for  
defining a reversible  
tip.

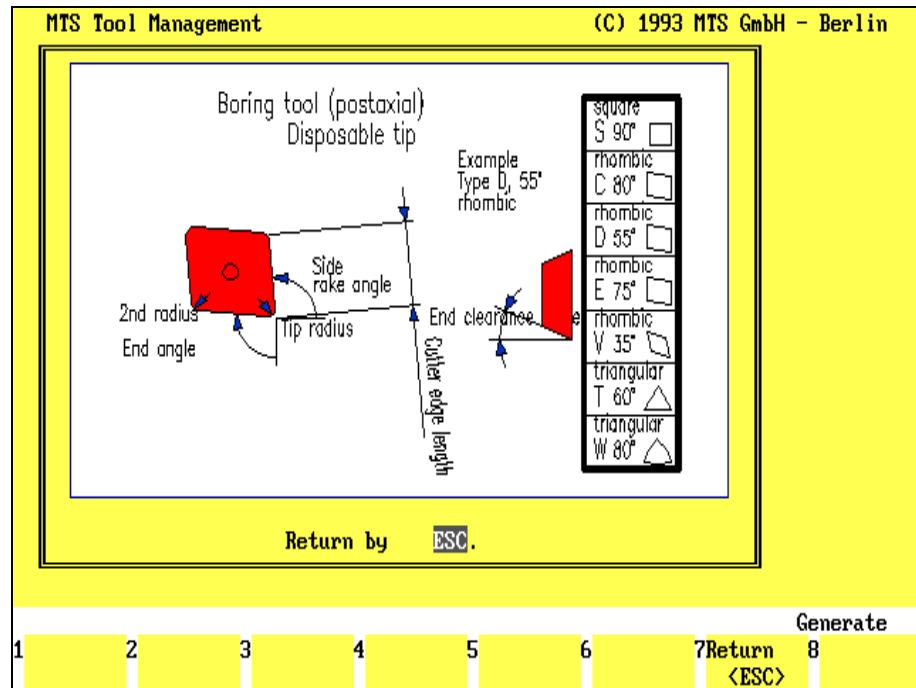
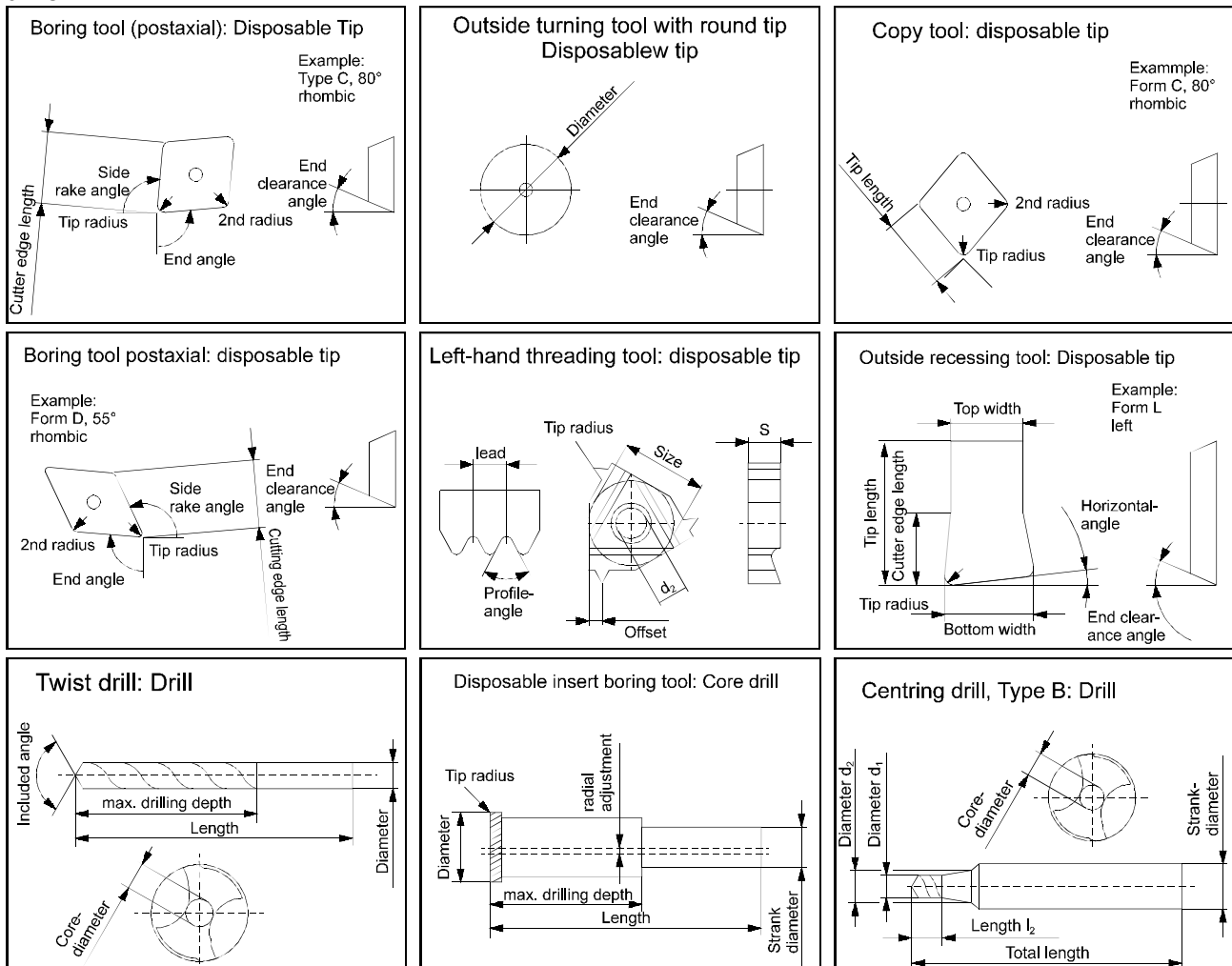


Figure 5.1.1-4:  
CNC Turning, Define/Delete Tools; example of entry parameters for the selection of reversible tips and drills.





**Procedure**

Use the key to move from input field to input field.



or



To move the cursor within the input field, use the or cursor keys.



or



Use the or cursor keys to select the input field for the tool entering angle.



or



Use the key to insert a character, and the key to remove a character.

**[Tool Name]**

Enter the name or tool number in this input field.

**[Reversible Tip]**

The entry data for reversible tips depend on the tool and reversible tip type selected. Use the Help Graphics and/or the Info On/Off options to obtain further details on the geometrical shapes and parameters ([see figures on the facing page](#)).

**[Entering Angle]**

The entering angle specifies at which angle the reversible tip of the internal or external corner tool is fixed. You have the option of defining the entering angle according to the DIN standards, or based on the side or end cutting angle (see also illustrations and help information). Select with the cursor key the desired entry, and use the key to move to the right input field. Enter either the DIN code letter or the size of the angle.

**[Drill Data]**

The entries defining the drill geometry depend on the drill type. Use the Help Graphics option to obtain information on the various parameters ([see figure](#).)



or



**Accept:** Use or to confirm the entered data, and to continue with the subsequent specifications regarding the toolholder or the tool adapter, etc.

**Note**

An overview of the available reversible tips and toolholders is enclosed as an appendix to this Programming Instructions.

Figure 5.1.1-5:  
CNC Turning,  
Define/Delete Tools;  
intermediate menu for  
copying existing tool  
data. The desired  
criteria are entered in  
the input fields.

MTS Tool Management		(C) 1993 MTS GmbH - Berlin	
-- Insert boring tool --			
		--- Tool name: -----	
		-	
		--- Insert boring tool: -----	
		Name:	
		Diameter:	
		Shank diameter	
		Length	
		Overall length:	
		Sense of rotation	
		---	
		--- Mounting: -----	
		Mounting name:	
		Stand. mount.:	
No record selected!			
			Edit
1 Info.	2Delete	3Spindle	4Scroll
On/Off	form	CW/CCW	down
			5Scroll
			up
			6Select
			7Return
			<ESC>
			8Continue/
			Accept

## Copying Existing Tool Data

If you wish to base your tool definition on already filed data use of the Copy Data function. With this function you search a specific data record to be used as a 'rough specification'. Use the Data Entry menu to modify then the existing values to meet your requirements. (See also chapter 4.2.2.):

**F5**

Tool Selection

**Copy Data:** The Copy Data function is accessed from the Data Entry menu for tool geometry by pressing **F5**.

This opens an intermediate Tool Selection menu consisting of three sections, with **initially no entries** in the input fields: The left part of the screen is a window for graphic display of the tool; the empty input fields for tool geometry (including toolholder and tool adapter) are on the right-hand side of the screen, and the function keys at the bottom of the screen are marked with the respective functions:

**F1**

**Info On/Off:** Press **F1** to display information on/off in each input field of the information column.

**F2**

**Clear Input Data:** To select specific data records from the file of available tools, enter the desired tool specifications in the respective input fields. Use **F2** to clear all entries in the input fields.

**F3**

**Spindle Rotation RH/LH:** If the rotation direction of the spindle (M03 or M04) is to be used as a selection criterion, you can activate this option by pressing **F3**.

**F4**

or

**F5**

**Scroll Forward/Backward:** Provided that you have already used the Search & List option to select specific data records from the file of available tools, you can now scroll through all the tools meeting the selection criteria.

**F6**

**Search & List:** With this function you can select specific entries from the extensive file of available tools. The entries in the input fields are used as search criteria. After the completion of the Search & List function, all tools meeting the search criteria can be accessed by scrolling through a new list.

**F7**

or


**ESC**

**Return:** Use **F7** or **ESC** to cancel the current operation, and to return to your starting situation.

**F8**

or



**Continue/Accept:** Use **F8** or  to accept the currently displayed data for further tool definition.

### Procedure

In this example, we assume that you can enter several geometrical values as search criteria:

**TAB**

Use the **TAB** key to move from input field to input field.

[Enter Values]

Enter the desired geometrical values in the input fields.

**F6**

**Search & List:** Use **F6** to select from the available tools file the tools meeting your search criteria. The number of selections is indicated in the information line as "Selected X of Y data records searched."

**F4**

or


**F5**

**Scroll Forward/Backward:** From the new list of tools in the list window, you can select the appropriate tool by pressing **F4** or **F5**.

**F8**

or



**Continue/Accept:** Use **F8** or  to confirm the currently displayed tool data as a tool definition. At the same time, you return automatically to the Data Entry menu.



**If you do not know the specific search criteria, leave the input fields empty and select the Search & List function to display all available tools in the current tool list.**

Figure 5.1.1-6:  
CNC Turning,  
Define/Delete Tools;  
Reversible tips and drill  
definition menu  
sequence schematic.

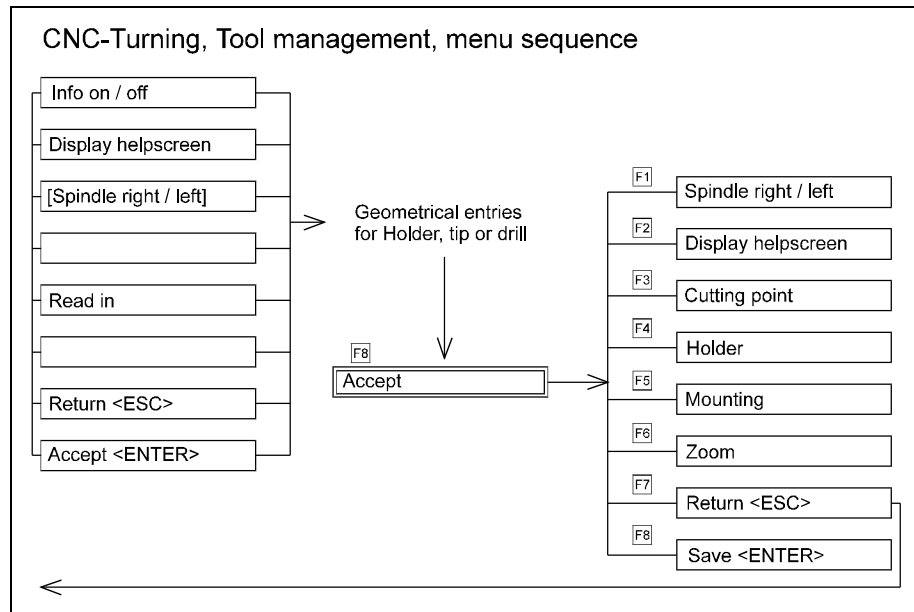
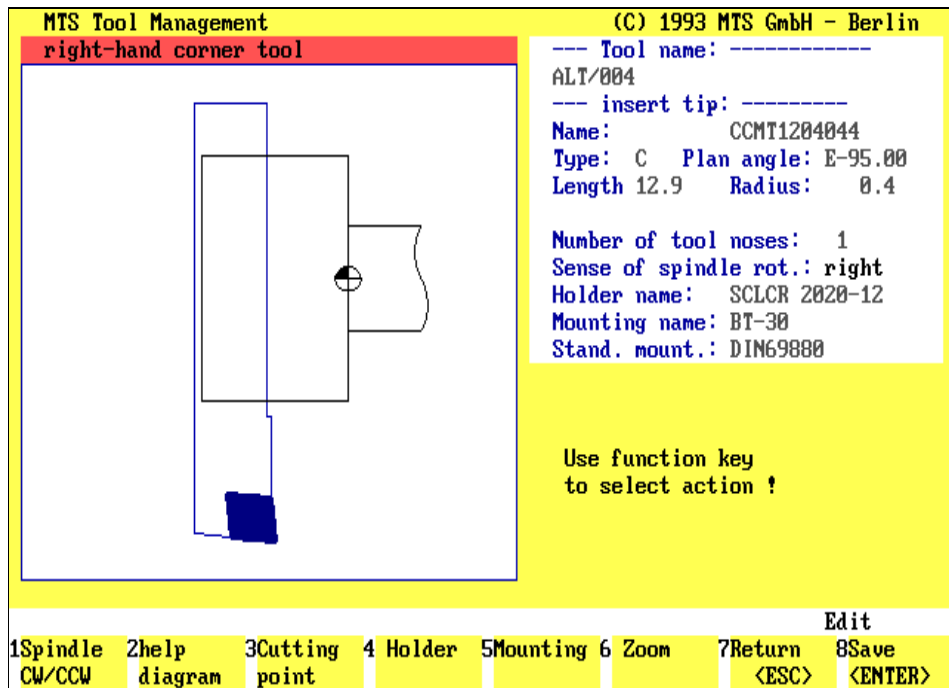


Figure 5.1.1-7:  
CNC Turning,  
Define/Delete Tools;  
Selection menu for  
defining toolholder, tool  
adapter, and cutting  
point.



## Further Steps in Tool Definition

After entering the geometrical data a menu for further steps, for defining the data for the toolholder, tool adapter and, if applicable, the cutting point, follows. The window to the left of the input data shows a graphic presentation based on the previously specified tool geometry.

**F1**

**Spindle Rotation RH/LH:** This function specifies the rotation direction (right-hand or left-hand rotation) of the spindle holding the tool.

**F2**

**Show Help Graphic:** Use **F2** to display the help graphics for the toolholder and the tool adapter.

You can return to your starting position by pressing **F7**.

**F3**

**Cutting Edge Point:** Select **F3** to define the cutting edge point or the nose of the reversible tip.

**F4**

**Toolholder:** This function defines a toolholder.

**F5**

**Tool Adapter:** This function defines a tool adapter.

**F6**

**Zoom:** If you wish to enlarge or reduce the displayed graphic, select **F6** (see below).

**F7**

or


**ESC**

**Return:** Use **F7** or **ESC** to finish your procedure, and to return to the main Define/Delete Tools menu.

**F8**

or



**Save:** Use **F8** or  to save the complete data record of a tool under the specified tool name.

### Note

After completing one of the main functions described above, you return to the selection menu either to access the next function or to conclude the operation. The functions described above also depend on the context. This means that they are not available with all tool types. For example, no toolholder is defined for drills since this was already specified in their geometry under "drill data" (see above). Also, with thread cutting tools and drills, the function Cutting Point is omitted.



The sequential order of the further tool definition steps is irrelevant. In the following description, we concentrate on the main functions.

Figure 5.1.1-8:  
CNC Turning,  
Define/Delete Tools;  
Cutting Edge Point  
definition, menu  
sequence.

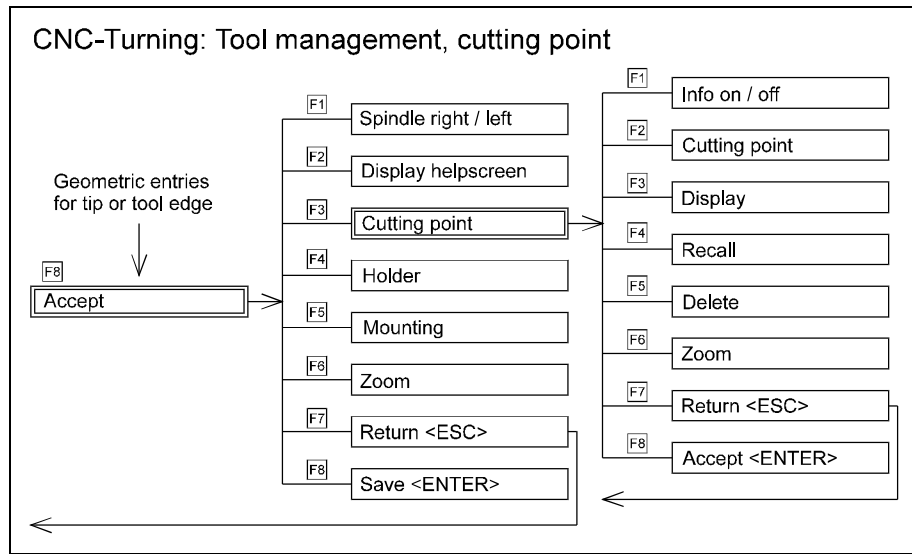


Figure 5.1.1-9:  
CNC Turning,  
Define/Delete Tools;  
Defining the cutting  
edge point.

**MTS Tool Management** (C) 1993 MTS GmbH - Berlin

right-hand corner tool

--- Tool name: -----  
ALI/004  
--- insert tip: -----  
Name: CCMT1204044  
Type: C Plan angle: E-95.00  
Length 12.9 Radius: 0.4

Number of tool noses: 1  
Sense of spindle rot.: right  
Holder name: SCLCR 2020-12  
Mounting name: BT-30  
Stand. mount.: DIN69800

**Tool nose**

serial No	K	I
1.1	0.400	-0.400
2		
3		

K: 0.400 I: -0.400

Entries are confirmed by pressing the "ACCEPT" softkey

Edit							
1 Info.	2Cutting	3Display	4Recall	5 Delete	6 Zoom	7Return	8Accept
On/Off	point					<ESC>	<ENTER>

## Cutting Edge Point Definition

### Compensation Values

As a default value, the control system of the CNC Turning Simulator always first uses the **tool reference point** as a reference when defining the tool coordinates. This reference point is a fixed point on the turret. It is normally located on the stop face of the tool adapter. In order to allow the calculation of the "tool tip", the distances between the real **cutting edge point** and the tool reference point must be defined for each tool. The calculated differences of these values are saved as **tool compensation values** in the so-called "**compensation value register**".

Due to the fact that the reversible tip has a radius, there is actually no real geometrical tool tip at all. In order to enable the program to set the real cutting edge point through internal calculations, the location of the "theoretical" tool tip, or nose of the reversible tip, is entered through this menu. In doing so, you have the choice between several possible cutting edge points. With this definition, you specify the compensation values, the work quadrant and the work range of the tool. This enables the program to evaluate the precise location of the cutting edge of the given tool (see also Chapter 4).

Starting from the selection menu, the location of the tool noses is accessed by pressing **F3**. In this menu, however, the tool nose does not refer to the tool reference point but to the center of the cutting edge circle. This is the basis for the program's internal calculation of exact compensation values.

### Cutting Edge Point Menu

The window displays an enlarged tool tip. The color highlighted arrow indicates one possible location for the theoretical cutting edge point (of the tool nose).

On the right, the main parameters are listed, and the display below contains a **list of the available tool noses**, as well as the current coordinate values of the arrowhead. **Only the first data record of the list is used to calculate the compensation values ("running count No. 1"); the rest of the data is for information only.**

The function keys at the bottom of the screen show the available functions:

**F1**

**Info On/Off:** Use **F1** to display help info in the information line.

**F2**

**Cutting Edge Point:** This function key is a switch showing in succession the possible cutting edge nose locations by means of the color-highlighted arrow.

**F3**

**Display:** If you have manually entered the coordinates of the cutting edge point, you can have it displayed as a graphic by pressing **F3**.

**F4**

**Reset:** This function places the arrow to the point in the graphic corresponding to the currently highlighted field of the listing of the cutting edge noses.

**F5**

**Delete:** Use **F5** to delete the currently selected entries in the listing of the cutting edge noses.

**F6**

**Zoom:** To enlarge or reduce the displayed graphic, select **F6** (see below).

**F7**

or

**ESC**

**Return:** Use **F7** or **ESC** to save the entered values, and to return to your starting position.

**F8**

or

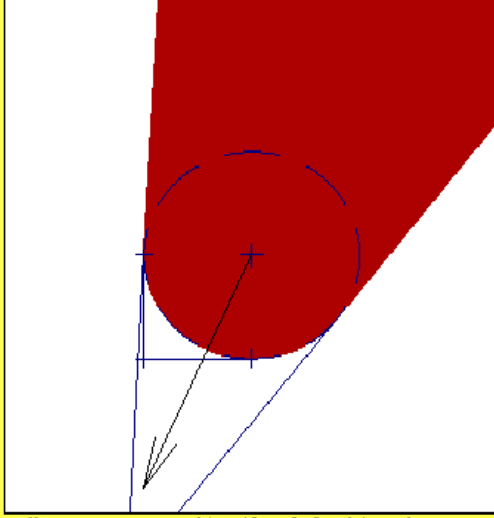
**↵**

**Accept:** Use **F8** or **↵** to confirm the values for the cutting edge nose marked by the arrow, and to transfer them to the selected field in the listing.

Figure 5.1.1-10  
 CNC Turning,  
 Define/Delete Tools;  
 Manual definition of a  
 new cutting edge point.  
 For example, to get a  
 finishing allowance of  
 0.5 mm for Z parallels  
 (diameter), the  
 allowance was entered  
 in this example by  
 using Y: -0.65 (radius).  
 This finishing  
 allowance is valid only  
 when working without  
 Nose Radius  
 Compensation (NRC).

**MTS Tool Management** (C) 1993 MTS GmbH - Berlin

- Left-hand corner tool -



--- Tool name: -----  
 ALT/508  
 --- insert tip: -----  
 Name: UBMT 160404  
 Type: V Plan angle: S-93.00  
 Length 16.6 Radius: 0.4

Number of tool noses: 1  
 Sense of spindle rot.: left  
 Holder name: SDJCL 2020 11  
 Mounting name: BT-30  
 Stand. mount.: DIN69880

**Tool nose**

serial No	K	I
1.1	-0.400	-0.400
2		
3		

K: -0.400 I: -0.9\_

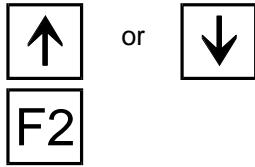
You may overwrite the default values

1 Info. 2Cutting 3Display 4Recall 5 Delete 6 Zoom 7Return 8Accept  
 On/Off point <ESC> <ENTER>



**Procedure**

The definition of a cutting edge point compensation vector can be made either automatically with the graphic supported entry of the cutting points, or by entering the coordinate values manually. For defining the cutting nose as a manual data entry you can on the finishing allowances (see figure), as long as the tool nose radius compensation is not activated during machining:



or

In both cases, first select with the cursor keys or the field in the list of the tool noses to enter the values.

**Cutting edge point:** For the graphically supported definition of cutting edge points, select the desired value with the function key; the currently selected cutting edge point is indicated in the graphic with a color-highlighted arrow.

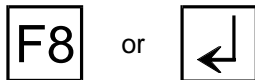


In manual entry of the coordinate values, you activate in succession the coordinate fields "K" and "I" ("K" refers to the Z axis, and "I" to the X axis), and

[Enter Values]

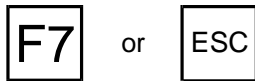
then enter the desired values; as an orientation you should use one of the set cutting edge points in the graphic.

After you have entered the coordinate values, reactivate with the key the input field in the list of the cutting edge points.



or

**Accept:** Use or to transfer the values of the selected tool nose, or the manually entered values, into the listing.



or

**Return** With or you save the entered values and return to the starting situation

**Notes**

Please note that the MTS program control currently uses only the uppermost tool nose listing entry for the calculation of compensation values. For example, if you want to define a recessing tool based on two tool noses, you actually need to define two separate tools differing only in their cutting edge point.

As the geometry of the tool is defined under tool management the values of the cutting edge point (correction value vector) are also considered in the correction values of "X" and "Y". For the definition of the tool nose the finishing allowance can only be entered for the parallels of "X" and "Y" as far as the procedure takes place without the cutting radius compensation value function having been switched on.

If, however, you wish to get a contour parallel dimensioning (with the cutting edge point correction having been switched on) you change the respective radius values of the tool cutting edge point in the correction value register.



**The coordinates of the cutting edge point are saved in the compensation value registers under the parameters I and K (cutting edge vector).**

Figure 5.1.1-11:  
CNC Turning,  
Define/Delete Tools;  
Defining a toolholder.

MTS Tool Management
(C) 1993 MTS GmbH - Berlin

**-- Boring tool (preaxial) --**

--- Tool name: -----  
ALT/521

--- insert tip: -----  
Name: DCMT 11T304  
Type: D Plan angle: S-107.50  
Length 6.25 Radius: 0.4

Number of tool noses: 1  
Sense of spindle rot.: left  
Holder name: SDJCL 1212  
Mounting name: BT-30  
Stand. mount.: DIN69880

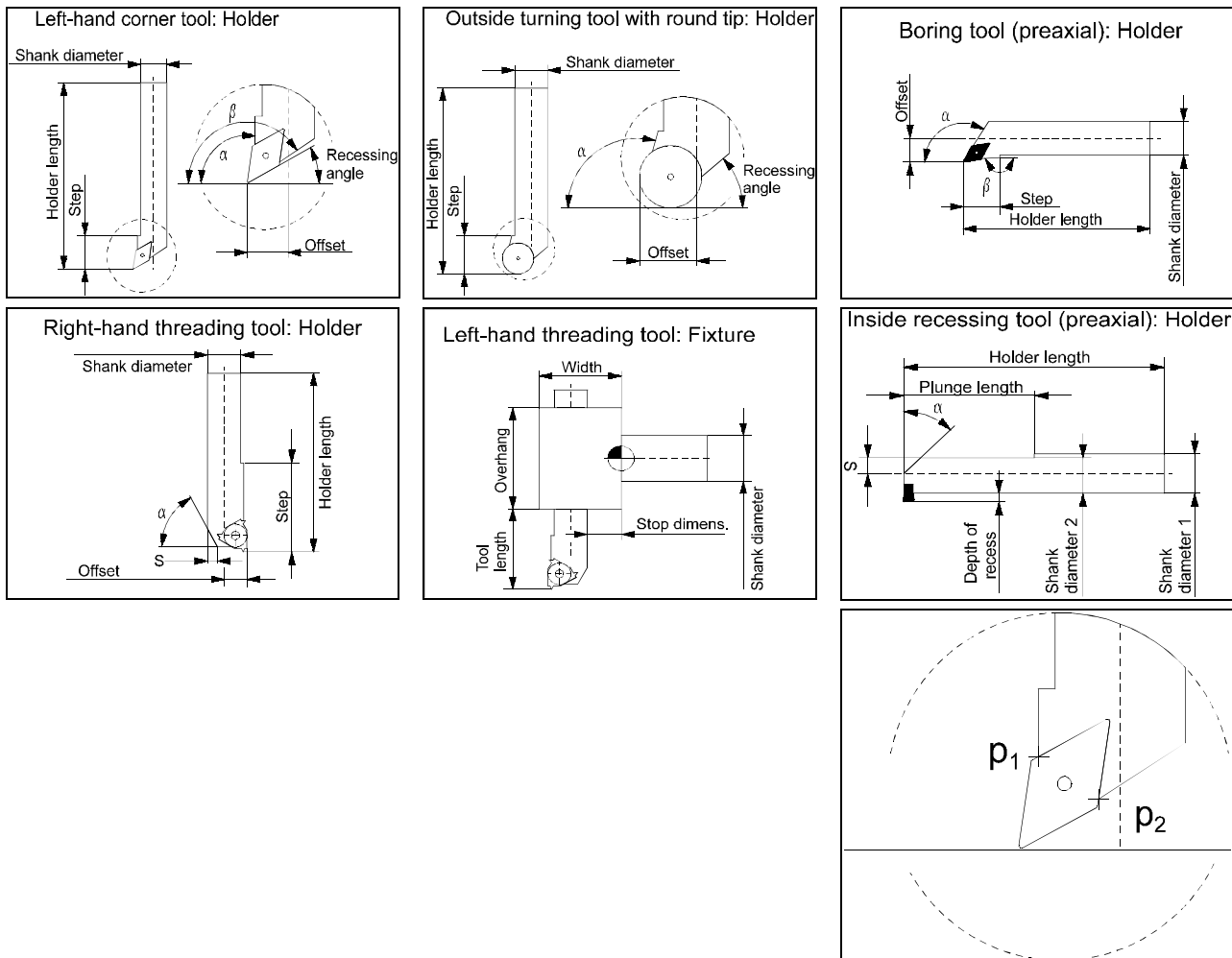
**Holder:**  
Holder name: SDJCL 1212  
Shank section: 12  
Holder length: 125  
Infeed angle 16.5

Alpha: 109.5 Beta: 180.0

Name of the holder

1 Info.	2help	3offset	y4 p1	5 p2	6 Zoom	7Return	8Accept	Edit
On/Off	diagram	es/no				<ESC>	<ENTER>	

Figure 5.1.1-12:  
CNC Turning, Define/Delete Tools; examples for various toolholders and entry parameters:



## Defining a Toolholder

Unlike the definition of a drill, where the entire tool geometry is specified in a single step in the previously described Data Entry menu, a toolholder is defined in a menu separate from those of other tool types. Starting with the selection menu, the Define Toolholder menu is selected by pressing **F4**.

### Toolholder

Like the Tool Selection menu, the Define Toolholder menu is divided into three areas: on the left-hand side, a tool is displayed based on the currently valid data. On the right, below the major parameters listed are the input fields for entering the geometrical data for the toolholder.

The function keys at the bottom of the screen are labeled with corresponding functions:

**F1**

**Info On/Off:** Use this function to toggle on/off help information on current operations.

**F2**

**Show Help Graphic:** Select **F2** to display the help graphic clarifying the entry parameters to be entered.

Subsequently you return to the Data Entry menu by pressing **ESC**.

**F3**

**Angled Yes/No:** To specify an angled toolholder, select **F3**. In the definition of a cranked toolholder, the input fields Step and Offset are added to the parameter list.

**F4**

or

**F5**

**p1 / p2:** Points  $P_1$  and  $P_2$  separate the range of the reversible tip that is located inside the toolholder from the cutting edge of the reversible tip (see figure). To define the toolholder, both points need to be defined on the reversible tip. The possible points are selected by repeatedly pressing these function keys. A modification of the location may also cause an automatic change in the infeed angle and in the angular pitch of the toolholder (alpha or beta, see figure).

**F6**

**Zoom:** To enlarge or reduce the displayed graphic, select **F6** (see below).

**F7**

or

**ESC**

**Return:** Use **F7** or **ESC** to cancel the current operation, and to return to your starting situation.

**F8**

or

**↵**

**Accept:** Use **F8** or **↵** to save the toolholder data, provided the data record is complete. For control, a graphic based on the toolholder data is displayed.

### Procedure

Depending on the toolholder type, several operational steps are required:

**TAB**

Use the **TAB** key to move from input field to input field.

**←**

Existing entries can be deleted by using the **←** cursor key or the **<BACKSPACE>** key.

**F4**

or

**F5**

**p1 / p2:** If previously only the shape of the reversible tip was specified, the graphic display is restricted to the reversible tip only, with color-highlighted crosses marking possible locations for points  $P_1$  and  $P_2$ . Use **F4** and **F5** to select the desired "attachment points" for the toolholder.

Alternately, the toolholder has been defined already, then new attachment points can be selected with the help of the displayed graphic by using **F4** and **F5**.

### [Enter Values]

Enter the desired geometrical values in the input fields.

**F8**

or

**↵**

**Accept:** Use **F8** or **↵** to accept the entry values for the toolholder.

Figure 5.1.1-13:  
CNC Turning,  
Define/Delete Tools;  
defining a toolholder.

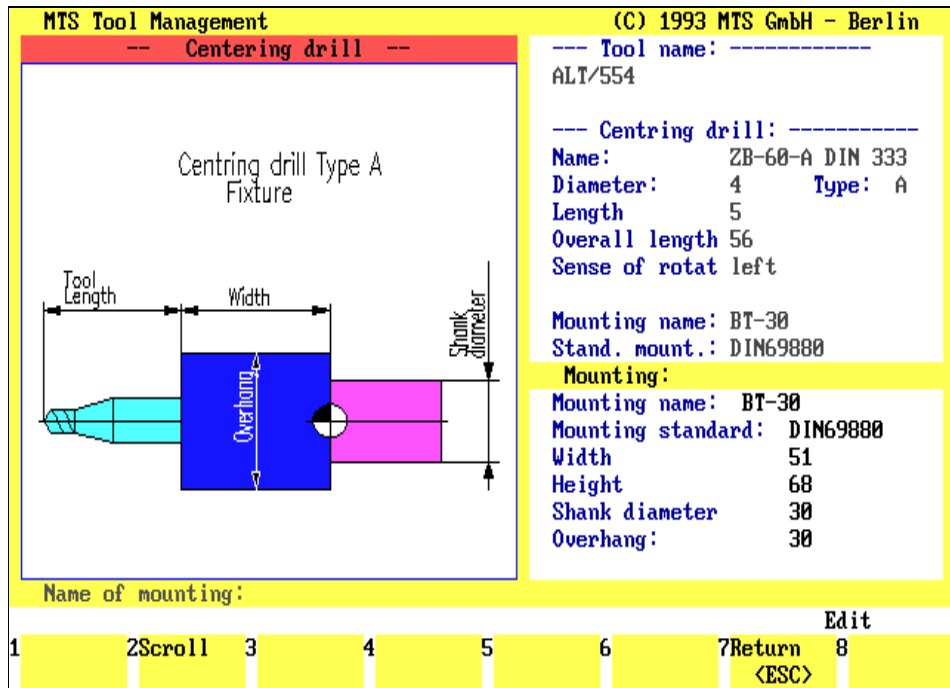
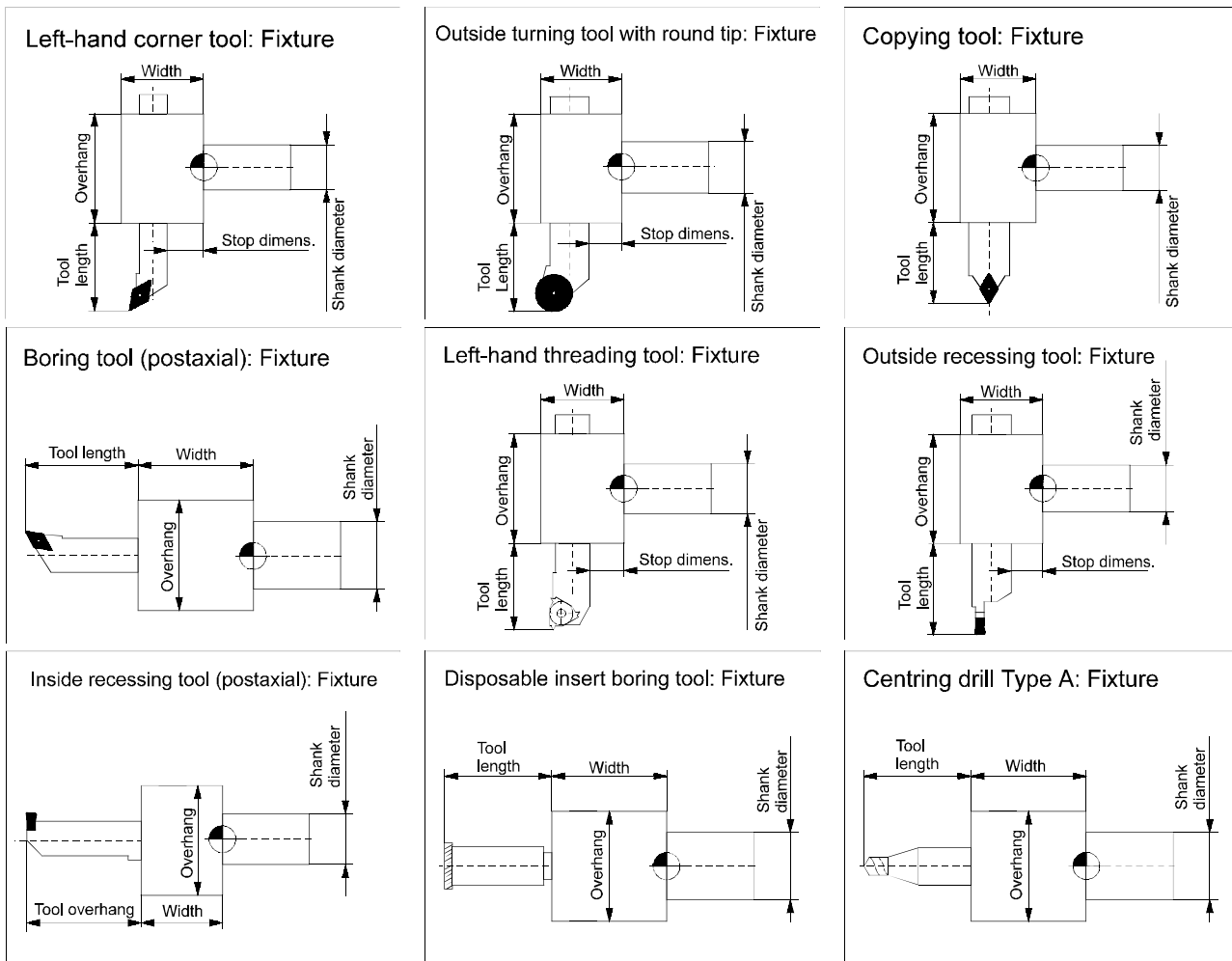


Figure 5.1.1-14:  
CNC Turning, Define/Delete Tools;  
examples of various tool adapters and entry parameters:



## Defining a Tool Adapter

For practical reasons, the definition of the tool adapter follows the definition of the tool itself. Like the toolholder, the tool adapter is defined in a separate menu: starting from the Selection menu, the Define Tool Adapter menu is accessed by pressing **F5**.

### Tool Adapter

The Define Tool Adapter menu has the same layout as the other data entry menus. A graphic of the already defined tool, or a help graphic is displayed in the window. On the right, the major parameters are listed; below them are the input fields to enter the geometrical data of the tool adapter. The function keys at the bottom of the screen are labeled with respective functions:

**F1**

**Info On/Off:** Use this function to obtain help information on the current operation.

**F2**

**Show Help Graphic:** Select **F2** to display a help screen clarifying the required entry parameters.

Subsequently, return to the Data Entry menu by pressing **ESC**.

**F6**

**Zoom:** To enlarge or reduce the displayed graphic, select **F6** (see below).

**F7**

or

**ESC**

**Return:** Use **F7** or **ESC** to cancel the current operation, and to return to the starting situation.

**F8**

or

**↵**

**Accept:** Use **F8** or **↵** to save the data for the tool adapter, provided the data record is complete. For control, a graphic based on the entered data is displayed.

### Procedure

Similar to the other data entry menus, you enter the desired values and designations in the input fields:

**TAB**

Use the **TAB** key to move from input field to input field.

**←**

Existing entries can be deleted by using the **←** cursor key or the <BACKSPACE> key.

[Enter Values]

Enter the desired geometrical values in the input fields (see also Fig. 5.1.1-14).

**F8**

or

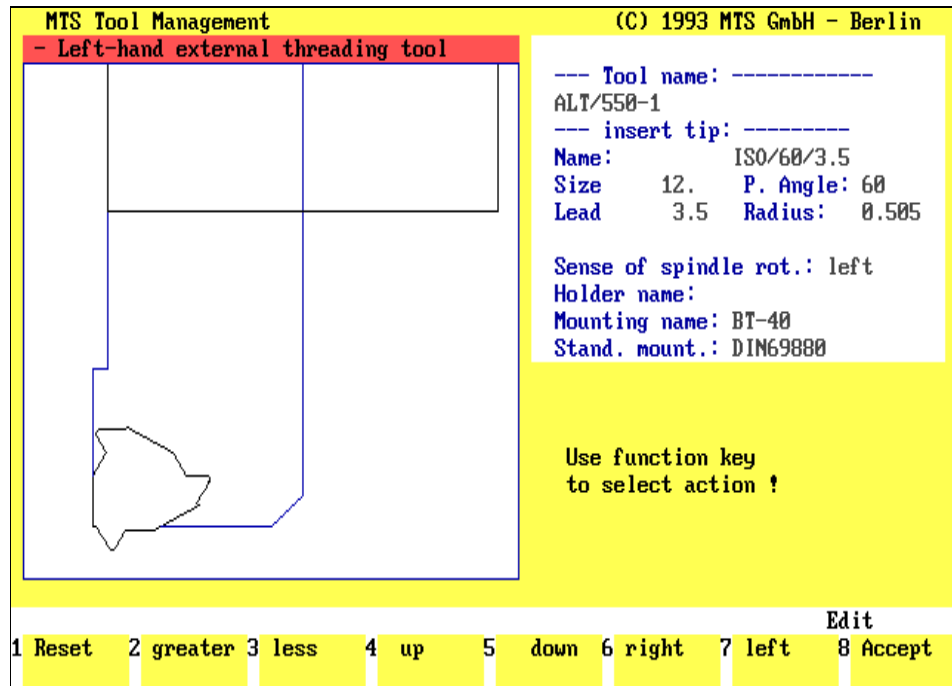
**↵**

**Accept:** Use **F8** or **↵** to accept the entry values.

### Note

As a rule, the mounting standard, and consequently also the shank diameter, is identical for all tools in a turret.

Figure 5.1.1-15:  
CNC Turning,  
Define/Delete Tools;  
Zooming.



## Zoom Function

To support the entering of geometrical data, several data entry menus within the Define/Delete Tools main menu feature a simple zooming function allowing the displayed graphic to be enlarged or reduced:

Zoom

The menu for the modification of the displayed graphic has the same structure as the data entry menus: the fully defined tool is graphically displayed in the window. The right-hand screen section remains unchanged during the use of the Zoom function; the function keys at the bottom of the screen are labeled with the corresponding functions:

F1

**Reset:** This option selects the default settings for the displayed graphic.

F2

or

F3

**Enlarge/Reduce:** Use F2 or F3 to enlarge or reduce the graphic display of the tool step by step.

F4

or

F5

**Up/Down:** Use F4 or F5 to shift the tool display vertically upward or downward step by step.

F6

or

F7

**Right/Left:** Use F6 or F7 to shift the tool display horizontally right or left step by step.

F8

or

↩

**Accept:** Use F8 or ↩ to accept the selected section of the graphic display, and to return to your starting situation.

Figure 5.1.2-1:  
CNC Turning,  
Define/Delete Tools;  
Menu sequence  
schematic for  
modifying an existing  
tool.

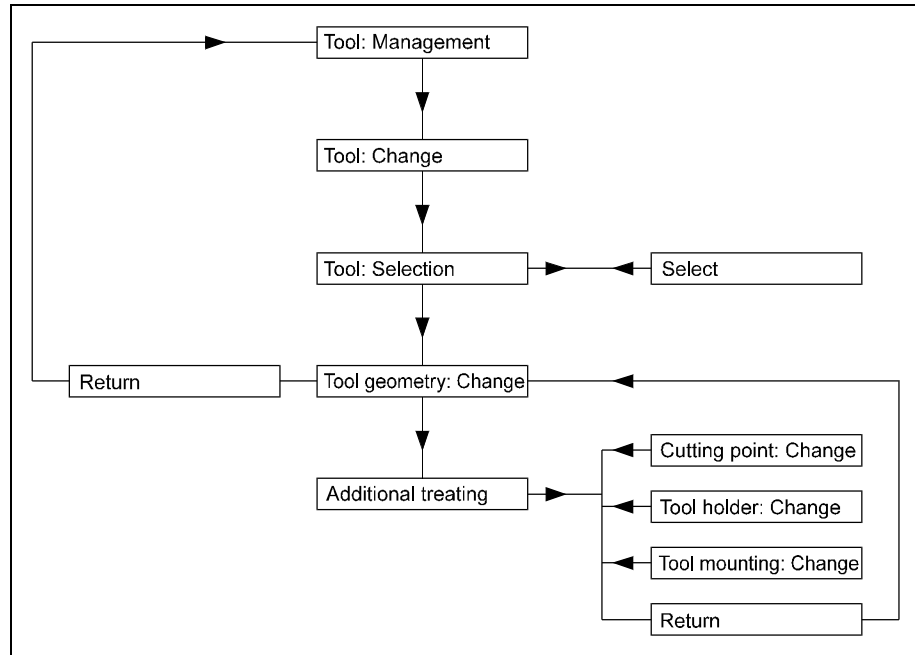


Figure 5.1.2-2:  
CNC Turning,  
Define/Delete Tools;  
Selecting a tool.

**MTS Tool Management** (C) 1993 MTS GmbH - Berlin

- Left-hand external threading tool

--- Tool name: -----  
**ALT/550-2**

--- insert tip: -----  
Name: **ISO/55/3.5**  
Size **12.** P. Angle: **55**  
Lead **3.5** Radius: **0.505**

--- Holder: -----  
Holder name:  
Length **150** Width: **25**

--- Mounting: -----  
Mounting name: **BT-40**  
Stand. mount.: **DIN69880**

Sense of spindle rot.: **left**

Name of the mounted tool (holder and mounting)

4.	of	9	cted	records			Edit
1	Info.	2	Delete	3	Spindle	4	Scroll
	On/Off		form		CW/CCW		down
							up
							Accept



### 5.1.2 Modifying an Existing Tool

In fact, the same steps are used to modify an existing tool of the available tools file as to create a new tool. Before you can modify a data record to meet your requirements, the tool to be modified needs to be selected (for this see the section [Copying Existing Tool Data](#) earlier in this chapter).

#### Procedure

The menus used to modify an existing tool have been discussed already. To avoid repetition, these menus are not described in detail here; only the sequence of the required steps is explained below:

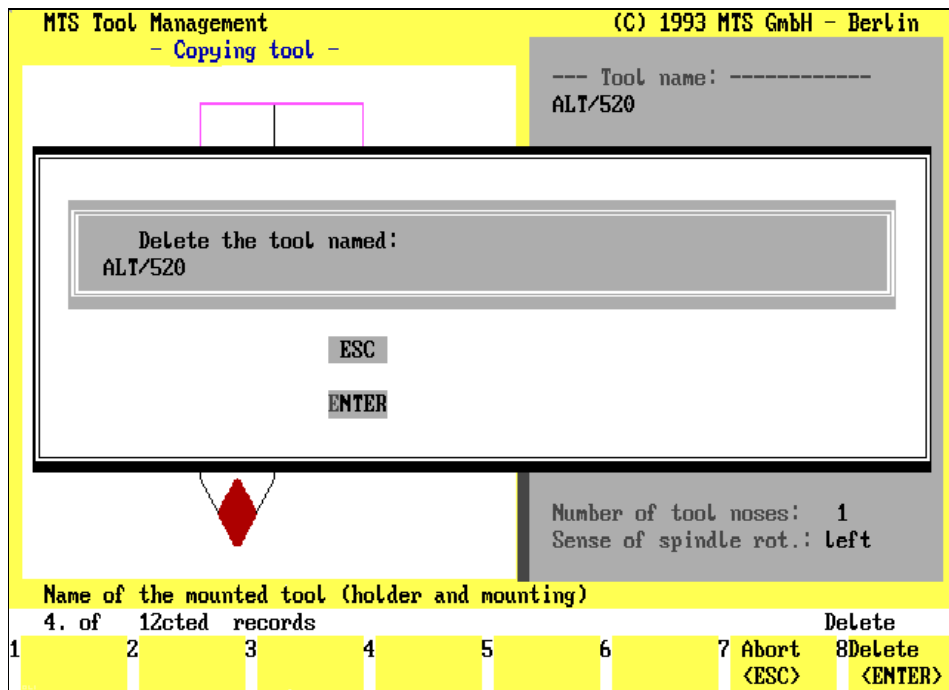
- 1.) Starting from the Define/Delete Tools main menu, select the Modifying function by pressing **F4**.
- 2.) Subsequently, like for copying an existing data record, an intermediate menu is loaded, allowing with the options to
  - specify certain search criteria,
  - search the file of available tools, and
  - select a specific tool from the resulting new list.
- 3.) Then, the program opens the data entry menu for reversible tip, tool cutting edge or drill, and you can modify all entries as required.
- 4.) After having concluded the entries regarding the tool geometry, the selection menu is opened, allowing further steps to be selected:
  - modify the cutting edge point or the tool noses;
  - modify the toolholder, and/or
  - modify the tool adapter.

After all required modifications have been completed, save them by pressing **F8**, and then press **ESC** to return to the Define/Delete Tools main menu.



**Please note that modifications of a tool file may necessitate changes in existing setup sheets or NC programs using this tool. It is therefore recommended that only necessary changes of tool files are made. When in doubt, (rather than changing an existing one), generate a new tool!**

Figure 5.1.3-1:  
CNC Turning,  
Define/Delete Tools;  
Control enquiry prior to  
deletion of a tool.



### 5.1.3 Deleting a Tool

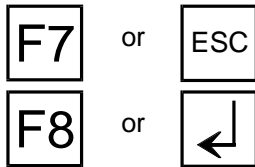
To delete an existing tool from the file of available tools is even easier than to modify the tool: you first select the tool to be deleted, and then you delete it.

#### Procedure

The menus applied have been discussed already. To avoid repetition, only the sequence of required steps is discussed here:

- 1.) Starting from the Define/Delete Tools main menu, select the Delete function by pressing **F5**.
- 2.) After using the Search & List function to select the file to be deleted, mark the file to be deleted in the list window.

Once you have confirmed your selection, the Delete function is activated, and as a control query the message, "Do you want to delete the tool named: ..... " is displayed.



**Cancel:** Use **F7** or **ESC** to cancel the operation.

**Delete:** Confirming the deletion by pressing **F8** or **↵** deletes the tool. Subsequently, you return to the Define/Delete Tools main menu.



**Please note that deleting a tool may necessitate changes in existing setup sheets or NC programs using this tool. It is therefore recommended that only necessary deletions of tool files are made. If possible, the default tools supplied with the program should not be deleted.**

Figure 5.2-1:  
CNC Milling,  
Define/Delete Tools;  
Main Menu schematic.

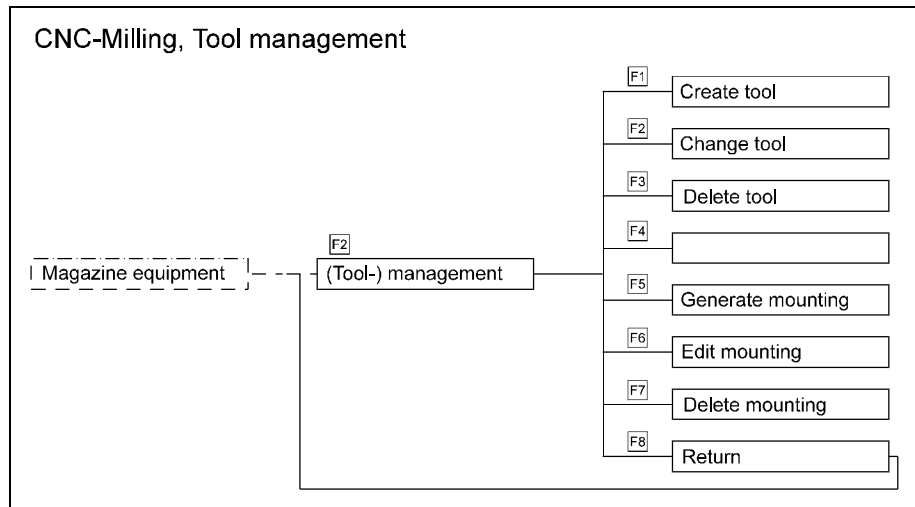
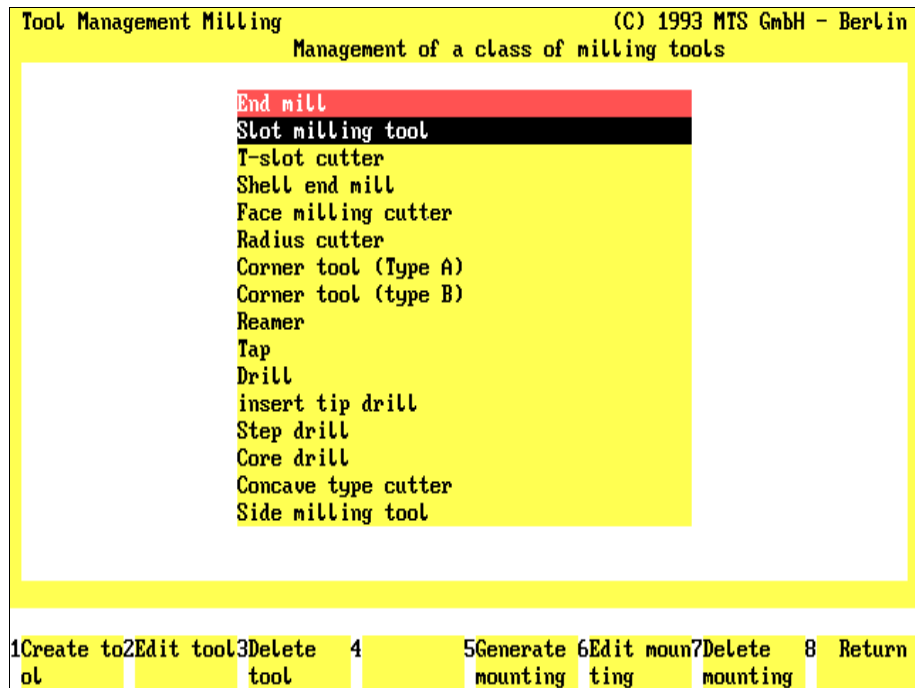
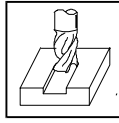


Figure 5.2-2:  
CNC Milling,  
Define/Delete Tools;  
Main Menu.



## 5.2. CNC Milling: Define/Delete Tools



The Define/Delete Tools menu is a separate program component of the CNC Milling Simulator, allowing the creation, definition, management, and/or modification of tools and tool adapters [e.g. according to DIN]. Accordingly, you can generate a tooling file corresponding to all the milling tools available on your milling machine.

The program supplies 326 tools as standard defaults:

### Milling Tools

Number	CNC Milling Tool Types
66	End mills
51	Slot drills
27	T-slot cutters
6	Shell end mills
5	Face end mills
16	Radial cutters
8	Angular cutters (type A)
8	Angular cutters (type B)
10	Reamers
23	Tap drills
37	Drills
10	Reversible tip drills
13	Stepped drills
16	Countersinks
10	Concave cutters
20	Side milling cutters

### Main Menu

The screen layout of the Define/Delete Tools main menu is divided into two sections: the upper screen area contains a listing of all available tool types; the field currently in use is highlighted in color. As usual, further steps for specifying or editing tool data are indicated on the function keys at the bottom of the screen:

#### Procedure



or





Use the  or  cursor keys to select the tool type.



or





**Create Tool/Tool Adapter:** To generate a new tool of the current tool type, select ; to define a new tool adapter, use .



or





**Modify Tool/Tool Adapter:** To modify an existing tool of the current tool type, select ; to modify an existing tool adapter, select .



or



**Delete Tool/Tool Adapter:** To delete an existing tool of the current tool type, select ; to delete an existing tool adapter, select .



or





**Return:** Use  or  to conclude the current operation, and to return to the Magazine Configuration menu.

Figure 5.2.1-1:  
CNC Milling,  
Define/Delete Tools;  
defining a shank end mill.

**Tool Management Milling** (C) 1993 MTS GmbH - Berlin

**Define slot cutter**

Tool length offset = 0

Name : \_

RotSens :                      No. of hobs

Shank diameter :                      :

Tip diameter :                      :

Tip length: :                      :

Overall length :                      :

Mounting:

Lgth of engag. slot :                      :

Length of mounting :                      :

Capacit.Min:                      Max:

Spdle. draw-in gear :                      :

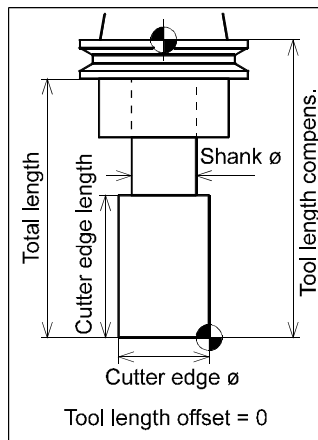
Length corr. data :                      :

Tool length offset :                      :

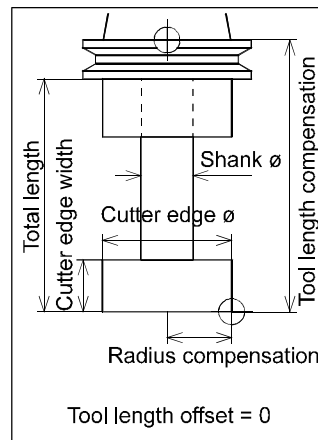
Tool radius offset :                      :

1Display 2 Graph. h3 Select 4 5Delete 6Alt. rot.7 Default 8Create to  
 tool elp mounting form sense data ol

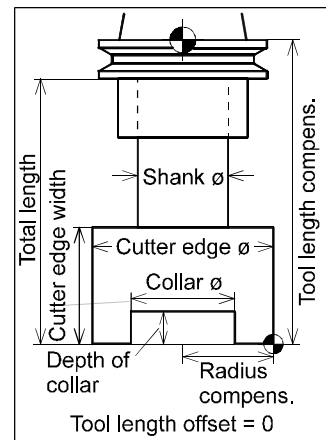
End mill / Slot drill



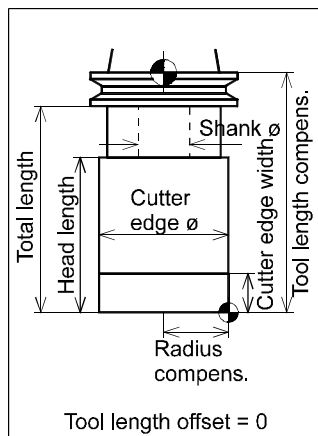
T-slot cutter



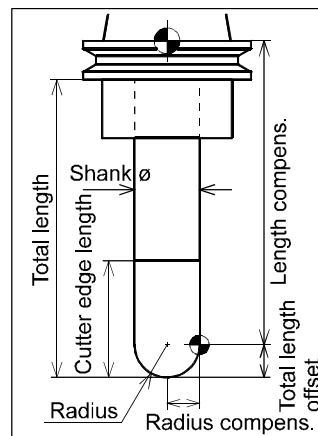
Shell end mill



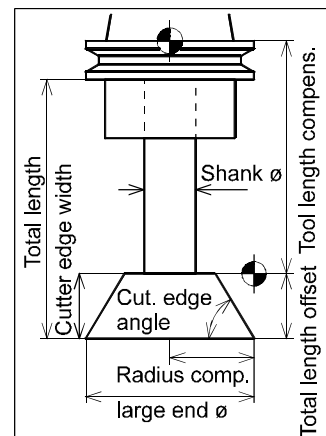
Face end cutter



Radius cutter



Angular cutter (type A)



## 5.2.1 Creating a New Tool

Each tool consists of tool geometry and of a tool adapter. Consequently, both data records are required to save a tool, however, the tool adapter can be managed separately as well.

A separate Data Entry menu is provided for each procedural step.

### Data Entry: Tool

Having started in the main menu by selecting the tool type, and subsequently selecting the Create Tool function by pressing **F1**, the Data Entry menu for defining the tool is loaded:

#### Data Entry Menu

The screen layout of the Data Entry menu is divided into three areas: the window on the left contains either a help graphic or a graphic corresponding to the data of the tool being defined (including the tool adapter). The input fields for the complete data record are located on the right.

The function keys at the bottom of the screen are labeled with their respective functions:

**F1**

**Tool Graphic:** If the data entry of all parameters is completed, you can obtain a graphic display of the tool with this function in the window area of the screen.

**F2**

**Help Graphic:** This function displays a help graphic in the window.

**F3**

**Select Tool Adapter:** Use this function to select an existing tool adapter for the tool (see below).

**F5**

**Clear Entry Data:** Press **F5** to delete the tool geometry data.

**F6**

**Rotational Direction:** Use **F6** to specify right-hand or left-hand (M03/M04) rotation.

**F7**

**Copy Data:** Use **F7** to access the selection menu for copying an existing data record as a default.

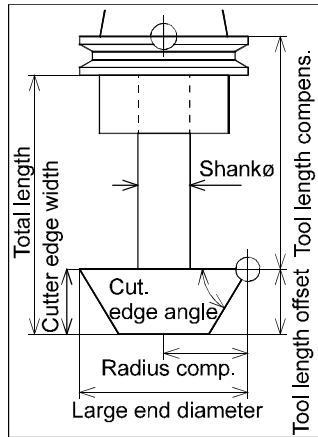
**F8**

**Create Tool:** When the data entry for all parameters of tool and tool adapter has been completed, you can save the tool under a certain name by pressing **F8**.

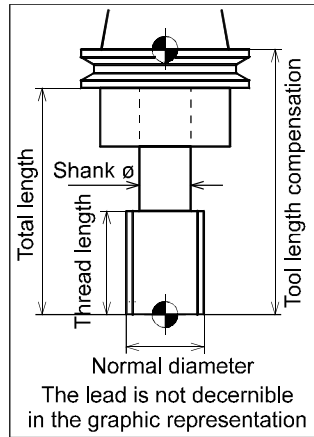


To save time, it is reasonable to define a new tool by first copying the data record of a similar tool, and then to modify the data to meet your requirements.

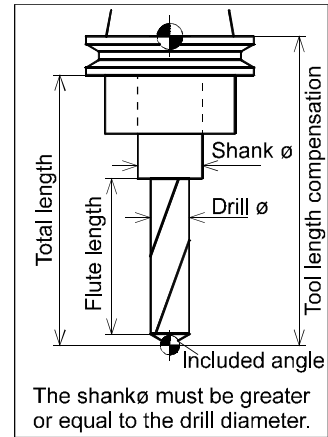
Angular cutter (type B)



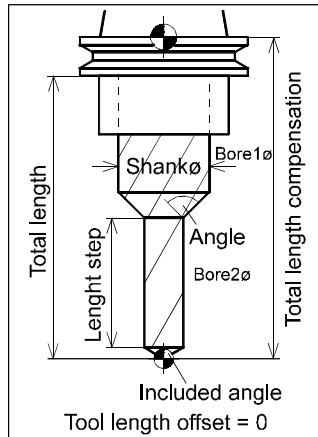
Tap drill



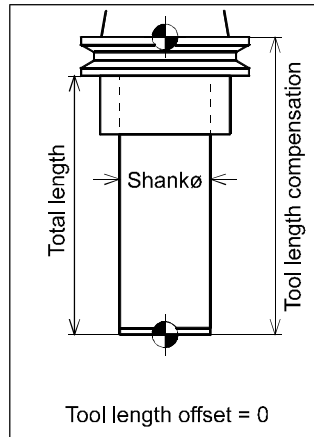
Drill



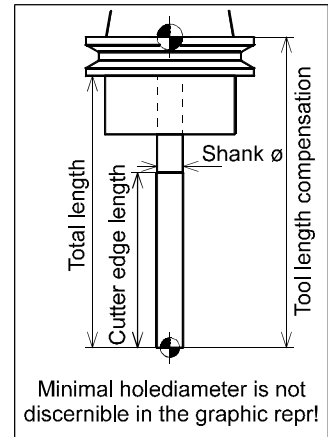
Step drill



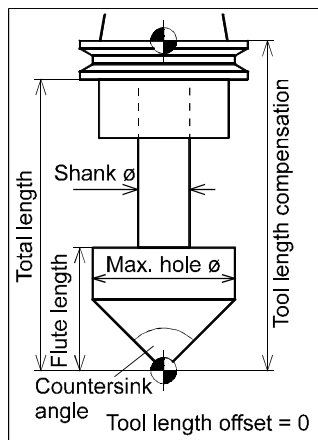
Reversible tip drill



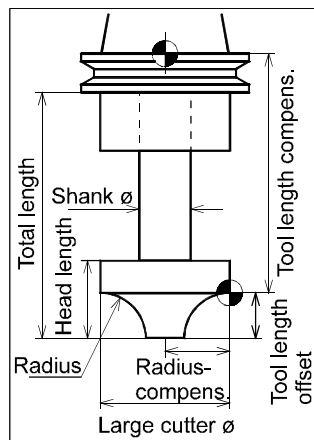
Reamer



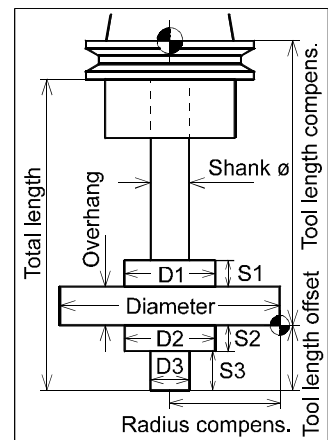
Countersink



Concave cutter



Side milling cutter





**Procedure**

You define a tool by manually entering the geometrical data, as well as the tool name and rotation direction. The desired tool adapter data can be automatically copied by selecting the Select Tool Adapter function.



Use the key to move from input field to input field.



or



Use the or cursor keys to move the cursor within the input field.



or



Use the key to insert a character, and the key to delete one.



If you confirm the entry in the input field with the key, the cursor moves automatically to the next input field.

[Tool Name]

Enter the tool name or number in this input field.

[Parameter]

The entries required for a tool depend on the tool type. Use the help graphics to obtain information on the parameters (see also the Appendix to the Programming Instructions).



**Rotation direction:** Use to specify either right-hand or left-hand rotation (RH/LH).



**Create tool:** When the data entry for all tool and tool adapter parameters has been completed, you save the tool under a certain name by pressing .



Use to conclude the operation, and to return to the Define/Delete Tools main menu.



Before adding the tools used in your actual production to the tool files of the CNC Simulator, be sure to get sufficient training in the procedures required in using the Define/Delete Tools menu.

Figure 5.2.1-2:  
CNC Milling,  
Define/Delete Tools;  
selecting a specific  
tool from the tool list.

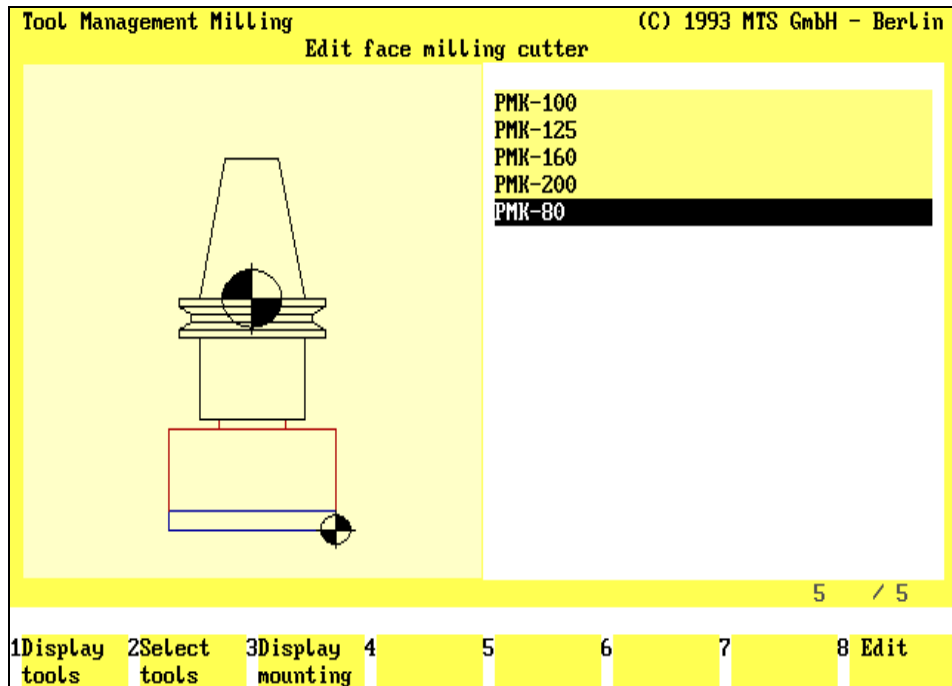
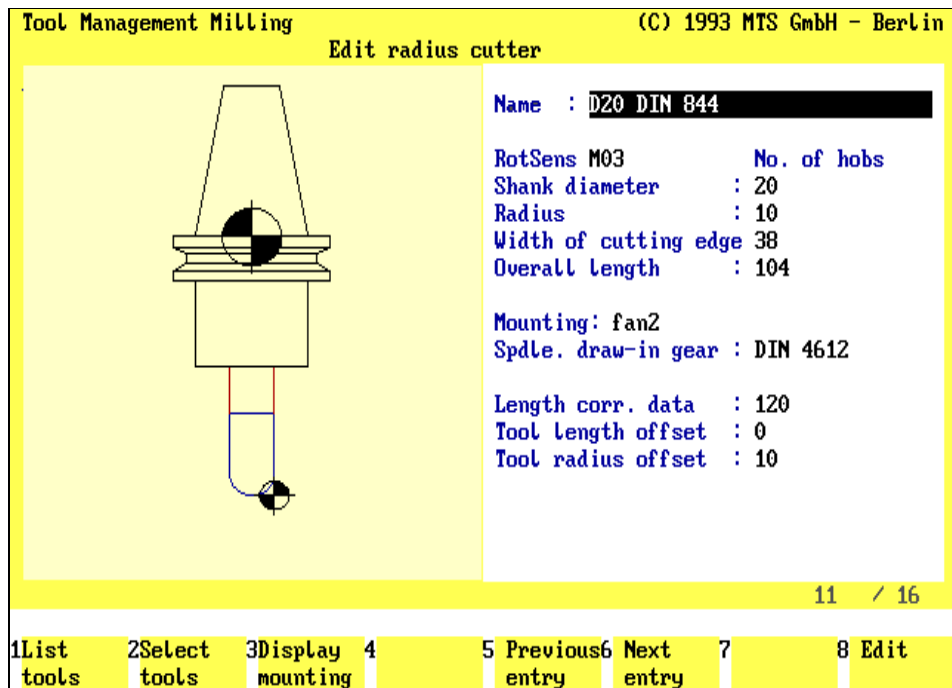


Figure 5.2.1-3:  
CNC Milling,  
Define/Delete Tools;  
selecting a specific  
tool from the listing of  
individual tools.



## Copying Existing Tool Data

If you want to use the data of an existing tool as a basis for a new tool, you can read in the data with the Copy Data function. This function is used to search for a specific data record to be used as a 'temporary presetting', to be then modified with the Data Entry menu to meet your requirements:

**F7**

Tool List Selection

**Copy data:** The Copy Data function is accessed for tool geometry from the Data Entry menu by pressing **F7**.

An intermediate Tool Selection menu is then available. It is divided into three sections (like the preceding example): the left part of the screen contains a window displaying the tool; a complete listing of all available tools of the selected tool type appears on the right. The current tool is highlighted in color, and the function keys at the bottom of the screen show available further functions:

**F1**

**Show Tools:** Use **F1** to switch to the graphic display of the current tool (see also the section Select Single Tool below).

**F2**

**Search & List Tools:** To select from the total quantity of the available tools of a certain type those meeting the geometrical criteria, you can have these searched and then displayed as a new list.

**F3**

**Show Tool Adapter:** A graphic display of the currently marked tool in the list is shown in the window on the left. Use **F3** to obtain a separate display of the tool adapter.

**F8**

**Create Temporary Tool:** Use **F8** or **↵** to copy the data of the currently selected tool as a temporary tool definition into the entry menu.

### Procedure

In this example, we assume that the list of the available tools is not too extensive, and that are immediately able to identify the tool whose data record should be copied as a temporary data set for the entry menu:

**↑**

or

**↓**

Use the **↑** or **↓** cursor keys to mark the desired tool.

**F8**

**Copy Data:** Use **F8** to copy the data record of the current tool into the entry menu, to serve as a temporary tool definition.

**ESC**

Use **ESC** to cancel the selection, and to return to the Data Entry menu.

Alternatively, you may wish to check the tool data. In this case, you press **F1** to access the selection menu displaying a single tool:

Select Single Tool

Similar to the entry menu, the second selection menu shows the currently selected tool on the left hand side of the screen. All tool data appear on the right, and the function keys at the bottom of the screen are marked with corresponding functions. In addition to the above mentioned functions, this menu contains the following options:

**F5**

or

**F6**

**Scroll forward/backward:** To view the preceding or following tool in the current tool list, scroll forward or backward by pressing the **F5** or **F6** function keys.

Figure 5.2.1-4:  
CNC Milling,  
Define/Delete Tools;  
Search & List function  
in the tool list;  
schematic of the menu  
sequence.

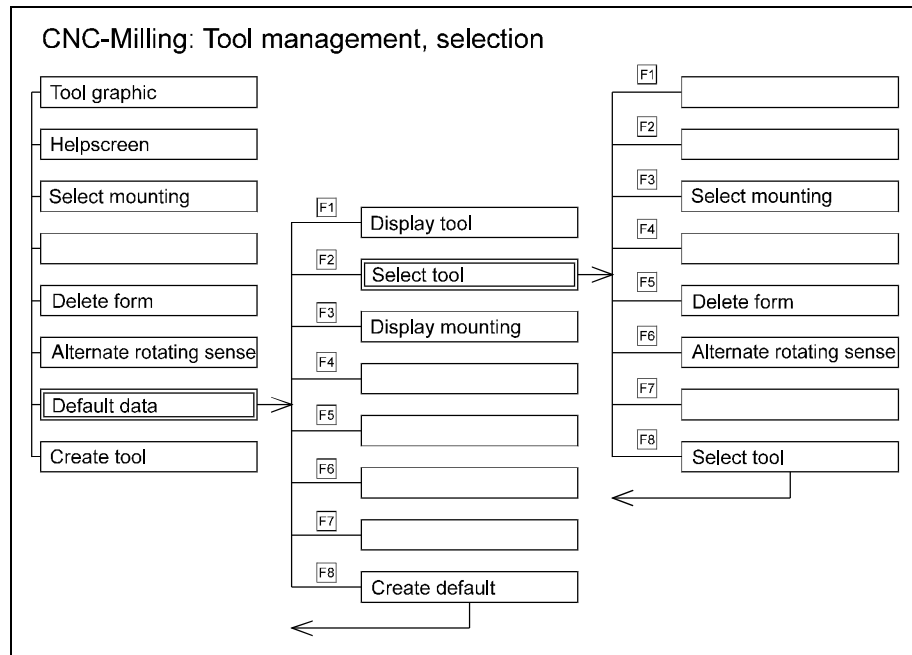


Figure 5.2.1-5:  
CNC Milling,  
Define/Delete Tools;  
Search & List menu:  
searching the tool list.  
This example uses a  
cutting edge angle of  
45° and a RH rotation  
(M03) as search  
criteria.

**Tool Management Milling** (C) 1993 MTS GmbH - Berlin

**Select angular cutter (type A)**

**Name :**

RotSens M03      No. of hobs

Shank diameter    :

diam. at large end :

Width of cutting edge

Cutting edge angle : 45

Overall length    :

**Mounting:**

Spdle. draw-in gear :

**Length corr. data :**

Tool length offset :

Tool radius offset :

1
2
3 Select mounting
4
5 Delete form
6 Alt. rot. sense
7
8 Select tools

## Available Tools List: Search & List Function

As indicated, this function is used to search in the file of available tools for tools meeting certain criteria, and to produce a new tool list. The entries in the data fields are used as selection criteria. After completing the selection, one tool can be selected as a temporary tool definition basis for the Data Entry menu.

**F2**

Selection Menu

**Select Tools:** From both selection menus, the Search & List function is accessed by pressing **F2**.

This opens a Selection menu divided into three sections (similar to the Data Entry menu): the left part of the screen is a window displaying the help graphic of the entry parameters; the input fields are on the right, and the function keys at the bottom of the screen are marked with the corresponding functions:

**F3**

**Select Tool Adapter:** Use this function to select an existing tool adapter for the tool.

**F5**

**Clear Entry Data:** Use **F5** to clear the entries in the input field.

**F6**

**Rotation:** Use **F6** to specify the rotation direction (LH or RH turning).

**F8**

**Search & List Tools:** Use **F8** to select in the file of available tools the tools meeting your selection criteria.



The size of the tool list is indicated by the X/Y values in the information line on the right: Y indicating the total number of tools, and X indicating the position of the selected tool in the list. For example, the statement "4/40" means that the list has 40 entries, and that the selected tool is the 4th entry in the list.

### Procedure

Let us assume, you have several geometrical values to enter as search criteria:

**TAB**

Use the **TAB** key to move from input field to input field.

[Enter Values]

Enter the desired geometrical values in the input fields.

**F8**

**Search & List:** Use **F8** to select from the available tools list the tools meeting your search criteria. You subsequently return to the Selection menu to see a reduced tool list.

**ESC**

Use **ESC** to cancel the search process.

### Note

Should there be no tool to meet the search criteria, an error message is displayed, "No tool available!" You can then enter new search criteria in the Data Entry menu.

If you wish to have a listing of all available tools displayed, use the Search & List function without entering any criteria in the input fields.

Figure 5.2.2-1:  
CNC Milling,  
Define/Delete Tools;  
Modify Tool menu  
sequence schematic.

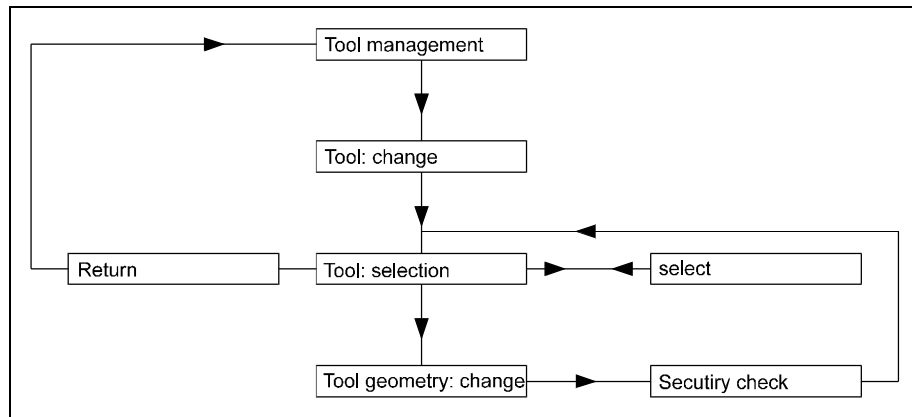


Figure 5.2.2-2:  
CNC Milling,  
Define/Delete Tools;  
selecting a specific  
tool with the Modify  
Tool function.

**Tool Management Milling** (C) 1993 MTS GmbH - Berlin  
**Select angular cutter (type A)**

**Name :**  
 RotSens M03                      No. of hobs  
 Shank diameter                    :  
 diam. at large end                :  
 Width of cutting edge            :  
 Cutting edge angle                : 45  
 Overall length                    :

**Mounting:**  
 Spdle. draw-in gear            :

**Length corr. data :**  
 Tool length offset                :  
 Tool radius offset                :

1	2	3 Select mounting	4	5Delete form	6Alt. rot.7 sense	8Select tools
---	---	-------------------	---	--------------	-------------------	---------------

## 5.2.2 Modifying a Tool

The same steps as were used to create a new tool are used to modify an existing tool from the available tools file. Before modifying a data record to meet your requirements, the tool to be modified needs to be selected (see the [Copy Tool Data](#) function described earlier in this chapter).

### Procedure

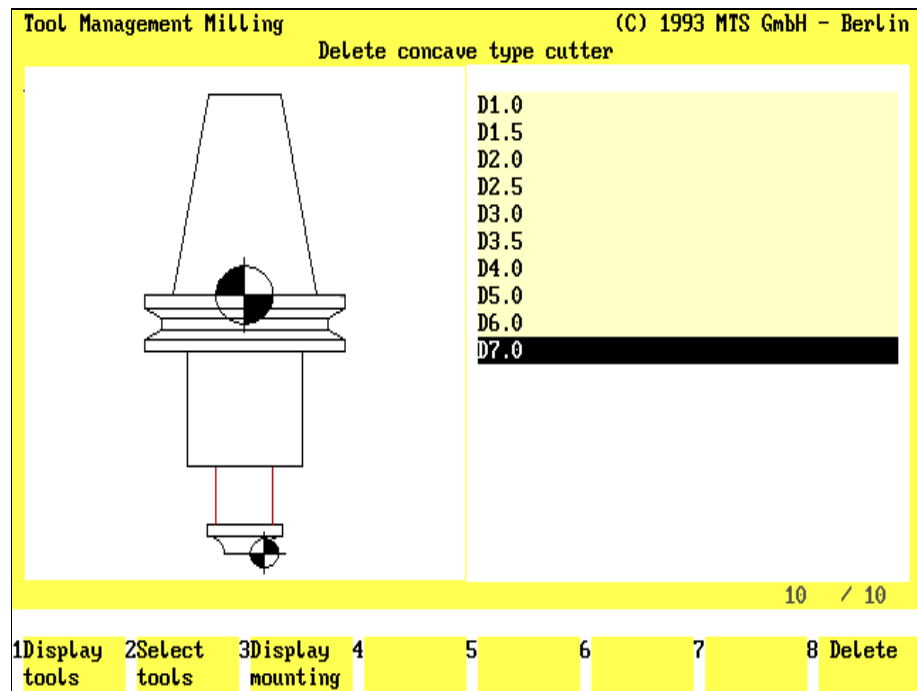
The menus used to modify an existing tool have been introduced in the previous passages. To avoid repetition, these menus are not described here in detail, only the sequence of the steps required is discussed below:

- 1.) Starting from the Define/Delete Tools main menu, select the Modify Milling Tool function by pressing **[F2]**.
- 2.) Subsequently, when copying an existing data record, the selection menu is loaded, with the following available options
  - select the desired milling tool and,
  - if applicable, first search the file of available tools, or switch to the display of individual tools.
- 3.) Subsequently, the program accesses the data entry menu and, with the exception of the tool name, all entries can be modified as required.
- 4.) Press **[F8]** to conclude the operation, and to save your modifications. The selection menu is then reopened for possible further modifications, or you can use **[ESC]** to return to the Define/Delete Tools main menu.



**Please note that modifications of a tool may also necessitate changes in existing setup sheets or NC programs using this tool. It is therefore recommended to change the tool file only when necessary. When in doubt, (rather than changing an existing one), generate a new tool!**

Figure 5.2.3-1:  
CNC Milling,  
Define/Delete Tools;  
selecting a specific  
tool with the Delete  
function.





### 5.2.3 Deleting a Tool

Deleting an existing tool from the file of available tools is even simpler than modifying: you first select the tool to be deleted, and then you delete it.

#### Procedure

The menu sequence for selecting the desired tool is identical with the previously described operation. To avoid repetition, only the sequence of required steps is explained:

- 1.) Starting from the Define/Delete Tools main menu, select the Delete Milling Tool function by pressing **F3**.
- 2.) After using the Search & List function to select the file to be deleted, mark the file for deletion in the list window.

Once you have confirmed your selection, the Delete function is activated, and for control reasons the warning message is displayed, "**Do you want to delete the tool?**"

**F7**

or

**ESC**

**No/Cancel:** Use **F7** or **ESC** to cancel the operation.

**F8**

or



**Yes/Delete:** Confirming the deletion by pressing **F8** or  deletes the tool. Subsequently, you return to the Define/Delete Tools main menu.



**Please note that deleting a tool may also necessitate changes in existing setup sheets or NC programs using this tool.**

**It is therefore recommended not to make any deletions from the tool file unless necessary. If possible, the default tools included in the program should be kept intact.**

Figure 5.2.4-1:  
CNC Milling,  
Define/Delete Tools;  
Defining a tool  
adapter.

**Tool Management Milling** (C) 1993 MTS GmbH - Berlin  
Select the tool mounting

**Name :** \_

**Spdle. draw-in gear:**  
**Short taper** Diameter :  
 Length :  
 Angle :

**Engag. slot:**  
 Diameter Length:  
 Depth : Width:  
 Angle :

**Chuck holder:**  
 Diameter Length:

**Min. Capac.:**  
 Max:

1 2 3 4 5Delete 6 7 8Select no  
 form untings

## 5.2.4 Creating a New Tool Adapter

The tool management functions include the definition of different tool adapters specified according to the DIN or other industrial standards. For data entry or modification, separate menus to be accessed from the Define/Delete Tools main menu are available. In this operation it does not make any difference which tool type has been selected.

### Note

Please keep in mind that the data record defining a tool consists of the tool geometry data, as well as of the tool adapter data. Assigning a tool adapter to a specific tool is done as a part of the tool definition.

### Data Entry: Tool Adapter

Starting in the Define/Delete Tools main menu, you access the Define Tool Adapter function by pressing **F5**. The Data Entry menu is then loaded:

Data Entry: Tool Adapter

The screen layout of the Data Entry menu is divided into three main areas: the window on the left contains either a help graphic or a graphic corresponding to the data of the tool adapter being defined. The input fields for the complete data record are located on the right.

The function keys at the bottom of the screen are labeled with their respective functions:

**F1**

**Tool Adapter Graphic:** When the data entry of all parameters has been completed, pressing **F3** enables you to obtain a graphic display of the tool adapter in the window.

**F2**

**Help Graphic:** The help graphic describes the input parameters.

**F5**

**Clear Entry Data:** Press **F5** to delete all data entries from the input fields.

**F7**

**Copy Data:** Use **F7** to copy an existing data record into the input fields.

**F8**

**Create Tool Adapter:** When the data entry for all parameters has been completed, you can save the tool under a certain name by pressing **F8**.

### Procedure

The steps required to define a tool adapter are similar to those used in tool definition.

**TAB**

Use the **TAB** key to move from input field to input field.

**→**

or

**←**

Use the **→** or **←** cursor keys to move the cursor within an input field.

**INS**

or

**DEL**

Use the **INS** key to insert a character, and the **DEL** key to delete one.

**↵**

When confirming an entry in an input field with the **↵** key, the cursor moves automatically to the next input field.

[Tool Name]

Enter the name or the number for the tool adapter in this input field.

[Parameter]

The entries required for a tool adapter depend on its type. Use the help graphics for information on the parameters.

**ESC**

**Use **ESC** to cancel the operation, and to return to the Define/Delete Tools main menu.**

**F8**

**Create Tool Adapter:** Use **F8** to save the tool adapter.

Figure 5.2.4-2:  
CNC Milling,  
Define/Delete Tools;  
Selecting a tool  
adapter from the list.

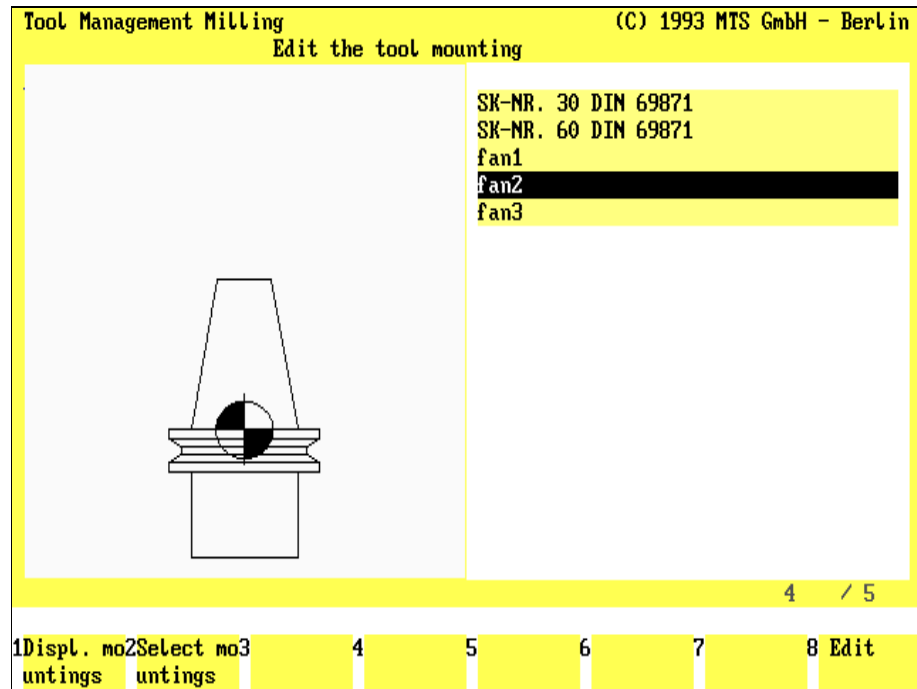
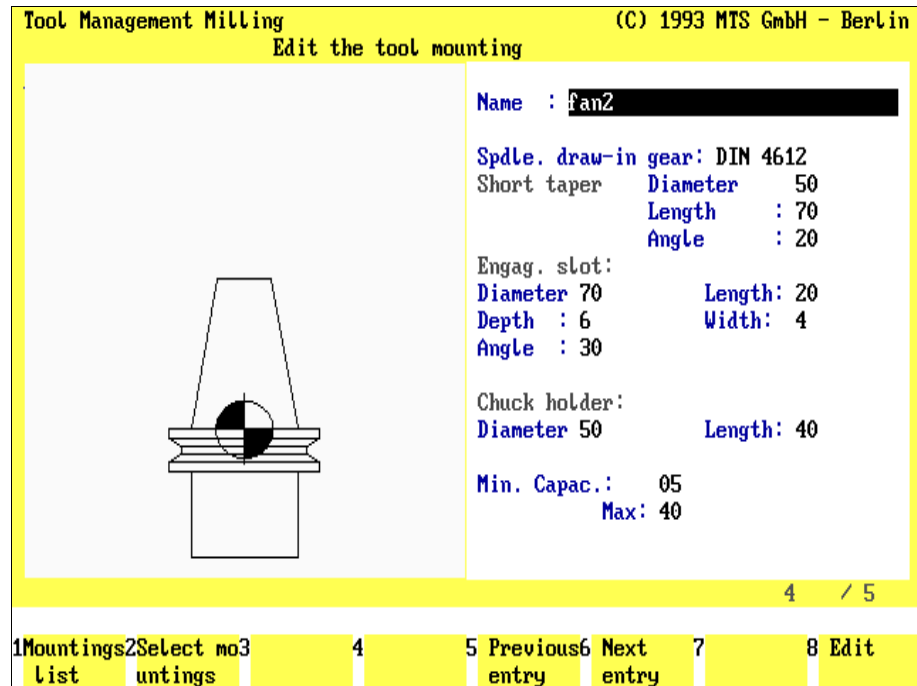


Figure 5.2.4-3:  
CNC Milling,  
Define/Delete Tools;  
Selecting a tool  
adapter from the  
individual display.



## Copying Existing Tool Adapter Data

If you want to use the data of an existing tool as a basis for a new tool, you can make use of the Copy Data function. You use this function to search for a specific data record to be used as a 'temporary specification', and then use the Data Entry menu to modify the existing values to meet your requirements:

**F7**

Search & List

**Copy data:** The Copy Data function is accessed from the Data Entry menu by pressing **F7**.

This opens a selection menu divided into three sections (as in the previous example): the left part of the screen contains a window displaying a graphic of the current tool adapter; a complete listing of all available tool adapters appears on the right. The current adapter is highlighted in color, and the function keys at the bottom of the screen are marked with their respective functions:

**F1**

**Show Tool Adapter:** Use **F1** to switch to the detailed listing of the current fixture (see also the section Select Single Tool Adapter below).

**F2**

**Search & List:** This function is used to select specific data records from a large file. In the subsequent menu you enter the criteria the tool adapters to be selected need to meet; this search will generate a new list (see below).

**F8**

**Create Tool Adapter:** Use **F8** or **↵** to copy the data of the currently selected tool adapter as a temporary definition into the entry menu.

### Procedure

In this example, we assume that the list of available tools is not too extensive, and that you are able to identify the tool adapter to be copied as a temporary data set for the entry menu:

**↑**

or

**↓**

Use the **↑** or **↓** cursor keys to select the desired tool adapter.

**F8**

**Copy data:** Use **F8** to copy the data record of the current tool adapter into the entry menu, to serve as a temporary definition.

**ESC**

Use **ESC** to cancel the selection, and to return to the Data Entry menu.

Select Single Tool

Alternatively, you may want to check the tool adapter data for control reasons. In this case, you press **F1** to access the selection menu, to display one single tool adapter:

Similar to the entry menu, the second selection menu shows the currently selected tool adapter on the left hand side of the screen. The data record is displayed on the right, and the function keys at the bottom of the screen are marked with the respective functions:

**F1**

**List Tool Adapters:** Press **F1** to return to the listing of tool adapters.

**F5**

or

**F6**

**Scroll Forward/Backward:** To view the preceding or the next tool adapter in the current list, scroll forward or backward by pressing the **F5** or **F6** function keys.

Figure 5.2.4-4:  
CNC Milling,  
Define/Delete Tools;  
Search & List function  
on list of available tool  
adapters; Menu  
sequence schematic.

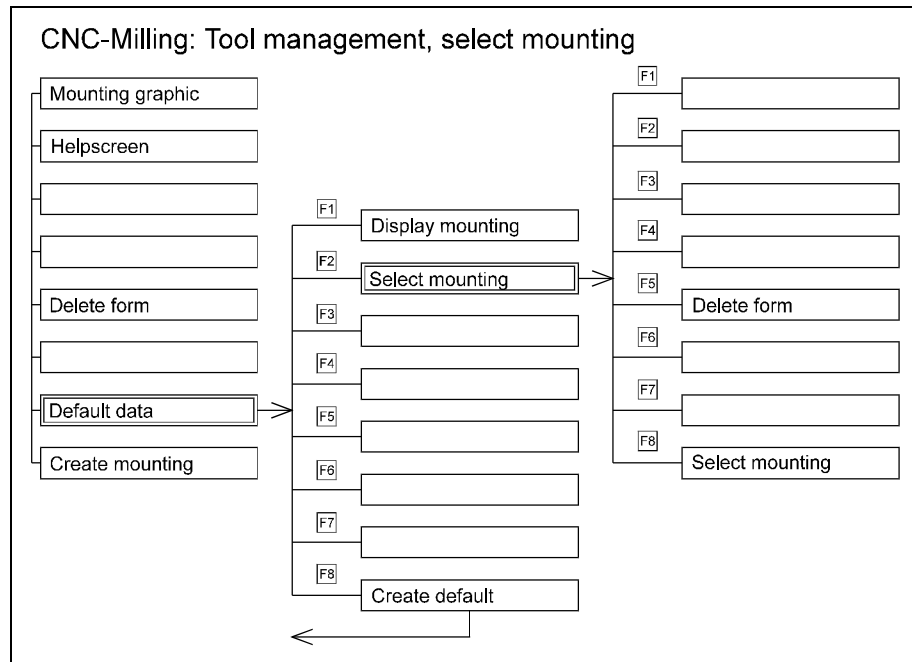


Figure 5.2.4-5:  
CNC Milling,  
Define/Delete Tools;  
Search & List menu for  
tool adapters.  
In this example, the  
minimum and  
maximum width for the  
opening were specified  
as 10 and 20 as search  
and selection criteria.

**Tool Management Milling** (C) 1993 MTS GmbH - Berlin  
Select the tool mounting

**Name :**

**Spdle. draw-in gear:**  
Short taper    **Diameter**  
                  **Length** :  
                  **Angle** :

**Engag. slot:**  
**Diameter**            **Length:**  
**Depth** :            **Width:**  
**Angle** :

**Chuck holder:**  
**Diameter**            **Length:**

**Min. Capac.:** 10  
**Max:** 20\_

1    2    3    4    5Delete form    6    7    8Select mountings

## Available Tool Adapters: Search & List

As already described, this function is used to search a large file for specific tool adapters meeting certain criteria, and to generate a new tool list. The entries in the data fields are used as selection criteria.

**F2**

Search & List  
Tool Adapters

**Search & List [Tool Adapters]:** From both selection menus, the Search & List function is accessed by pressing **F2**.

This opens a Selection menu which is divided into three sections (similar to the Data Entry menu): the left part of the screen contains a window displaying the help graphic for the entry parameters; on the right, you enter the desired selection criteria in the input fields, and finally, the function keys at the bottom of the screen are marked with the corresponding functions:

**F5**

**Clear Entry Data:** Use **F5** to clear all entry data from the input fields.

**F6**

**Search & List Tool Adapters:** Use **F6** to select those tool adapters from the current file that meet your selection criteria.

### Procedure

Let us assume that you have several geometrical values to be entered as search criteria:

**TAB**

Use the **TAB** key to move from input field to input field.

[Enter Values]

Enter the desired geometrical values in the input fields.

**F8**

**Search & List:** Use **F8** to select from the file the tool adapters meeting your search criteria. You then automatically return to the selection menu.

The size of the tool list is indicated by the X/Y value on the right above the information bar, X indicating the location of the current tool adapter in the list, and Y indicating the total number of items in the list.

**ESC**

Use **ESC** to cancel the search process.



**If you do not know the search/selection criteria to be entered, leave the input fields without entries and use the Search & List function. This causes all available tool adapters to be included in the resulting list.**

Figure 5.2.5-1:  
CNC Milling,  
Define/Delete Tools;  
Modify Tool Adapter;  
Menu sequence  
schematic.

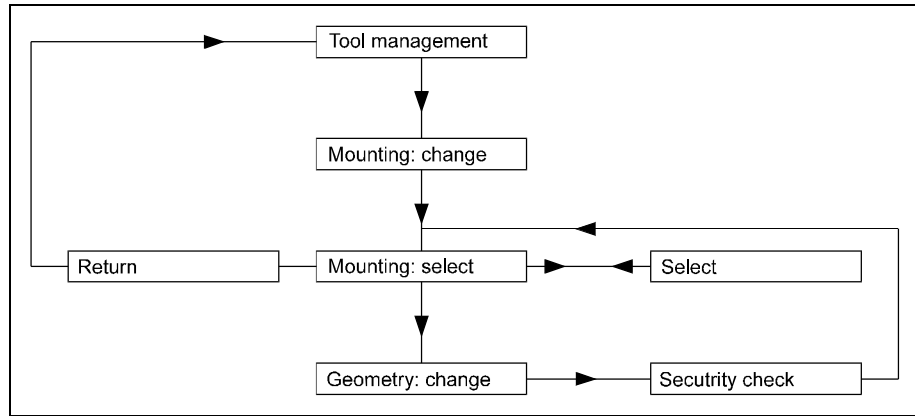
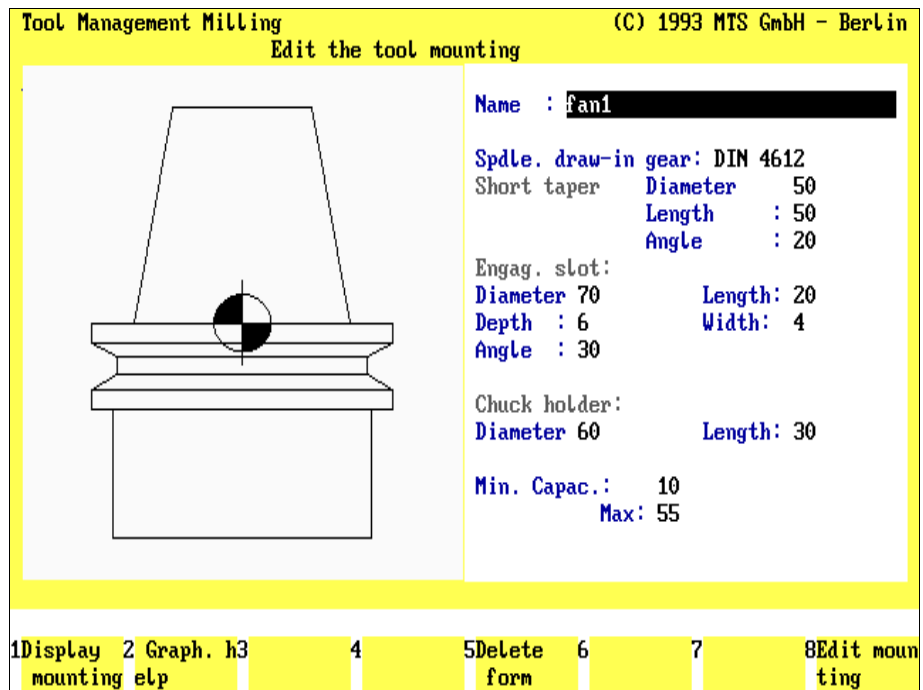


Figure 5.2.5-2:  
CNC Milling,  
Define/Delete Tools;  
using the Modify Tool  
Adapter function to  
select a specific  
adapter.





## 5.2.5 Modifying a Tool Adapter

To modify an existing tool adapter of the available tools file is identical to creating a new tool adapter. Before modifying the existing data record to meet your requirements, the tool needs to be selected first.

### Procedure

The menus used to modify an existing tool adapter have been introduced in the previous passages. To avoid repetition, these menus are not discussed here in detail. Only the sequence of the necessary steps is discussed below:

- 1.) Starting from the Define/Delete Tools main menu, select the Modify Tool Adapter function by pressing **[F6]**.
- 2.) Subsequently, as was the case when reading in an existing data record (Copy Data), the selection menu is loaded, with the options to
  - select the desired tool adapter and,
  - if applicable, first search the data file, or switch to the display of individual tool adapters.
- 3.) Subsequently, the program accesses the data entry menu, and you can modify all entries as required.
- 4.) Press **[F8]** to conclude the operation, and to save your modifications. This causes the selection menu to reopen for eventual further modifications, or you can use **[ESC]** to return to the Define/Delete Tools main menu.

## 5.2.6 Deleting a Tool Adapter

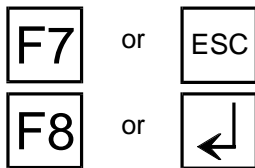
To delete an existing tool adapter is even simpler than modifying it: you first select the tool adapter to be deleted, and then you delete it!

### Procedure

The menu sequence for selecting the desired tool adapter is identical to the previously described operation. To avoid repetition, only the order of the required steps is discussed here:

- 1.) Starting from the Define/Delete Tools main menu, select the Delete Tool Adapter function by pressing **[F7]**.
- 2.) After using the Search & List function, as previously described, to select the file to be deleted mark the tool adapter in the list window.

Once you have confirmed your selection, the Delete function is activated, and for control reasons the control enquiry is displayed, "**Do you want to delete the tool adapter?**"



**No/Cancel:** Use **[F7]** or **[ESC]** to cancel the operation.

**Yes/Delete:** Confirming the deletion by pressing **[F8]** or **[↵]** deletes the tool. Subsequently, you return to the Define/Delete Tools main menu.



**Please note that modifying a tool adapter may also necessitate changes in existing setup sheets or NC programs. It is therefore recommended not to modify a tool adapter unless necessary. If possible, the default tool adapters of the program should remain unmodified. When in doubt, generate a new data record for the tool!**



## 6. Automatic Mode

The automatic mode is used to run and **test** programs created by the CNC Simulator, the INCAD NC/CAD system or some other programming system, in real-time simulation under consideration of collision monitoring. Since the machine and control configuration of the simulator allows a reality-identical performance of the machine tool in terms of geometrical and technological parameters as well as those of the CNC control, the program evaluation takes place under conditions highly identical with the actual workpart machining operations. Accordingly, various configurations provide professional applications for the purpose of **optimizing** the NC programs.

It is also convenient for the user to be able to change directly from the automatic mode to the Interactive Programming mode (and subsequently to the editor, the Workshop-Oriented Programming interface or the Teach-In functions) without having to interrupt the program. Since the Programming Aids are available as well, error corrections and program improvements can be carried out quickly and conveniently.

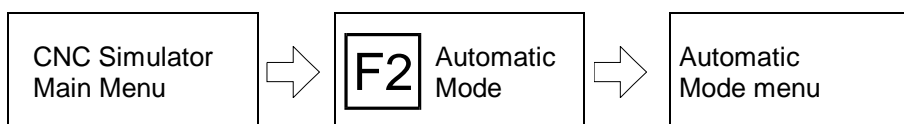
Depending on your needs and operating habits you may wish to make use of the full performance potential of the CNC Simulator. You have the choice between different simulation modes to be made as a flying change, as well as the option to add certain supplementary functions, such as zooming (CNC turning), measuring, 3D-view, traverse path monitoring, and the calculation of machining cycles and downtimes.

### Note

The conditions for the Automatic Mode are the configuration defaults and the predefined machine setup.

Since the simulations can be run with different tool qualities and workpart materials, etc., the programmed feed rates and rpm values are **not** subject to error and collision monitoring. It is recommended to have them checked individually, before transferring an NC program to a CNC control system.

### Selection:



Starting from the Main Menu, Automatic Mode is accessed by pressing the **F2** function key.

### Setup

Please consider, before starting the Automatic Mode, the CNC Simulator needs to be properly set up (i.e., going to the reference point, defining turret and magazine configuration, saving tool compensation values, etc., see also [Chapter 4](#)). Alternatively, you may wish to have the Automatic Mode set up automatically.

### Setup Sheet

Apart from manually setting up the machine tool, you have the option of performing an automatic setup by using a setup sheet. The corresponding data is introduced to the respective NC program, and can be generated -- also automatically -- according to a specific machine status. **When a setup sheet has been implemented in the program, and the interpreter is switched on, the machine tool is set up according to the setup sheet data upon starting the NC program.**

Figure 6.-1:  
CNC Turning & Milling,  
Automatic Mode menu  
schematic.

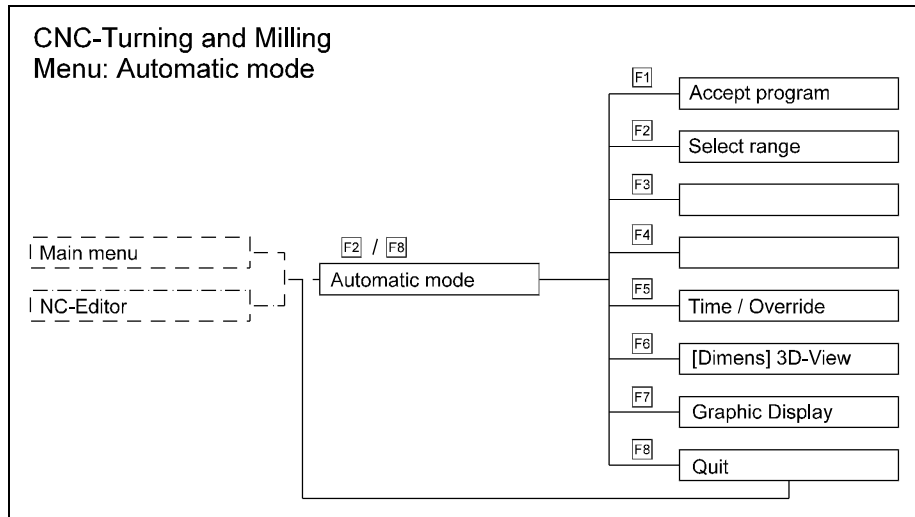


Figure 6.-2:  
CNC Turning,  
Automatic  
Mode menu.

Figure 6.-3:  
CNC milling,  
Automatic Mode;  
Selecting the  
simulation mode.

## Automatic Mode Menu Structure

A status message in the information column ("**Automatic Mode**") as well as the system information „modal commands“ and „runtime“ indicate that Automatic Mode has been activated. Further, to start an NC program in Automatic Mode, two procedural steps are required, both of which are supported by menus:

### Main Menu

- 1.) As a first step after starting Automatic Mode, the **main menu** is loaded, allowing you to enter the **name of the NC program** to be simulated.

Other menu functions:

**F1**

**Accept program:** Use **F1** or **↵** to confirm the program name appearing in the information line.

If the program is available, it is subsequently loaded into program memory; if not, an appropriate error message is displayed.

**F2**

**Select range:** If only a part of the NC program is to be run, select the range definition option by pressing **F2**.

**F5**

**Override/times:** To modify the feed rate override, start the test run or evaluate the tool engagement times, select **F5**.

**F6**

**[Measuring]/3D view:** Use **F6** to access the measuring function (available for CNC turning only) or the 3D View option.

**F7**

**Screen graphic:** The screen graphic function allows you to make certain modifications for the screen display.

**F8**

**Return:** Use **F8** to return to the main menu.

### Simulation Modes

- 2.) Continue by selecting the desired simulation mode for program execution:

**F1**

**Automatic:** Select **F1** to execute the program specified in the dialogue line under continuous automatic control.

**F2**

**Single NC block:** Use **F2** to activate single block operation.

**F3**

**Interactive:** To create or modify the NC program interactively, press **F3** to switch to Interactive Programming mode.

**F4**

**Traverse paths:** To track program execution by means of traverse path monitoring, press **F4**.

**The additional functions are identical to those discussed above.**

### Automatic Run

- 3.) Finally, during program simulation the following functions are available:

**F2**

**Single NC block:** Press **F2** to switch to single block operation.

**F4**

**Dynamic zoom:** During CNC Turning, the screen graphics can be enlarged and reduced by pressing **F4**.

**F8**

**Stop/resume:** Use the **F8** toggle function to interrupt the current simulation, and to continue after an interruption.

**The additional functions are identical to those discussed above.**

Figure 6.-4:  
CNC Turning,  
menu during  
continuous automatic  
run.

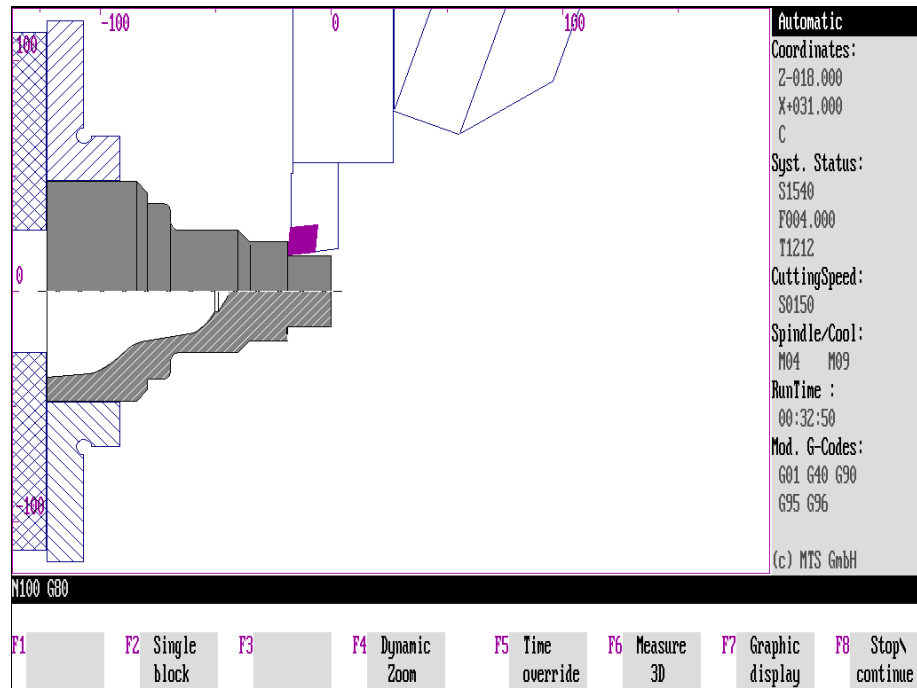
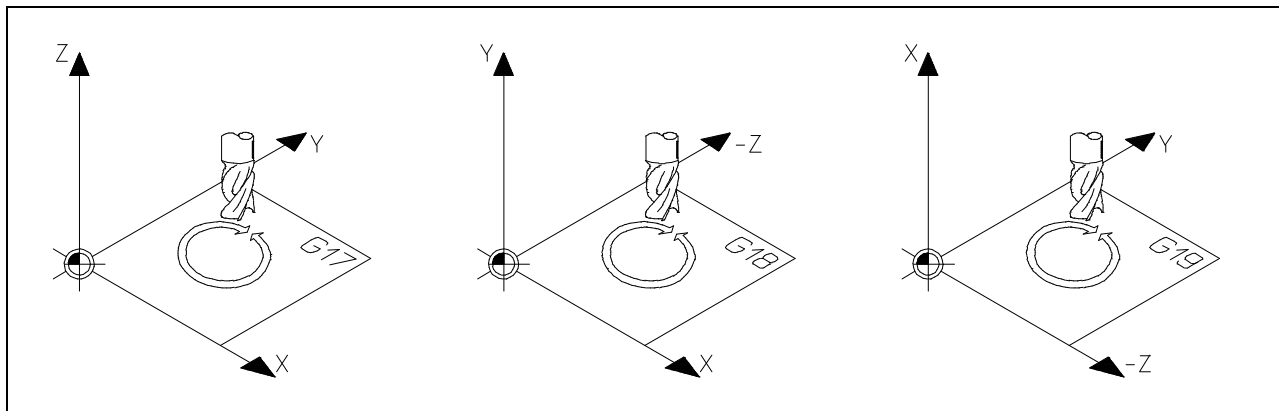
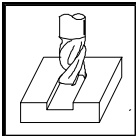


Figure 6.-5:  
CNC Milling;  
Presentation of a circular movement in the three machining planes, G17, G18, and G19:



Machining Plane:		longitudinal	lateral	vertical (feed)
Plane Selection 	<b>G17</b>	X	Y	Z
	<b>G18</b>	X	Z	- Y
	<b>G19</b>	Z	Y	- X

## Screen Layout

### Screen Window

The **screen layout** is similar to the screen in Setup Mode: the window on the left shows a section of the work space of the machine tool, including the blank or workpart, clamping device(s) and, depending on its position, the tool. During automatic run, the workpart milling operation is displayed by means of dynamic graphics. During milling operations the top view can be supplemented by cross section graphics of the workpart and tool, aligned on the coordinate axes. During turning operations, the workpart graphic can be enlarged or reduced (see also [Chapter 10](#)).

### Information Column

To supplement the workpart graphics during machining, current machine status information appears in the information column on the right:

Machine status	CNC Turning	CNC Milling
<b>Coordinates<sup>1</sup></b>	X / Z / C current tool position [in mm]	X / Y / Z current tool position [in mm] <sup>2</sup>
<b>System status</b>	S : Spindle rpm max <sup>3</sup> : 9000 rpm min <sup>3</sup> : 0010 rpm	S : Spindle rpm max <sup>3</sup> : 6000 rpm min <sup>3</sup> : 0010 rpm
	F : feed rate max <sup>3</sup> : 50.0 mm/U min <sup>3</sup> : .001 mm/U	F : feed rate max <sup>3</sup> : 999.0 mm/min min <sup>3</sup> : .001 mm/min
	Txxyy : working tool xx : turret location yy : comp. value reg.	Txxyy : working tool xx : magazine location yy : comp. value. reg.
<b>Cutting speed.</b>	in m/min	in m/min
<b>Spindle/Coolant</b>	M03 - M05 / M07 - M09	M03 - M05 / M07 - M09:
<b>Runtime</b> (incl. tool change)	machining time [min]:[sec]:[1/100 sec]	machining time [min]:[sec]:[1/100 sec]
<b>Modal Commands<sup>4</sup></b>	auto-stop G commands	auto-stop G commands
<b>Override</b>	10 - 150 %	10 - 150 %

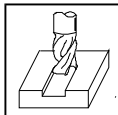
1 under consideration of tool compensation values and tool zero point; C is reserved for expansion with powered tools

2 consider also the plane selection

3 based on standard configuration

4 The movement commands of the NC program that remain valid until they are overwritten (see also Programming Instructions) are called "Modal commands".

### CNC Milling



Please consider the plane selection ("G17", "G18" or "G19") specified in the configuration; it determines the coordinate information ([see table on opposite page](#)).

Figure 6.-6:  
Automatic Mode;  
Systematic menu level  
schematic.

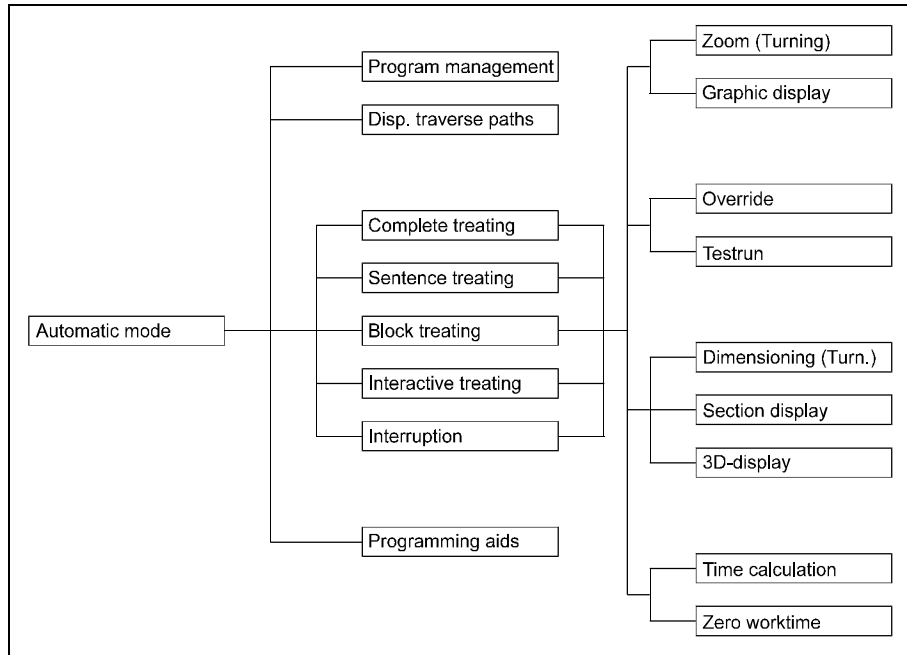


Figure 6.-7:  
CNC Milling;  
Example of an  
Automatic Mode  
menu.

**Automatic**

**Coordinates:**  
X-010.417  
Y+017.083  
Z-010.000

**Syst. Status:**  
S0500  
F120.000  
T1111

**CuttingSpeed:**  
S0063

**Spindle/Cool:**  
M03 M07

**RunTime :**  
17:25:57

**Mod. G-Codes:**  
G00 G40 G90  
G94 G97

(c) MTS GmbH

N200 G79 X+000.000 Y+000.000 Z+002.000

F1	F2 single block	F3	F4	F5 Time override	F6 3D-view	F7 Graphic display	F8 Stop continue
----	-----------------	----	----	------------------	------------	--------------------	------------------



**Dialogue Bars**

The dialogue lines at the bottom of the screen contain **information** on the current operation:

The upper line displays the **currently used NC program block**, the number of displayed NC blocks can be increased (see below).

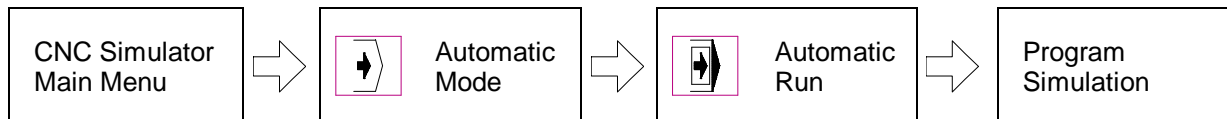
The bottom line is reserved for **error** and **collision messages** and for the **user dialogue**.

**Procedure**

As a whole, the operation of the Automatic Mode is structured to provide clear and convenient functions for the testing, modifying and optimizing of NC programs. For a better control, the comprehensive collision monitoring enables you to locate and correct program errors quickly.

**CNC Icons**

To keep the handling of the CNC Simulator as identical as possible with that of a machine tool, CNC icons can be displayed instead of the normal function keys; program operation is also possible by means of a CNC keyboard or a digitizer tablet with a CNC template:

**Selection via CNC Icons:****CNC Keyboard**

In some cases, certain program functions (i.e. setting Override, or Measuring/3D View) can be directly selected with CNC keyboard or digitizer tablet, without having to access the menus. You can choose the selection mode that is most convenient for you.

Figure 6.1-1:  
CNC Turning,  
Automatic Mode menu  
systematic sequence  
schematic.

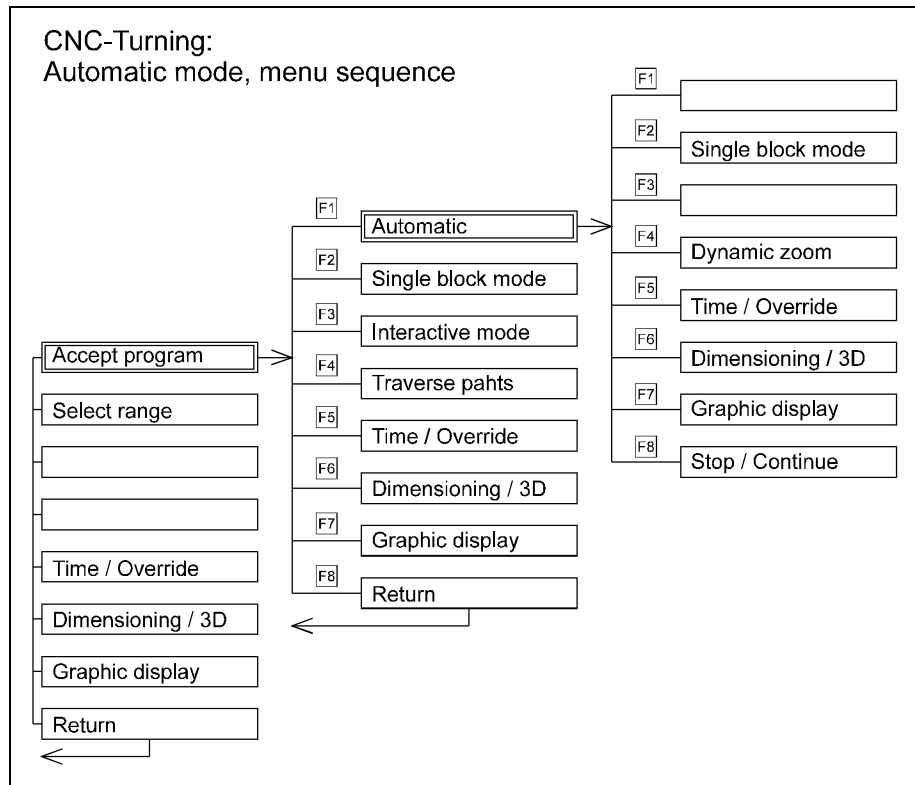


Figure 6.1-2:  
CNC Milling,  
Automatic Mode;  
Filename query of the  
specified NC program;  
in this example, the  
NC program name  
"%550" was entered.

Automatic  
Coordinates:  
X+040.000  
Y+116.417  
Z+160.000  
Syst. Status:  
S0500  
F120.000  
T0707  
CuttingSpeed:  
S0031  
Spindle/Cool:  
M03 M07  
RunTime :  
17:26:01  
Mod. G-Codes:  
G01 G40 G90  
G94 G97  
(c) MTS GmbH

Program name: %550  
Please enter program name and acknowledge

F1 Accept program   F2 Select range   F3   F4   F5 time override   F6 3D-view   F7 graphic display   F8 quit

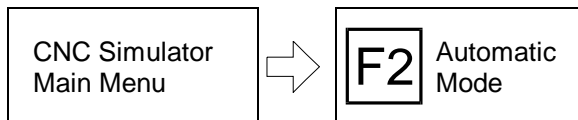
## 6.1 Starting Simulated Machining

Three procedure steps are required to start the machining simulation of an NC program:

- 1.) Starting from the CNC Simulator main menu, press **F2** to access Automatic Mode.
- 2.) Enter the name of the NC program to be executed during simulation.
- 3.) Select the simulation mode for the program execution.

The last two steps are supported by a program dialogue in the dialogue line.

### Selection:



### Note

Please check that the CNC Simulator has been set up correctly (i.e. referencing, defining tool compensation values, etc., see also related discussion in [Chapter 4](#)), prior to starting the Automatic Mode.

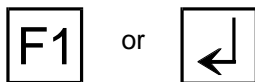
## Selecting NC Program

After the Automatic Mode has been started, you will be prompted for an entry, "Please enter and confirm/accept a program name." Simultaneously, the upper dialogue line is activated for this entry:

[Program Name]

Enter the name of the NC program that you wish to use.

If the program is stored in program memory (i.e., if it has been previously edited), its name appears automatically as a default in the upper dialogue line. If some other NC program is to be loaded, the existing program name can simply be overwritten.



**Accept program:** Use **F1** or **↵** to accept and confirm the program name shown in the dialogue line.

If this program is available, it is then loaded into program memory, if it is not, a corresponding error message is displayed.



**Select range:** If only a certain part of the NC program is to be executed, select the range definition by pressing **F2**.



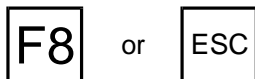
**Override/Times:** To modify the feed rate override, activate the test run, or evaluate engagement times, select **F5**.



**[Measuring]/3D view:** Use **F6** to activate the Measuring function (CNC turning only), or the 3D View option.



**Screen graphics:** To change the graphic display, select **F7**.



**Return:** Use **F8** or **ESC** to return to the main menu.

### Note

The selected NC program needs be located in the current subdirectory (with the default name, ".DPROG" or ".FPROG"). If the program is not available, an error message is displayed. When in doubt, use the Program Manager (see Chapter 4) to ascertain which directory is currently active, and which programmes are located in this directory.

Figure 6.1-3:  
CNC Milling,  
Automatic Mode;  
Selecting simulation  
mode.

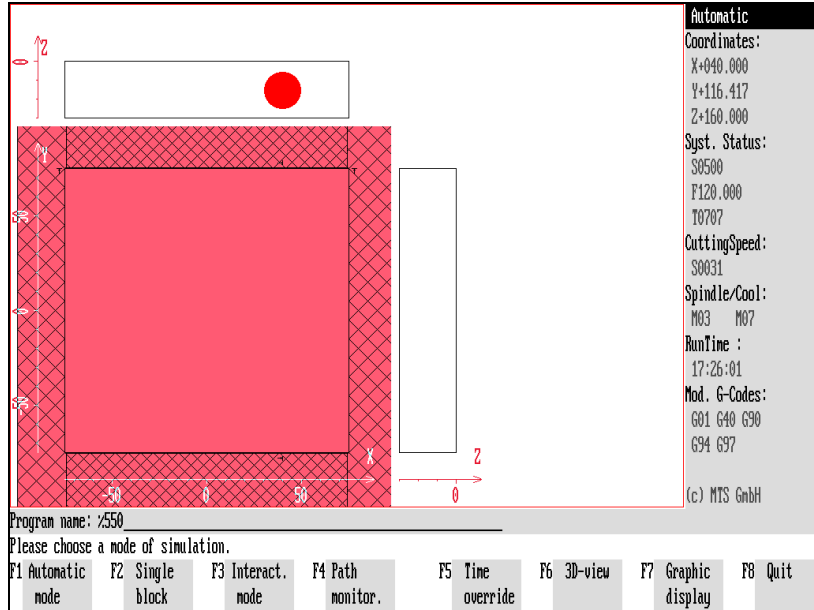


Figure 6.1-4:  
CNC Milling,  
Automatic Mode;  
Complete automatic  
run.

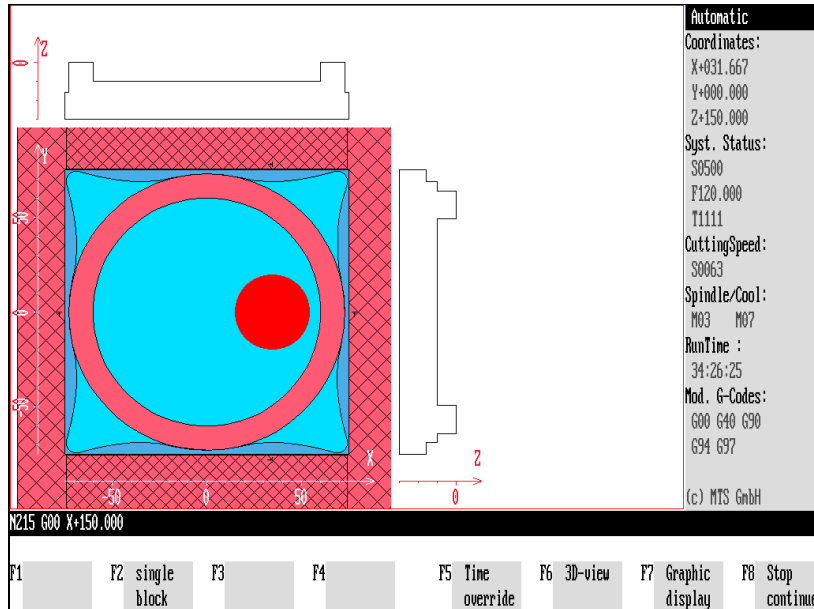
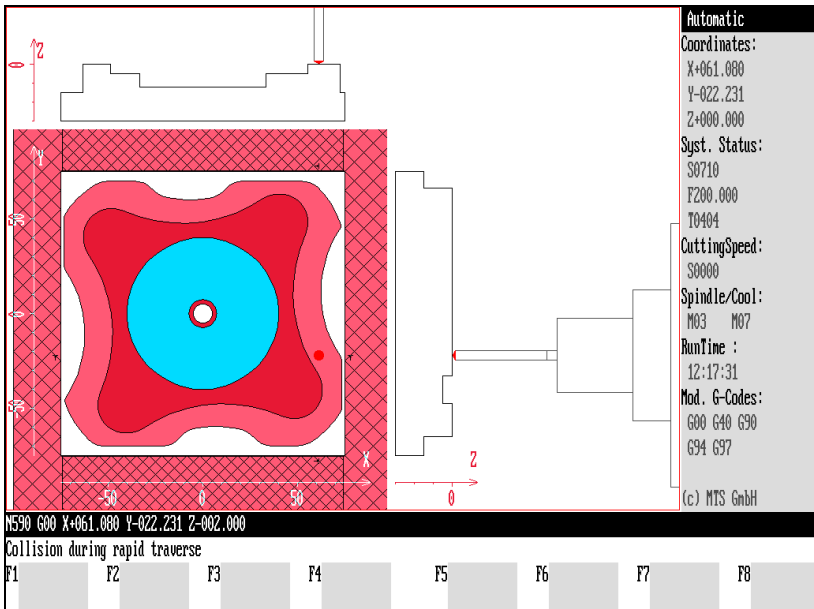


Figure 6.1-5:  
CNC Milling,  
Automatic Mode;  
Example of a collision  
during automatic run:  
an incorrect feed  
advance in "Z"  
(Z+002.000 would be  
correct) results in a  
collision at rapid  
traverse.



## Selecting Simulation Mode

After entering the program name, you select the simulation mode for program execution:

**F1**

**Automatic Mode:** Use **F1** to execute the selected NC program under continuous automatic control.

**F2**

**Single NC block:** Use **F2** to activate the single block function.

**F3**

**Interactive Mode:** To generate or edit the NC program interactively, press **F3** to switch to Interactive Programming mode.

The Interactive Programming functions are discussed in detail in [Chapter 8](#).

**F4**

**Traverse paths:** To track program operation by means of traverse path monitoring, press **F4**.

The use of the traverse path monitoring function is described in [Chapter 6.4](#).

**The additional functions are identical to those discussed above.**

## Complete Automatic Run



The NC program is loaded after the simulation mode has been selected. Should a setup sheet be available, the CNC Simulator is configured accordingly. Subsequently, the machining simulation is executed in real time. During this process, the workpart manipulation is demonstrated as a dynamic graphic screen display, while the current NC control block is displayed in the uppermost dialogue line (see also options for changing the graphic display). This allows the direct execution of the NC commands on the actual machine tool. In addition, the individual commands (such as feed rate, rpm, coolant, spindle) are taken into system status information, and the tool engagement time is evaluated and displayed.

Providing no operation errors occur, the Simulator uses the complete automatic run to execute the entire NC program without interruptions. Having completed the run it returns to the system status Automatic Mode. (As a rule, the program conclusion is effected by the "M30" command.)

During **program simulation**, the following functions are available:

**F2**

**Single NC block:** Use **F2** to switch to Single Block mode.

**F4**

**Dynamic zoom:** During CNC turning, the screen graphics can be enlarged and reduced by pressing **F4**.

**F8**

**Stop/Resume:** Use the **F8** toggle function to interrupt the current simulation, and to continue again.

**←**

Alternatively, after an interruption, the complete automatic run can be continued by pressing the **←** key.

**Further functions are identical to those discussed above.**

Figure 6.1-7:  
CNC Turning,  
Automatic Mode;  
Single Block mode.

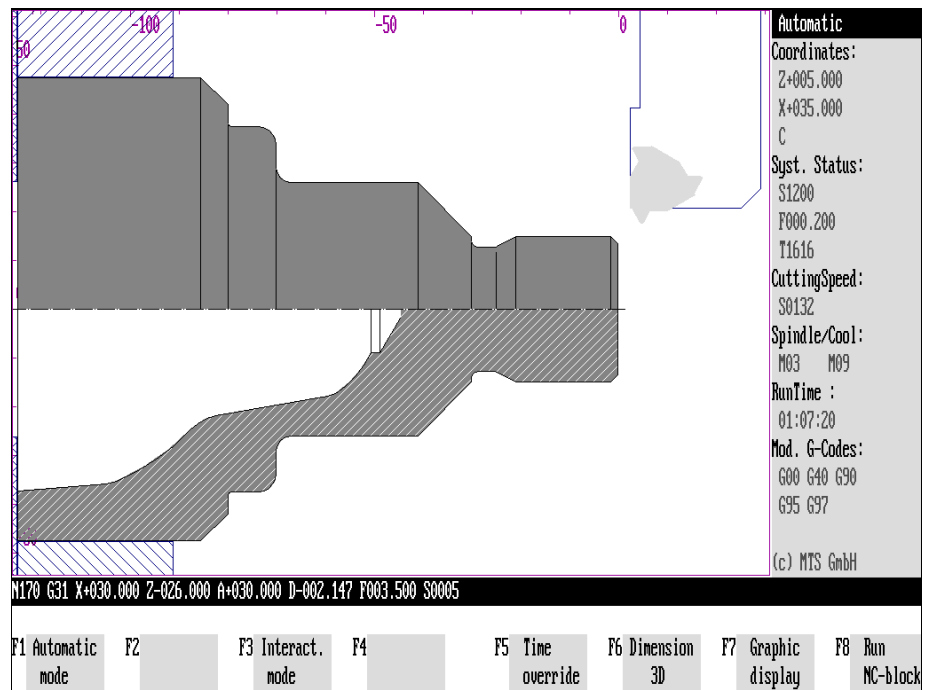
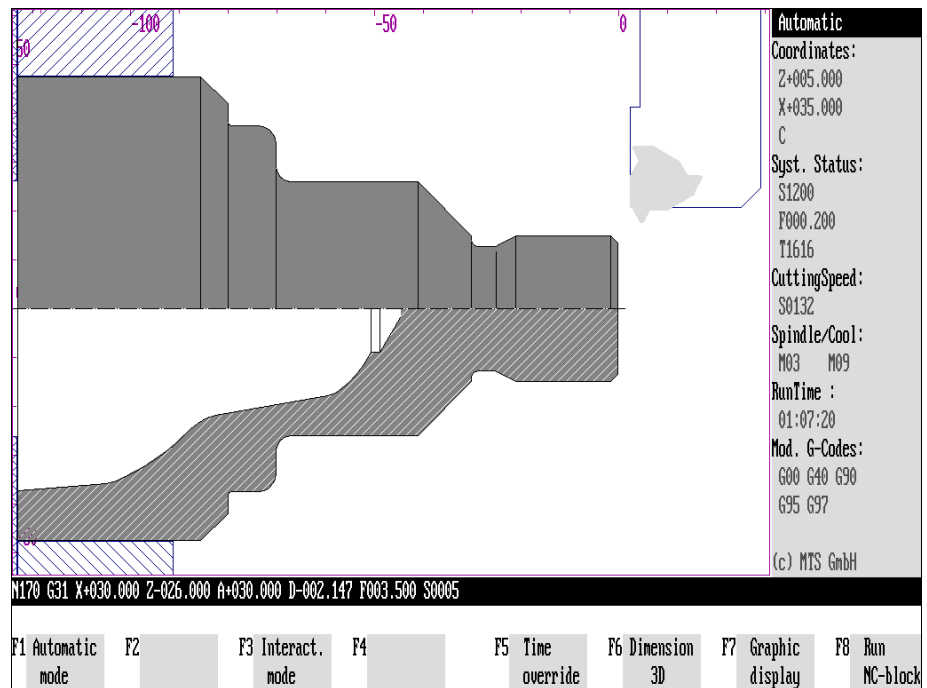


Figure 6.1-7:  
CNC Milling,  
Automatic Mode;  
Single Block mode.



## Single NC Block Mode



During single NC block mode, individual NC command blocks (or one machining cycle) are executed in succession. The simulation is "interrupted" until the next NC block is confirmed.

Error or collision messages, program end and interruptions are identical to those of the continuous automatic program simulation (see above).



or



**Execute NC block:** After each NC block or machining cycle, the workpart machining is interrupted, and to continue, the next NC block needs to be confirmed by pressing **F8** or **↵**.



**Automatic Run:** Use **F1** to select the continuous automatic run.



**Interactive:** Use **F3** to switch to the Interactive Programming mode.

**The additional functions are identical to those discussed above.**

## Cancel



The simulation can be interrupted at any point by pressing **ESC**. The system status is then displayed to be the Automatic Mode again.

## Collision Monitoring

If a program error or collision occurs during the simulation run, a corresponding message is displayed, and the execution of the NC program is canceled. The error message needs to be confirmed by pressing any key, after which the Simulator returns to the Automatic Mode main menu (unless you are in Interactive Programming mode, see below).

Figure 6.1-1: CNC Turning, Automatic Mode menu sequence schematic.

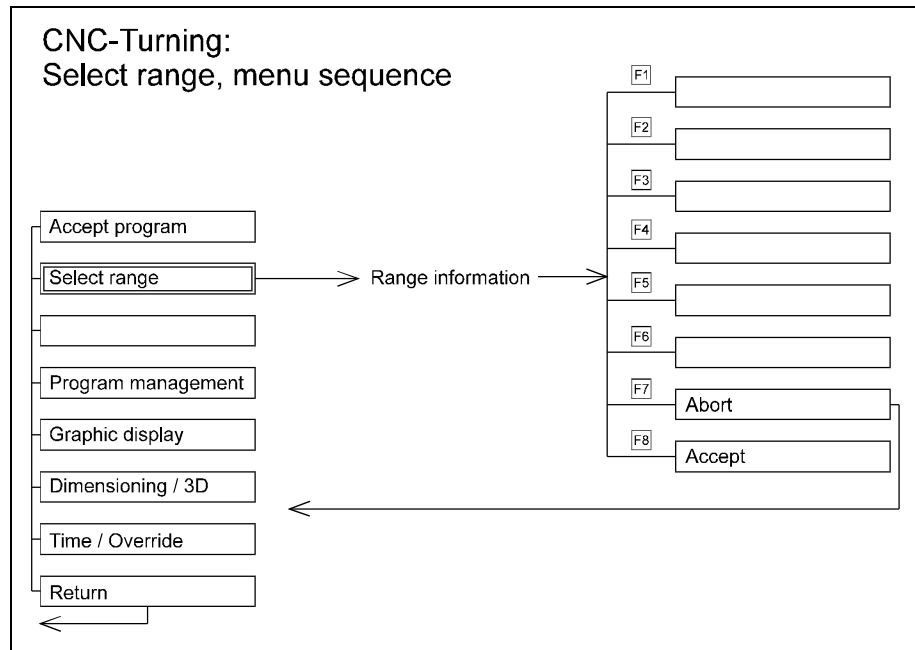
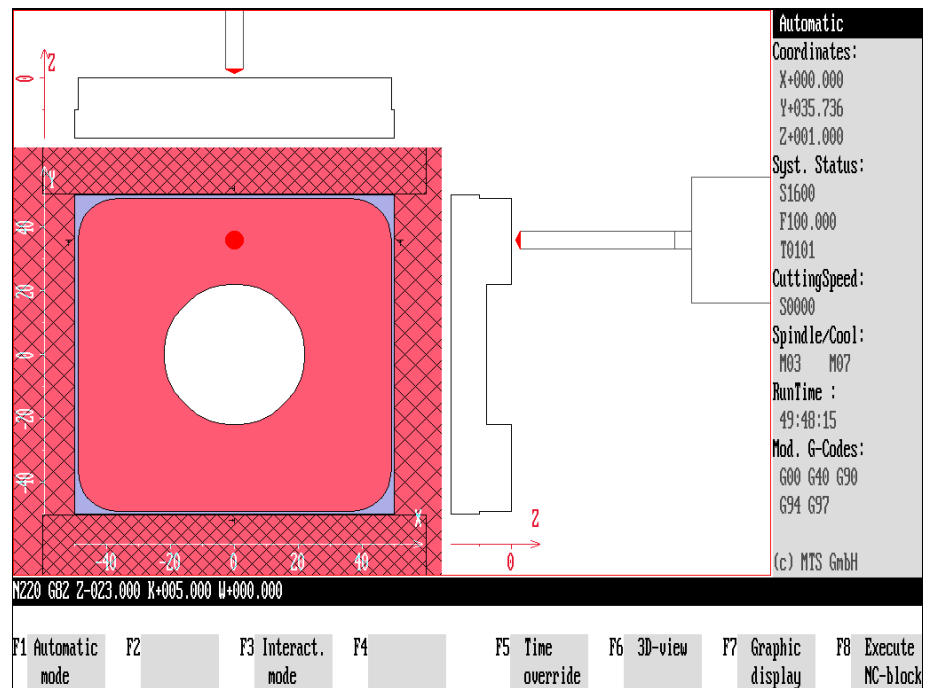


Figure 6.2-1: CNC Turning, Automatic Mode; to start the simulation of a specific part of an NC program, the starting and ending block numbers are entered.





## 6.2 Simulation of a Selected Program Part

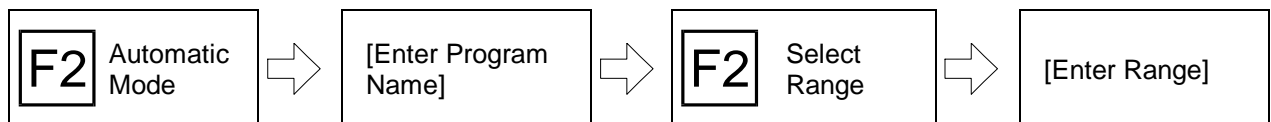


As a rule, the program blocks of an NC program are numbered by a three-digit block number preceded by the address character "N" (see also Programming Instructions). In automatic mode, irrespective of the use of the Continuous Automatic Run or the Single NC Block option, it is possible to simulate only specific program "segments", for example., to run a program range from block "N001" to "N180", or from "N265" to "N555". With this program function you are able, for instance, to test only those program sections you have previously modified or reprogrammed, without having to run through the entire NC program.

### Note

Please note that starting an NC program at a specific block number generally presupposes certain workpart geometry: If, for example, a blank has been inserted, this can result in a collision if certain working procedures are presupposed! Therefore, after having modified the workpart save it (see also above, and Chapter 11), and set up the Simulator accordingly.

### Selection Example:



It is assumed that, the Automatic Mode has been activated already.

[Program Name]

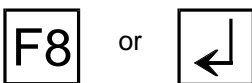
First, enter the name of the NC program in the dialogue line.



**Select Range:** To execute certain parts of a program simulation, **the range definition function is first chosen by pressing F2**, and only **after this the program name is confirmed by pressing F8**.

[Enter Starting & Ending Block Numbers]

To define the desired NC program range, enter the last or first block number before or after the program range that is to be simulated. If the entered block number is not available, the next closest block number will be used.



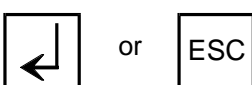
**Confirm:** Furthermore, both block numbers need to be confirmed first by pressing F8 or .

If no block numbers are entered, the program uses as default values "N000" for the start, and "N999" for the end of the block to be simulated.

An example: The NC program "%220200" is to be run from the beginning up to block number "N135". To do this, you confirm the default of "0" by pressing , enter "136" in the second field, and confirm again with .

Subsequently, you choose the simulation mode, as described above.

This causes the entire NC program to be loaded (to load the machine status and the modal functions); however, the actual program execution is limited to the defined range.



After completion of the simulation a dialogue query is displayed whether the remaining NC blocks are to be executed or not.

Figure 6.3-1: Automatic Mode, switching simulation modes (simplified schematic).

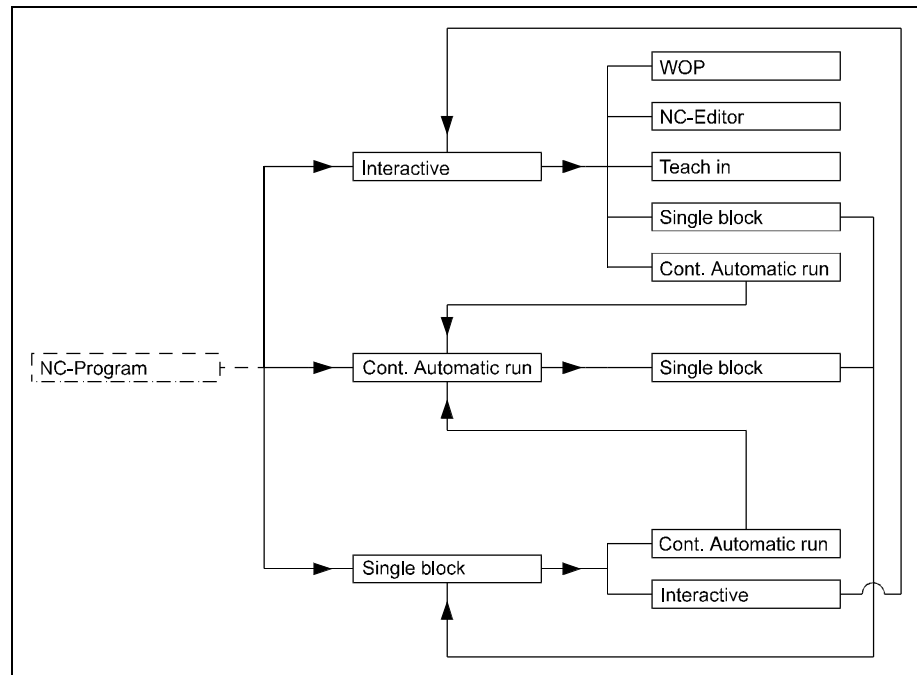


Figure 6.3-2: CNC Turning, Automatic Mode; use **F2** to switch from Complete Automatic Run to Single Block mode.

The screenshot displays a 3D CAD model of a turned part with a tool cutting it. The coordinate system shows Z from -100 to 0, and X from 0 to 100. The control panel includes function keys F1 through F8 with labels: F1, F2 Single block, F3, F4 Dynamic Zoom, F5 Time override, F6 Measure 3D, F7 Graphic display, and F8 Stop\continue. A status window on the right provides the following information:

Automatic	
Coordinates:	Z-038.000
	X+053.000
	C
Syst. Status:	S0901
	F004.000
	T1212
CuttingSpeed:	S0150
Spindle/Cool:	M04 M09
RunTime :	03:01:65
Mod. G-Codes:	G01 G40 G90
	G95 G96
(c) MTS GmbH	

At the bottom left of the screen, the text "N100 G80" is visible.

## 6.3 Switching Simulation Modes

As a special feature, the CNC Simulator offers the option to switch from one simulation mode into another. At any time, you can switch to one of the following modes:

- Complete Automatic Run
- Automatic Single NC Block Run
- Interactive Programming

without interrupting program execution. Furthermore, as long as Interactive Programming is active, all additional CNC Simulator programming options, such as Editor, Teach-In, and the Workshop-Oriented Programming are available. With these parallel operating functions, the CNC Simulator provides convenient and practice-oriented applications in NC program generation, testing and optimizing. These can be effectively used in actual production as well as in CNC training.

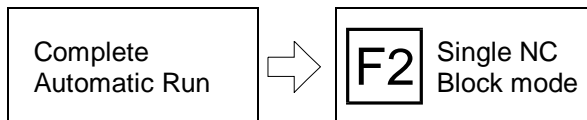
The two examples below serve to demonstrate the switch from one simulation mode to another. The schematic on the facing page illustrates how easy it is to switch between the available options.

### Cross Reference

The Interactive Programming mode is discussed in detail in [Chapter 8](#).

### Switching from Automatic Run to Single Block Mode

#### Selection:



#### Procedure

In this example it is assumed that the currently selected mode is Complete Automatic Run, and that you wish to switch to the Automatic Single Block mode.

**F8**

**Stop/Resume:** For better understanding of the procedure, we suggest that you first interrupt the simulation by pressing **F8**. However, this step is not obligatory. With certain procedures (i.e. tool changes), the current work will be completed before the simulation is halted.

**F2**

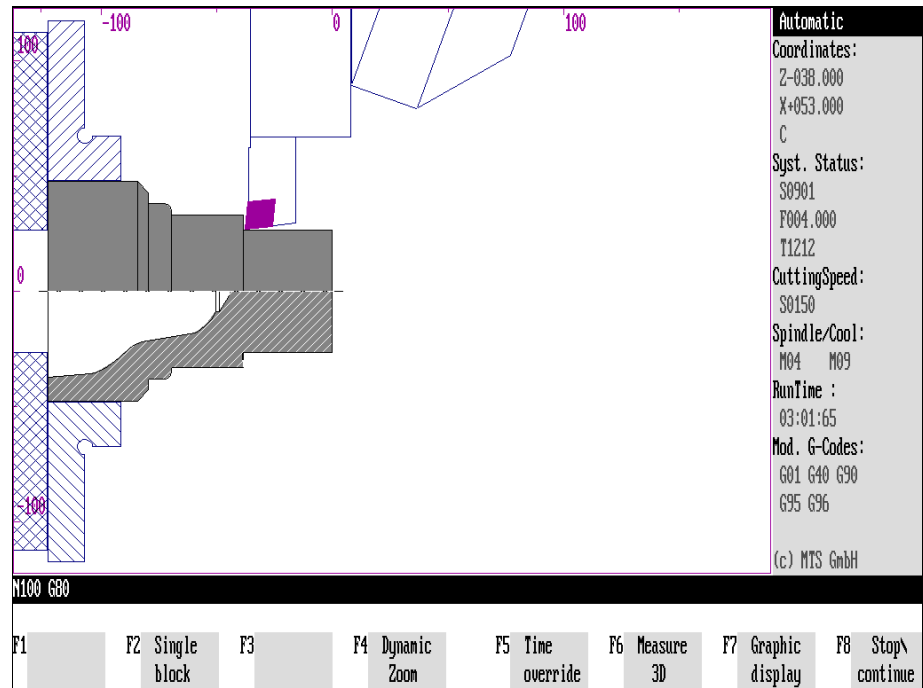
**Single Block Processing:** Subsequently you can switch to Single NC Block mode by pressing **F2**.

If you had interrupted your simulation before the switch, you can now continue it by pressing **F8**,

**F1**

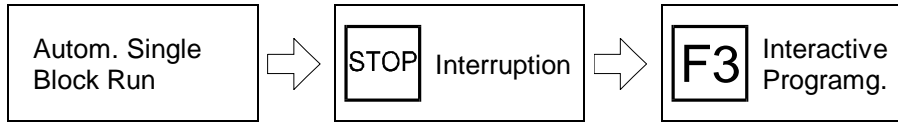
**Complete Automatic Run:** From the Single Block mode you can return to the Complete Automatic Run mode by pressing **F1**.

Figure 6.3-2:  
CNC Turning,  
Automatic Mode;  
from Single NC Block  
mode you can switch  
to Interactive  
Programming by  
pressing **F3**.



## Switching from Single Block Mode to Interactive Programming

### Selection Example 1:



### Selection Example 2:



### Procedure

In Interactive Programming as well as in Single Block Mode, the machining simulation is processed block by block. For this reason, the Single Block Mode needs to be activated (see schematic) first to be able to switch to Interactive Programming during a machining simulation.

**F8** or [End block]

To switch from one simulation mode into an other, you either interrupt the current operation by pressing **F8** or you can change the mode after having completed an NC block:

**F2**

Interactive Programming is accessed by pressing **F2**.



**This places you in the Programming mode, enabling you to continue to the NC Editor or the Workshop-Oriented Programming mode.** With this switch you have consequently changed the operating mode of the CNC Simulator.

Figure 6.4-1: CNC Turning, Traverse Path Monitoring; menu sequence schematic in Automatic Mode.

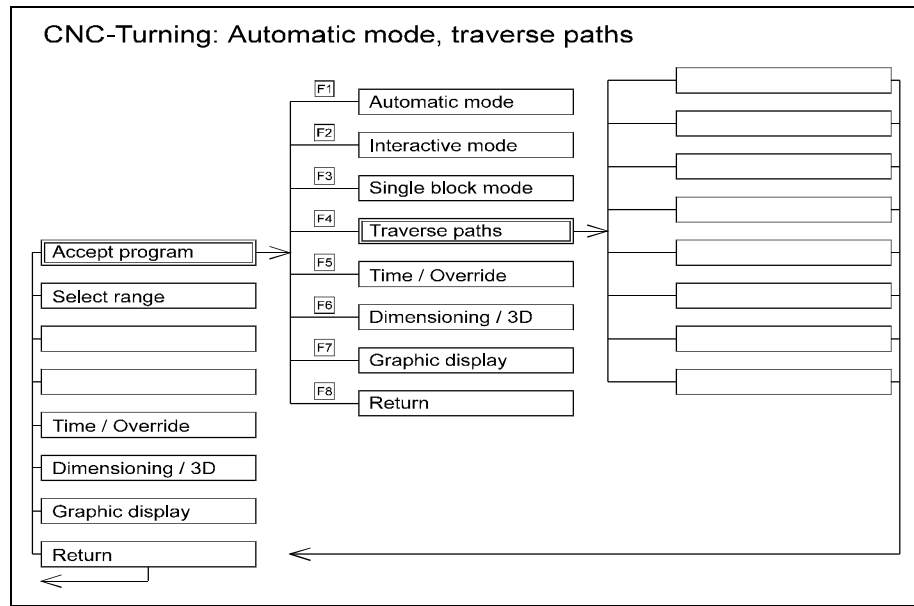


Figure 6.4-2: CNC Turning, Automatic Mode; Traverse Path Monitoring for an NC program range.

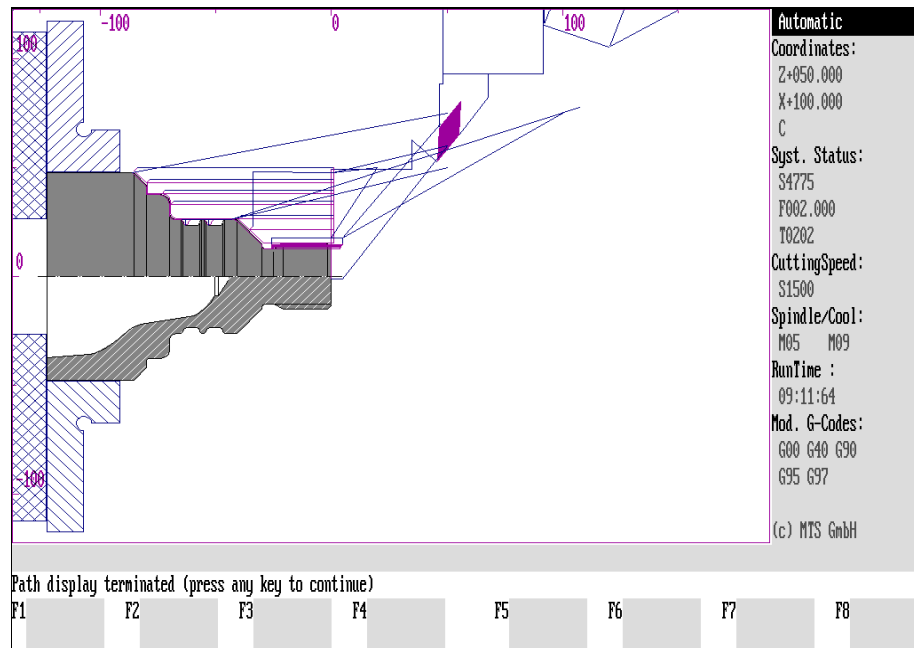
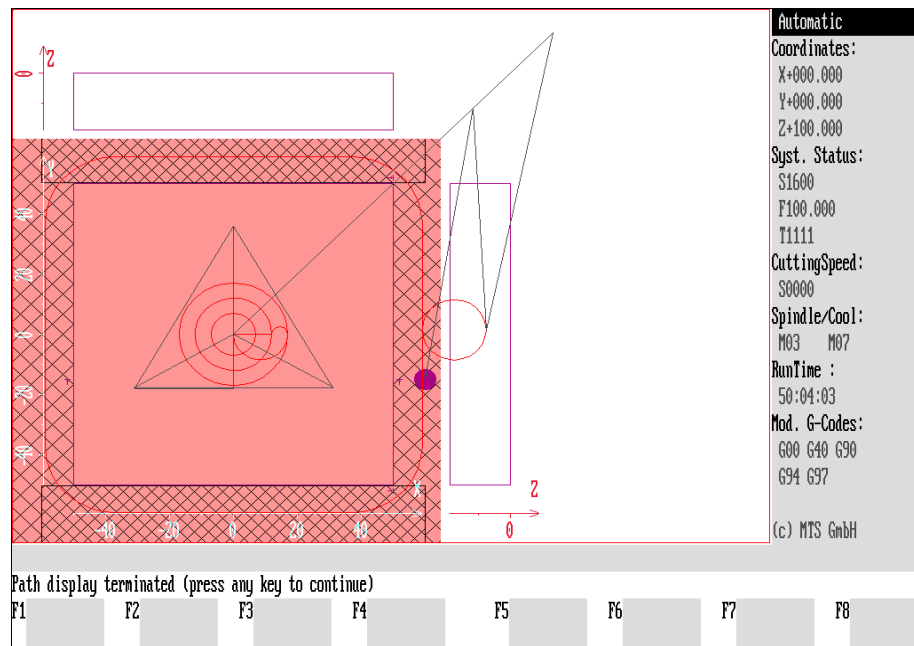
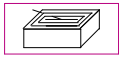


Figure 6.4-3: CNC Milling, Automatic Mode; Traverse Path Monitoring for an entire NC program.

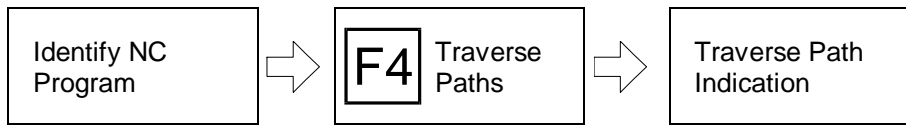


## 6.4 Traverse Path Monitoring



The traverse path monitoring serves as a monitoring function during the execution of an NC program. For the CNC Simulator, it can be selected for an entire NC program or for a part of it.

### Selection:



### Procedure

The Traverse Path Monitoring can be activated in the Automatic Mode.

#### [Enter Program Name]

Start by entering the name of the desired NC program. It is logical enough, that this program is located in the current subdirectory (as default ".\DPROG" or ".\FPROG").

If the program is not available, the corresponding error message is displayed. When in doubt use the Program Manager function (see Chapter 4) to ascertain which program files are located in the current directory.



**Traverse paths:** The traverse paths function is accessed by pressing **F4**.

The NC program is then loaded, and the traverse paths is graphically displayed. After the traverse path monitoring has been concluded a message is displayed: "**End of Traverse Path Monitoring** (press any key to continue)."



**Select Range:** If the **traverse path of only a part of an NC program is** to be displayed, first enter the program name with **F2** (see above) and define the deseired range before accessing the traverse path function.

### Note

If the program contains further NC blocks located after the monitored traverse paths of the currently displayed range, their paths can also be displayed:



or



To do this, confirm the following message: "Continue with **←**; cancel with **ESC**." by pressing the **←** key. Or, you conclude the traverse path display by pressing **ESC**.

### Return:

To return to Automatic Mode after concluding the traverse path display, press any key.

Figure 6.5-1:  
CNC Turning,  
Override/Times menu  
schematic.

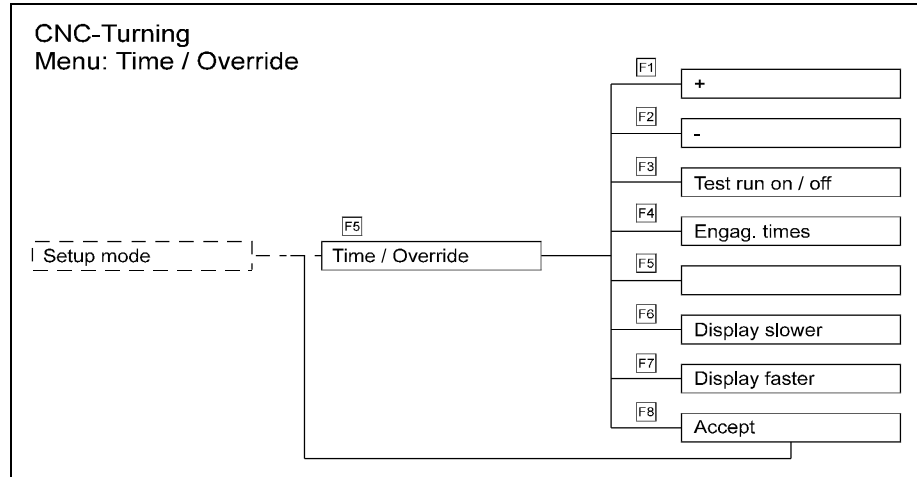


Figure 6.5-2:  
CNC Turning,  
Automatic Mode;  
Override/Times menu.

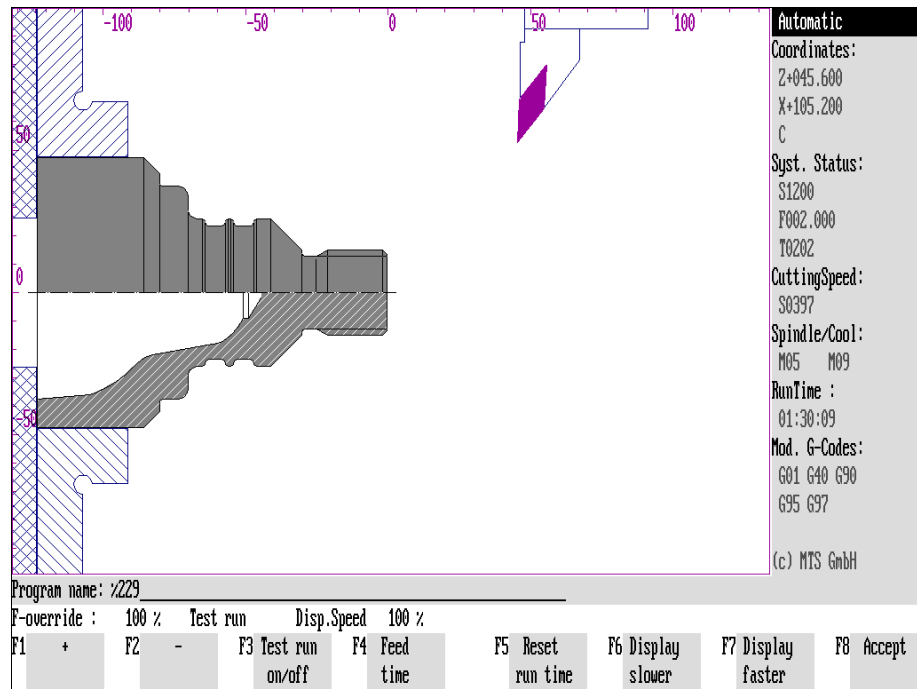
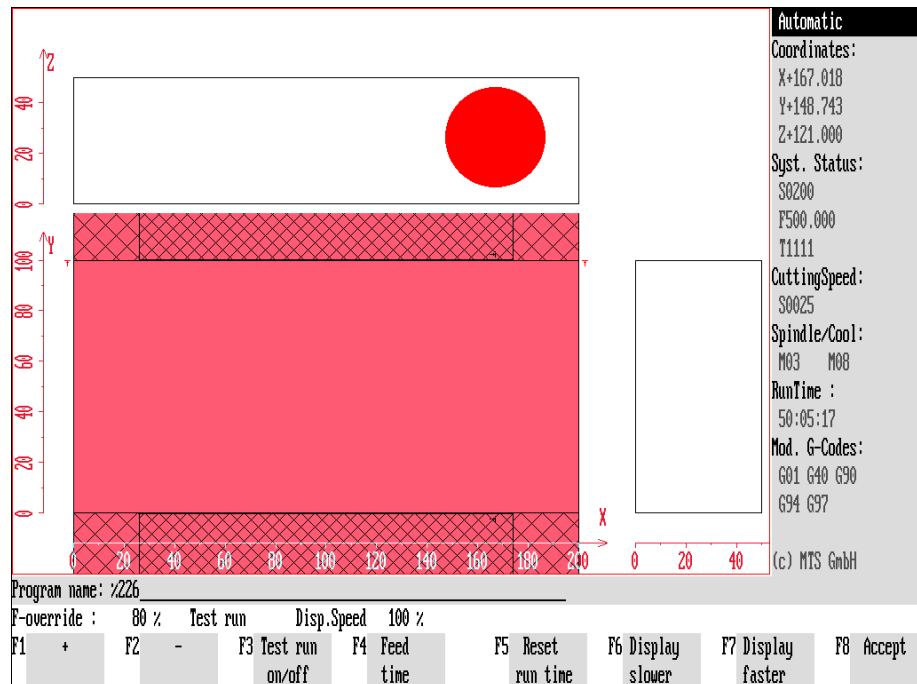


Figure 6.5-3:  
CNC Milling,  
Automatic Mode;  
Override/Times menu.





## 6.5 Override/Times

The supplementary Override/Times function is available in Automatic Mode as well as in Setup Mode. It is used to modify CNC Simulator feed rates and time settings. This function also provides "access" to the already calculated tool engagement times.

### Override

The feed rate override, also called feed rate overlapping, is used to increase or decrease the feed rate. This function enables improved monitoring of the tool movements, and optimizing cutting values during the test runs of an NC program. The CNC Simulator override can be set between 10% and 150%.

### Test Run/Slow Motion

The override being 100%, the workpart machining takes place in real time, i.e., its representation by the CNC simulation requires the same time as the machining of the actual workpart.

To accelerate the simulation of complex processes without having to change the technology data of the NC program, a "rapid motion" option "Test Run" is also available. This modus displays the graphic simulation of the workpart machining processes, depending on the processing power of your personal computer, at the maximum speed possible.

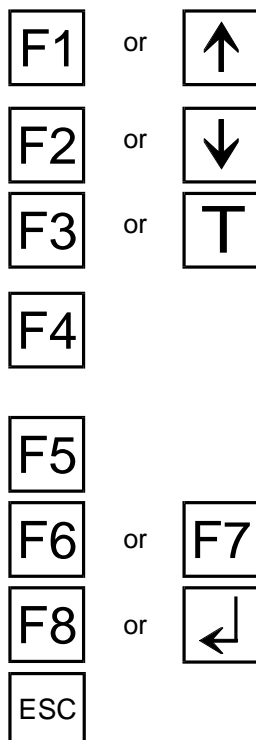
The opposite function "Slow Motion" allows a gradual slowing down of the displayed machining speed. It is especially useful to obtain a fluent and jerk-free tool movement with computers of lower processing power.

### Note

The time calculation which is displayed in the information column remains unchanged irrespective of the run mode selection of either Test Run or Slow Motion. It indicates the actual machining time, however, considering the given override.



The settings for Override or Test Run are directly entered in the Override/Times menu. During all working situations, this menu is accessed by pressing **F5**, producing a display of the current override value (in percent) in the dialogue line:



**+ / -:** To change the Override, the value is set to the desired percentage (between 10% and 150%) with the **F1** or **F2** function keys; or it can be

decreased with the **↑** or **↓** cursor keys, or increased with the **↑** or **↓** cursor keys.

**Test Run On/Off:** Use **F3** or the **T** key to activate or switch off the Test Run function. After activation, the message "Test Run" appears next to the override percentage value; the message disappears when Test Run is switched off.

**Engagement Times:** This function gives direct "access" to the calculated tool engagement times. Since the submenus concerned have already been discussed under the Turret/Magazine Configuration menu (see above), it is not repeated here.

**Time Zero Reset:** Use **F5** to reset the calculated machining time ("Runtime") to zero (Automatic Mode only).

**Slow Motion Faster/Slower:** This function is used to set the displayed machining speed between 10% and 1000%.

**Accept:** Use **F8** to confirm your settings, and to return to the Setup Mode main menu level.

Press **ESC** to cancel the operation without confirming any new settings.

Figure 6.6-1: CNC Turning, CNC Turning, Measuring/3D View menu schematic.

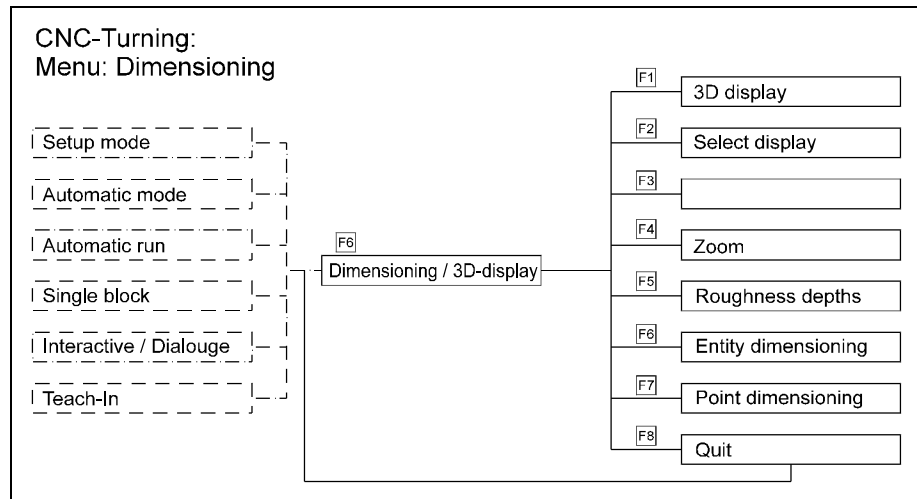


Figure 6.6-2: CNC Milling, CNC Milling, 3D View menu schematic.

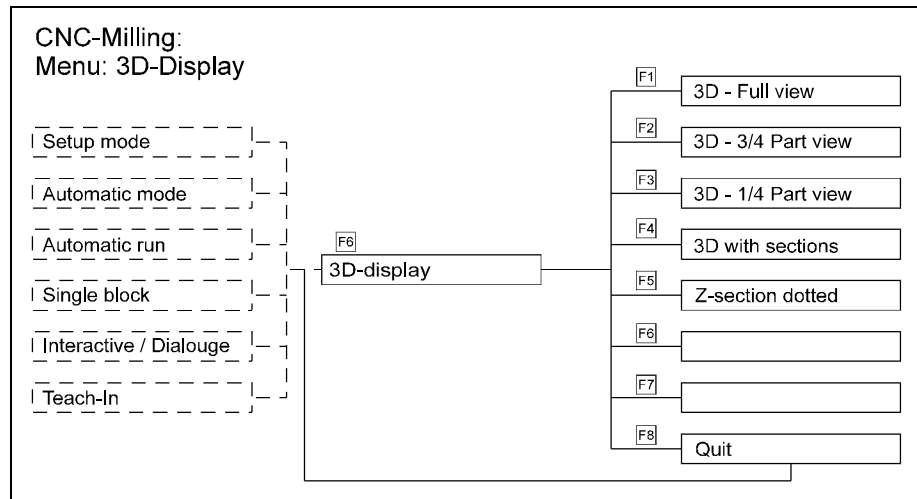
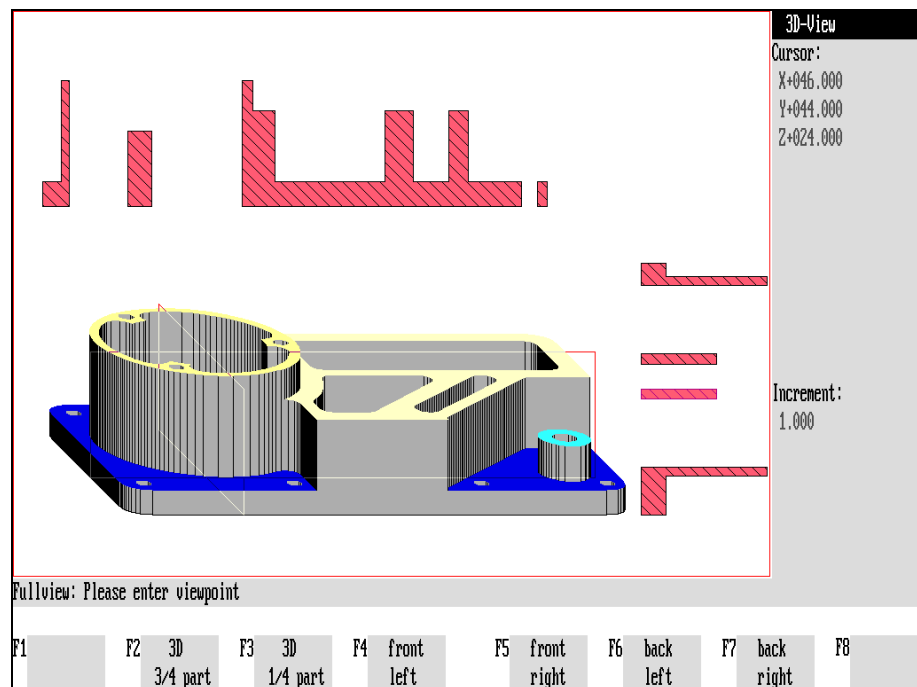
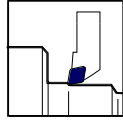


Figure 6.6-3: CNC Milling, 3D View, three-quarter view.



## 6.6 Measuring, 3D View

### Measuring CNC Turning

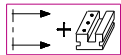
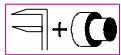
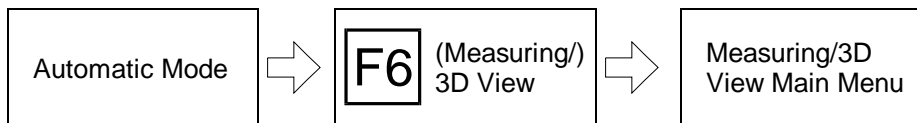


For the purpose of a **quality control** prior to production, **turned workparts** (including **threads**) **can be measured** with mathematical precision. Also, for the calculation of the machining processes the theoretical **roughness** is evaluated and displayed to check the surface quality.

### 3D Views

A function supporting CNC training is given by the option to display, at any time, 3D Views of the workpart, seen from different viewing angles. Version 5 of the program features 3D displays in both Turning and Milling Simulators. To display machining inside the workpart, any cylinder sectors (CNC Turning) or workpart quadrants (CNC Milling) can be cut out.

### Selection:



The Measuring (CNC Turning) and 3D View functions can be used during both the Complete Automatic Run and Automatic Single Block functions, as well as during Interactive Programming.

Depending on whether you are using the Turning or Milling Simulator, further options are available.

### Cross Reference

To avoid repetition (both the Measuring and the 3D View functions are featured in their own separate chapters), we would like to refer you to [Chapters 9](#) and [10](#), containing detailed descriptions of these operating functions.

Figure 6.7-1: CNC Turning, Graphic Display; Menu sequence schematic.

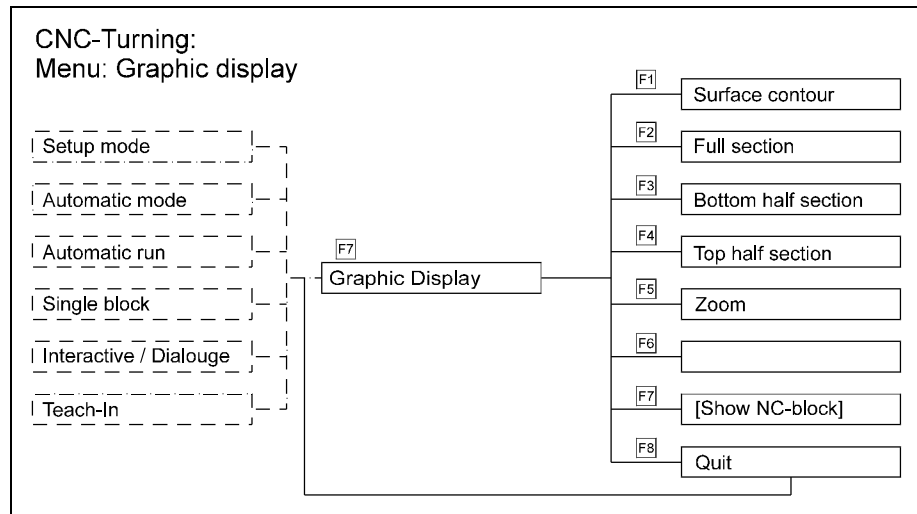


Figure 6.7-2: CNC Turning, Modifying Graphic Display. Depending on the current setting, several NC information blocks can be displayed.

**Automatic**  
 Coordinates:  
 Z-022.727  
 X+040.851  
 C  
 Syst. Status:  
 S1938  
 F000.300  
 T0202  
 CuttingSpeed:  
 S0250  
 Spindle/Cool:  
 M04 M08  
 RunTime :  
 17:11:71  
 Mod. G-Codes:  
 G00 G40 G90  
 G95 G96  
 (c) MTS GmbH

N200 G23 P135 Q170  
 N205 G40  
 N210 G00 Z-017.000  
 N215 G00 X+046.000  
 N220 G78 X+044.000 Z-025.000 I+002.000 X+005.000

F1 [ ] F2 Single block F3 [ ] F4 Dynamic Zoom F5 Time override F6 Measure 3D F7 Graphic display F8 Stop\ continue

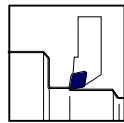
## 6.7 Modifying Graphic Display, Zoom Function

The CNC Turning and Milling Simulators feature various functions to emphasize specific machining operations by modifying the screen graphics. This includes displaying sections, displaying several NC program lines, and the dynamic Zooming function (turning).

### Cross Reference

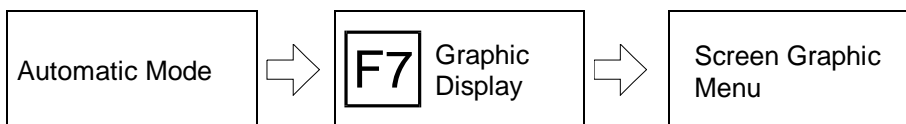
Since the modifications of the graphic display are discussed in detail in [Chapter 10](#), they are presented here as a summary only.

### CNC Turning



When using the CNC Turning Simulator, you can modify the screen display and enlarge and reduce a section of the displayed piece (Dynamic Zoom function):

#### Selection Example:



GRAPHIC

**Screen Graphic:** By far from all operating functions you can access the Screen Graphic menu by pressing **F7**.

**F1**

**Exterior View:** To show an exterior view of the workpart, select **F1**.

**F2**

**Full Sectional View:** To display full section of the workpart, select **F2**. To differentiate the representation of a section from that of an elevation, the cut surfaces (section) are represented with a cross-hatch pattern according to industry standards.

**F3**

**Half-Section, Bottom:** Use **F3** to display the workpart as a section below its longitudinal axis, and as an elevation above its longitudinal axis.

**F4**

**Half-Section, Top:** Use **F4** to display the workpart as a section above its longitudinal axis, and as an elevation below its longitudinal axis.

**F5**

**Zoom:** To enlarge or reduce the displayed representation, press **F5** to access the zoom function.

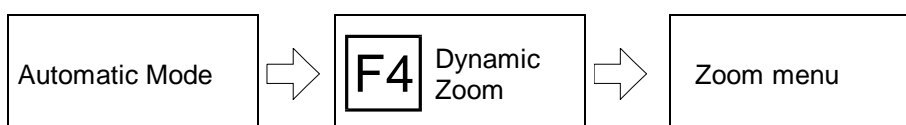
**F7**

**Show NC program lines:** During machining or simulation, the current NC block is displayed in the upper dialogue line (default). To display several NC blocks, you can redefine the required number of blocks by pressing **F7**. This function can be activated in Automatic Mode only.

**F8**

**Accept:** Use **F8** to confirm your modifications, and to return to your starting situation.

#### Selection Example:



**F5**

**Dynamic Zoom:** Even during a current machining run, the graphic display section can be modified by accessing the dynamic Zoom function with **F4**. Refer to [Chapter 10](#) for a detailed description of the Zoom function.

Figure 6.7-3:  
CNC Milling,  
Graphic Display;  
menu sequence  
schematic.

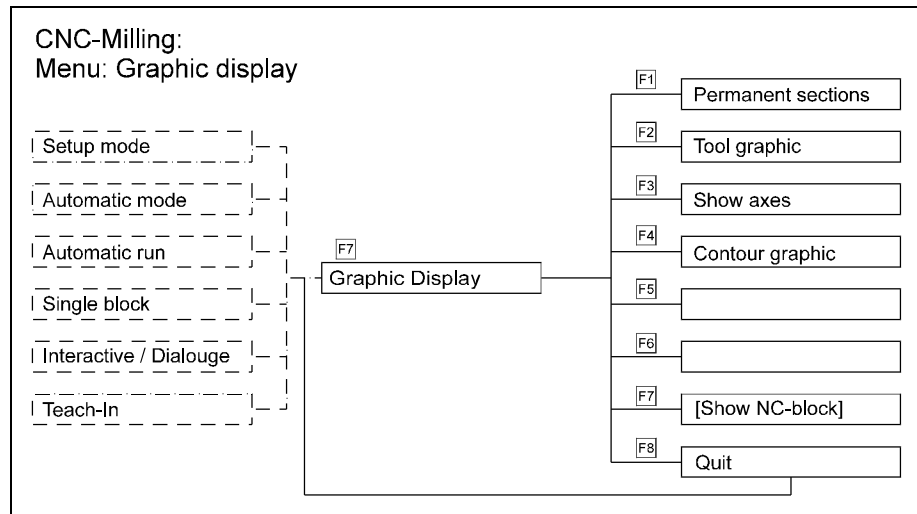
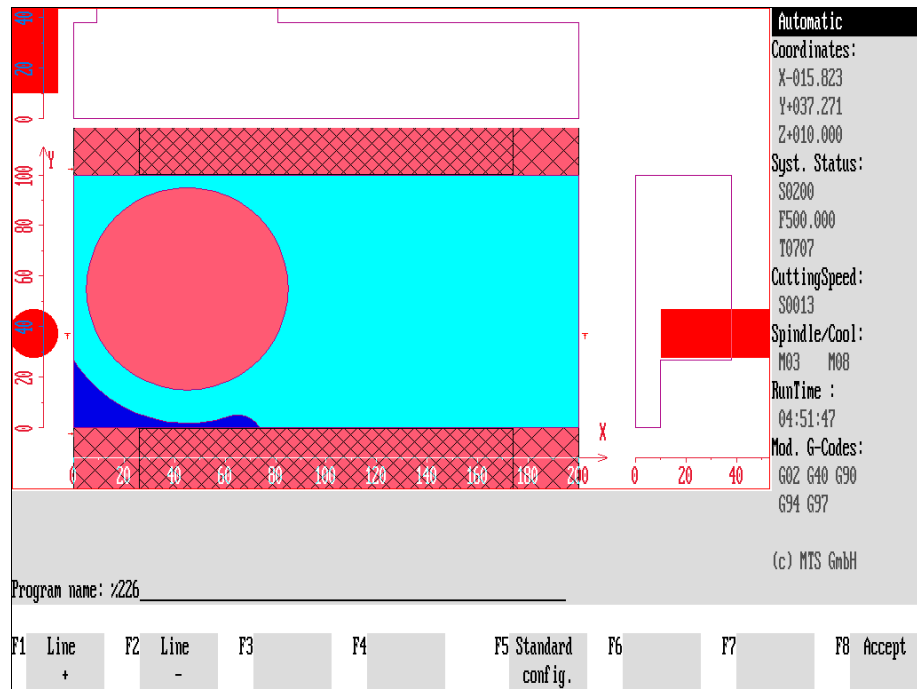
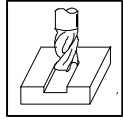


Figure 6.7-4:  
CNC Milling,  
Automatic Mode;  
the number of  
displayed NC lines can  
be varied during both  
**turning** and **milling**  
operations.

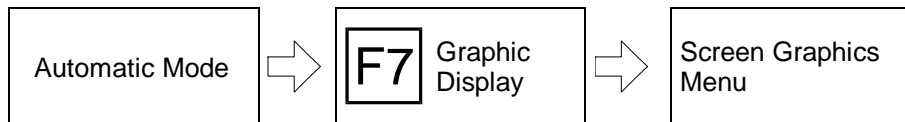


## CNC Milling



To modify the the screen graphics during CNC Milling, the following functions are available:

### Selection Example:



GRAPHIC

**Screen Graphics:** It is possible to access the Screen Graphics menu by pressing **F7** almost from all operating functions.

**F1**

**Permanent Sections:** In addition to the default workpart top view, it is possible to toggle section representaitons on and off by selecting **F1**.

**F2**

**Tool Display:** To toggle representation of the workpart additionally into the section graphics of the workpart, select **F2**.

**F3**

**Show Coordinate Axes:** Use **F3** to toggle a display of the coordinate axes on and off.

**F4**

**Workpart Outline:** Use **F4** to toggle on and off the workpart contours highlighted in color.

**F7**

**Display NC lines:** As a default in Automatic Mode, the current NC block is displayed in the upper dialogue line during the machining process. To display several NC blocks, redefine the number of blocks to be displayed by pressing **F7**.

**F8**

**Accept:** Use **F8** to confirm your modifications, and to return to your starting situation.



You can change the screen display also during workpart milling simulation. Make use of this option and try to select the most ideal display mode!

Figure 6.8-1: CNC Simulator, Programming Aids menu sequence schematic.

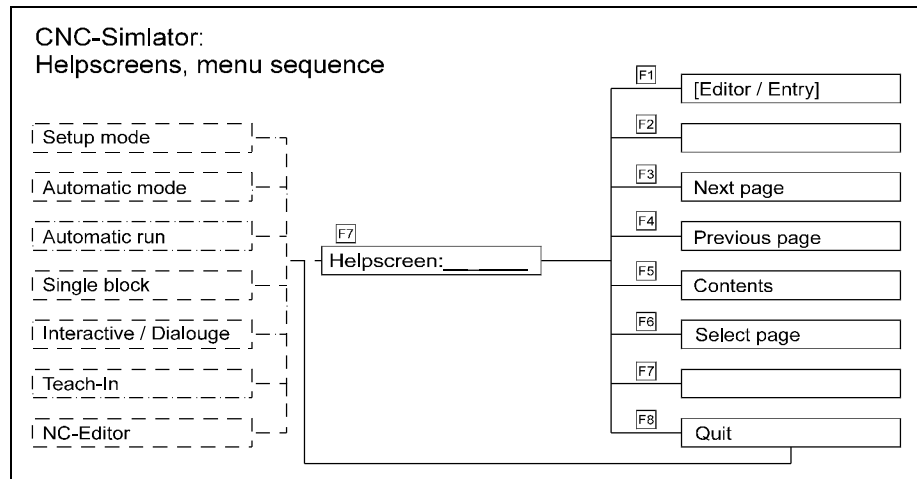


Figure 6.8-2: CNC Turning, Programming Aid on the "G83" cycle with travel range limitation "G36".

MTS CNC 5.1 G83 page 1/1

Deep drilling cycle with chip breaking

The command G83 defines a deep drilling cycle with chip breaking on the first plane.

**G83 Z... W... A... B... D... K...**

- Z- Drilling, incremental to the 1st plane
- W 2nd plane
- A Dwell time for chip removal
- B Dwell time on the drill bottom in seconds
- D Degression (decrease of the drilling depth)
- K 1st drilling depth

If W had been programmed, the tool must be positioned on the 2nd plane with cycle invocation by G77, G78 or G79.

Diagram: Shows a drill bit drilling into a workpiece. The 1st plane is the top surface, and the 2nd plane is a lower depth. Green arrows indicate rapid traverse, and purple arrows indicate slow feed motion. The drill bit is shown at the 1st plane and then moving down to the 2nd plane.

1 Editor input 2 3 Next page 4 Prev. page 5 Directory 6 Select help 7 8 Return

Figure 6.8-3: CNC Milling, Programming Aid on cutter radius compensation with contour-parallel approach "G42 - G46".

MTS CNC 5.1 G42G46 page 1/1

Cutter radius compensation right of the contour

The command for the approach in a semicircle and for the activation of the cutter radius compensation (CRC) is:

**G42 A... G46 G01 X... Y... Z...**

- G42 Activates CRC right of the contour
- A Diameter of the semicircle between infeed point and contour starting point
- G46 Approach in a semicircle
- X,Y Coordinates of the contour point KP1
- Z Infeed depth
- G01 Approach to infeed point ZP which is computed by the control.

Diagram: Shows a semicircular approach of a cutter to a contour point KP1. The diameter of the semicircle is labeled 'A Diameter'. The cutter position is labeled 'Cutter-position'. The infeed point is labeled 'ZP'.

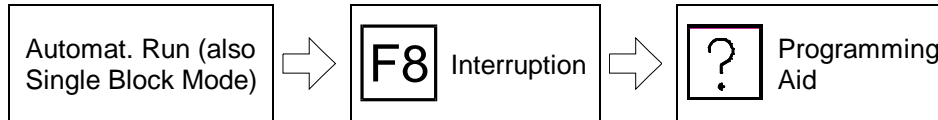
1 Editor input 2 3 Next page 4 Prev. page 5 Directory 6 Select help 7 8 Return



## 6.8 Programming Aids

The Programming Aid functions are available during simulation as well. For this purpose, the program execution is stopped, and the appropriate help information is called. Subsequently the program simulation can be resumed.

### Selection:




### Procedure

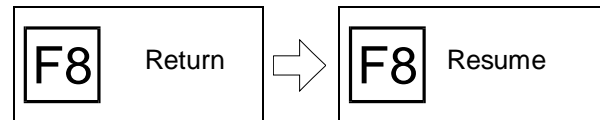


A Programming Aid function can be accessed either during the interruption of a program simulation or while the Automatic Mode system status is active.




Press the  key to load the help screen management. Continue by entering the name of the desired help screen, or select a group register from which you can select the desired programming aid. Refer to [Chapter 11](#) for a detailed discussion of these help functions.

### Return:



To close the help function again, press the  function key.

You then return to the function from which you initially called the help function. After an interruption of the simulation, you can resume the operation by pressing .



## 7. NC Programming

### Introduction

**Programming Modes** For creating, modifying and optimizing of NC programs, the CNC Simulator provides four programming modes:

- **Editor**
- **Interactive Programming**
- **Teach-In**
- **Workshop Oriented Programming (WOP interface).**

The first three of the listed functions are supplied as standard software components, while the Workshop Oriented Programming option is available as an optional program enhancement package.

**Program Management** The NC Editor is equipped with a easy NC program management to delete and print NC programs conveniently without prior knowledge of the operating system.

The programming modes are structured to meet various requirements and to support the user with comfortable operating functions as well as some error checking or collision monitoring functions during program generation.

**NC-Editor** The Editor with its user interface, especially designed for NC programs, as well as a its "syntax checking" function, is the easiest way to enter and edit NC data blocks.

**Interactive Prog.** The Interactive Programming function is a reciprocal extension of Automatic Mode and Editor. Block by block, the generation of an NC program is accompanied by machining simulation and collision monitoring.

**Teach-In** The Teach-In function is a modification of interactive programming. Like in Setup Mode, the workpart is machined manually. The CNC Simulator uses the traverse paths for the automatic generation of simple "G commands" (based on the DIN 66025 standard), and transfers them into the NC program.

**WOP Interface** Finally, the WOP (Workshop Oriented Programming) function allows easy programming of complex contours without additional programming aids. With the support of dialogue guidance and help graphics, the dimensional data of the workshop drawing (even if the dimensions are not according to NC conventions) are entered in the entry menu, and the programmed workpart contour and the generated NC data block are displayed simultaneously. WOP-generated contours can be gauged and zoomed as well.

Corresponding to the option of the four programming modes, it is possible to switch from one mode into the other during NC programming. The user can so benefit from the specific advantages of each entry system. Of course, the various Programming Aid functions can be called at any time to generate or edit an NC data block.

Overview of the programming commands according to the MTS command code (excerpt):

	<b>Turning and Milling:</b>
G00 G01-G03 G04 G09 G20,G21 G22 G23 G24 G25 G26 G40-42 G54-56 G53-59 G71-73 G90-91	Rapid traverse Linear and circular interpolation Dwell In-position programming Inch/Metric switch Subprograms up to 11-fold nesting depth Program segment repeat Unconditional jump Approach reference point Approach tool change point Tool nose / cutter radius compensation Set zero reference point Shift zero reference point Segment contour programming Fixed / incremental dimensions (absol. relative coordinate system)
	<b>Turning</b>
G28 G31 G33 G36 G50, G51 G57 G58 G65, G66 G75, G76 G81, G82 G83 G84 G78, G85 G79, G86 G87, G88 G92 G94, G95 G96, G97	Move tailstock Thread cycles Non-standard thread Traverse path limitation for G83 Definition of desired finished contour Finishing allowance for cycles G81-G83 Set zero point Longitudinal/tranv. roughing cycle, tapered profile Longitudinal/tranv. roughing cycle, rectang. contour Longitudinal/tranv. roughing cycle, any contour sequence Contour-parallel roughing, multiple cycle Deep thrilling cycle Undercuts Recesses Radius (chamfer) cycle RPM Limitation Feed rate in mm / min / per revolution Constant cutting speed / constant RPM
	<b>Milling:</b>
G10 G11-G13 G45-G47 G57 G67 G61, G77 G78 G81-G86 G87-G88 G89	Rapid traverse (polar coordinates) Lin. / circular interpolation (polar coordinates) Approach / departure conditions for CRC Set zero point Rectangular pocket Drilling cycle: hole pattern on a circle Drilling cycle: hole pattern on a straight line Drilling, reaming, thread cycles Pockets (rectangle, circle) Pins

## Programming Code

The terms "programming code" or "command code" refer to all G commands or preparatory functions, cycle invocations and machine commands, including their respective parameters and their combinations, which can be executed by a CNC control system. Especially within the free ranges of the DIN standard, there are considerable programming differences between manufacturers. As a result, NC programs written for different control systems cannot be changed with each other.

For this reason the programming code, and at the same time the command syntax<sup>1</sup>, of the MTS Simulators, has not been rigidly defined but can be re-defined and specified in an assignment table by means of a User Defined Programming Code module. This enables the user to create various DIN block-oriented programming codes, and to use them in his NC programming work. The User Defined Programming Code program module is an optional enhancement of the CNC Simulator software.

The generation of additional programming codes uses the default command codes supplied with the MTS Simulators. Accordingly, their language has been structured to be as comprehensive as possible. (See also table on facing page; for detailed discussion on programming, refer to the section on Programming Instructions in this manual, and to the instructions for the User Defined Programming Code.)

Naturally, in addition to the above mentioned G commands, machine functions selected through the letter "M", as well as other control commands (letters "F", "S", and "T"), are available. It is also possible to use NC programming with parameters or subprograms.

### CNC Controls

As mentioned, the CNC Simulators can be adapted to various programming codes. Provided that the corresponding assignments of respective NC commands including formats, passwords and register values are available, the user can simulate the programming code of other CNC control systems. This feature produces a two-fold advantage for production and training:

1. The user can operate the CNC Simulator with the same programming code as the machine tool control system uses. It is possible to demonstrate during training the differences between CNC controls or, for example, the PAL Programming code can be used (see also the section on PAL Programming Code Instructions).

2. If the CNC Simulator has been configured for a specific control system, and NC programs have been created with this specific programming code, a post processor is not required any more to transfer NC programs to this control system, since the program can be executed (with the exception of the possibly expanded control characters required for the transfer itself; see also the section on User Defined Programming Code Instructions.)

Also, for NC programs generated in the MTS programming code, more than 60 post processors and one generalised post processor are available.

---

<sup>1</sup> In this context, 'syntax' means the formal structuring of the NC commands, i.e., the parameter sequence and assignment, and the signs of values.

Figure 7.-1:  
CNC Turning,  
Supplementary  
Programs menu;  
use **F2** to call the  
Format Conversion  
Program.

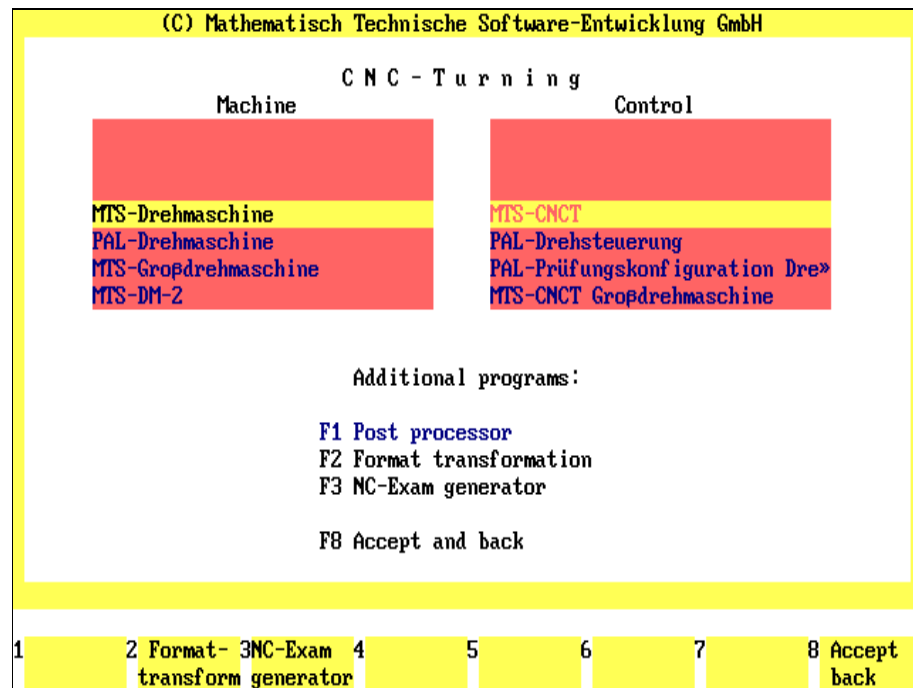
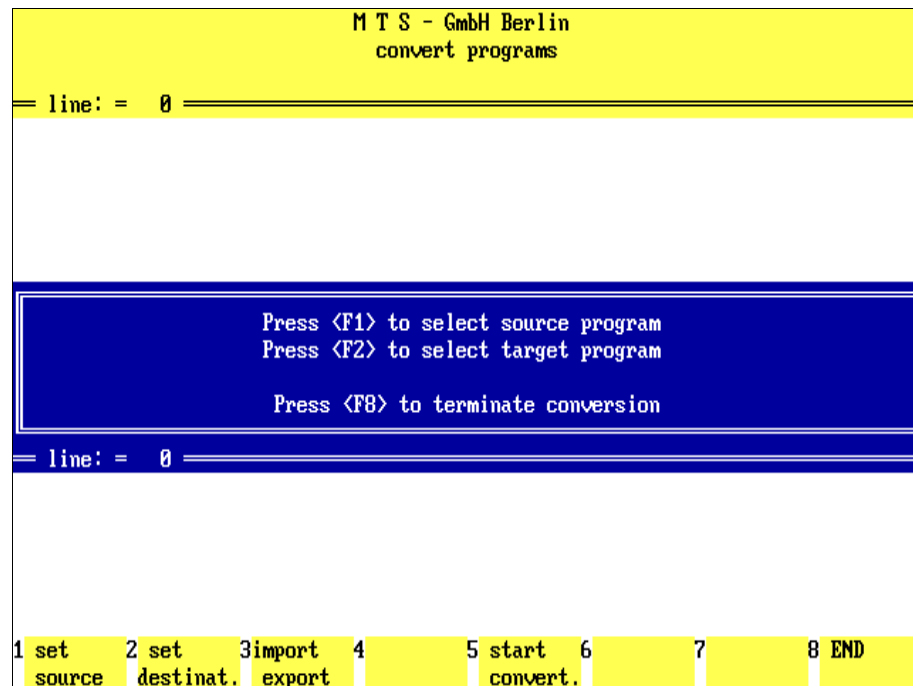


Figure 7.-1:  
Format Conversion  
Program menu;  
NC programs written  
with an older version  
of the CNC Simulator  
or another  
programming system  
need to be converted  
before they can be run.



## Format Conversion



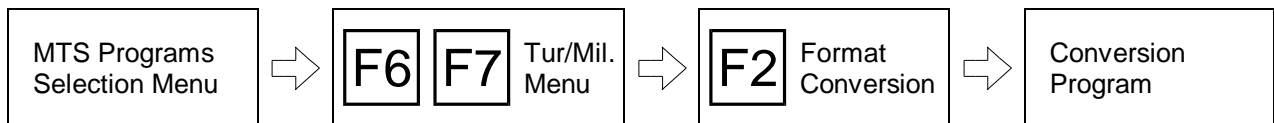
**This section is relevant only for those who wish to work with programs generated by a previous version of the CNC Simulator or the INCAD programming system!**

As a rule, NC program data information has a uniform structure to allow their editing with an NC editor or programming system. The information structure may include definitions such as upper and lower case letters, or the allowable digits after or before the decimal point. Such definitions are called "dataformats".

In most cases, data formats are automatically assigned by the programming system used to create the NC program, and do not require any configuration by the user (see also [section 7.1](#)). However, if the NC program is to be interchanged or transferred between two programming systems, and different data formats have been used, a format conversion is necessary even if both are based on the same program code. To facilitate format conversion, the CNC Simulator has an automatic conversion program providing you with a quick and error-free way of adapting the data formats to the MTS standard.

Since the new version of the CNC Simulator features a slight change in the data format, you are advised to run the format conversion for all NC programs that you wish to use and that were created with an older version of the CNC Simulator or the INCAD NC programming system. Please note, however, that this conversion is not required if you wish to edit the NC programs by means of the Free Format Mode (see below).

### Selection:



Starting from the MTS Program Selection menu, first select the option Turning Menu or Milling Menu (supplementary programs) by pressing **F6** or **F7**, respectively. Continue by pressing **F2** to go to the Format Conversion Program menu.

### Screen Layout

The main screen area is horizontally divided into two sections in which the NC data blocks of source and target programs are simultaneously displayed during the conversion run. Above each window, the name and identifier of the respective source and target program file is displayed. As usual, the function keys at the bottom of the screen are labeled with their respective functions:

**F1**

**Specify Source Program:** To specify the source program, select **F1**.

**F2**

**Specify Target Program:** To name the target program, select **F2**.

**F3**

**Import/Export:** this function is reserved for the new program management, but is not implemented in the present CNC Simulator software version.

**F5**

**Start Conversion:** After you have defined the source and target programs, press **F5** to start the data conversion.

**F8**

**Cancel:** Use **F8** to cancel the current operation, and to return to your starting situation.

Figure 7.-3:  
 NC Program  
 Conversion; specifying  
 source and target  
 program; prior to the  
 conversion, an existing  
 program is selected.  
 The name and  
 directory for the  
 program resulting from  
 the conversion (target  
 program) are entered.

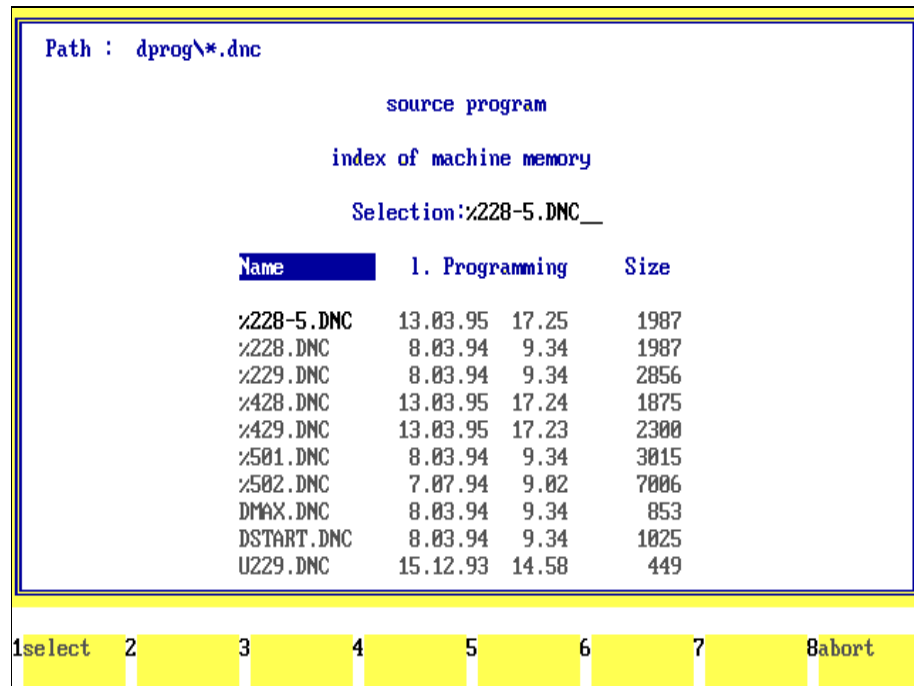
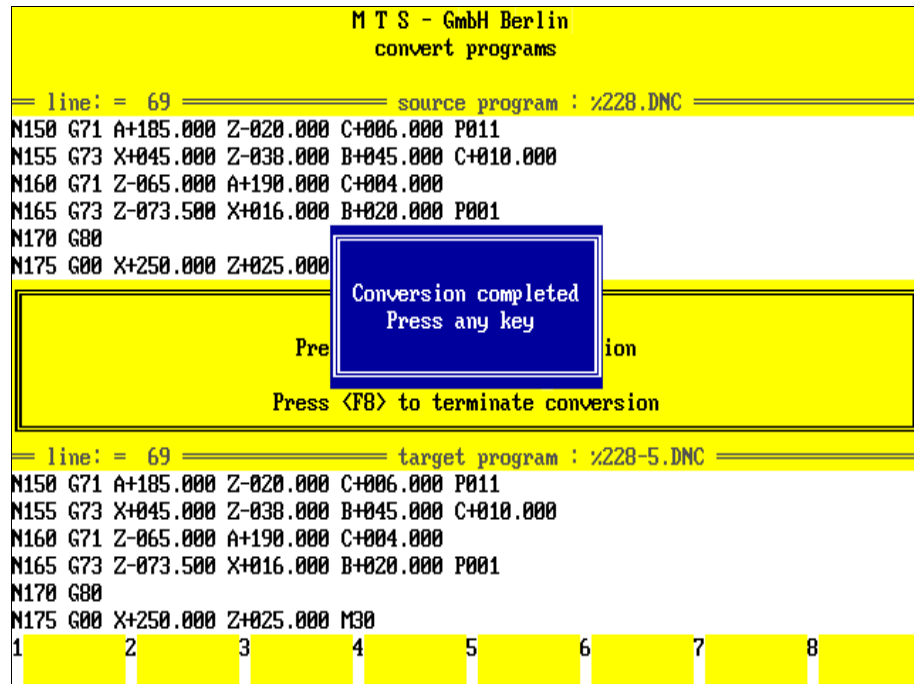


Figure 7.-4:  
 NC Program  
 Conversion;  
 during conversion the  
 source program is  
 displayed in the upper  
 screen section, while  
 the re-created  
 (converted) program is  
 displayed in the lower  
 screen section.





**Procedure**

Data conversion takes place automatically. You only need to select the NC program to be converted (source program), to specify the program name and, if applicable, the directory for the new NC program with the converted data formats. These definitions are made in a "file selection window" (see also [Chapter 2](#)).

**File Selection**

The file selection windows for the selection of the source program and the identification of the target program have analogous lay-outs. The display shows all files located on the currently active path. For your control, the previously selected operation mode is also indicated above the list of NC-programs.



Use the **TAB** key to activate the **directory display** (Path: ...), and continue by entering the desired directory name, including the global (wildcard) file identifier. **The same directory must be selected for the source and target program.**



If you confirm the selected directory by pressing **Enter**, the cursor moves back into the selection field, while the programs appear in the file list according to the indicated path definition.



and



To change the **sort order criterion**, press the combination of **Shift** and **Right Arrow** keys. This causes the highlighted selection bar to move to the next field, and a new sorting of the file list.

**[Enter Program Name]**

Enter the **name** of the desired file in the selection field. You can also select the file by using the cursor keys:



or



To choose a file from the list, select it with the **Up** or **Down** cursor keys. The filename is then automatically written in the selection field.

If the next higher and lower directory levels are displayed in the file list, you can change directories by selecting the desired directory in the list, and confirming it with the **Enter** key.



or



**Cancel:** Use **F8** or **ESC** to cancel the file selection, and to return to your starting situation.



or



**Select:** After you have entered or selected the program name, confirm it by pressing **F1** or **Enter**. You then return to the conversion program menu, with the file being displayed as a source or target file.



**Start Conversion:** After you have specified the source and target file, start the data conversion run by pressing **F5**.

In the event that the filename of the target program is identical with the name of an existing program, you receive a **control enquiry**: "Target file exists already! Do you wish to replace it with the conversion? (Y/N)":



or



Enter **Y** to overwrite the existing file, or **N** to cancel the operation.

**Note**

All programs created with a previous version of the CNC Simulator are compatible with the present version. Since Version 5 inserts a so-called checksum into the program, older programs must be converted, even if the data formats are identical. The checksums are automatically assigned, and constitute error checking routines to guarantee the files are complete.

Figure 7.1-1:  
Turning Control  
Configuration;  
programming code  
default settings.

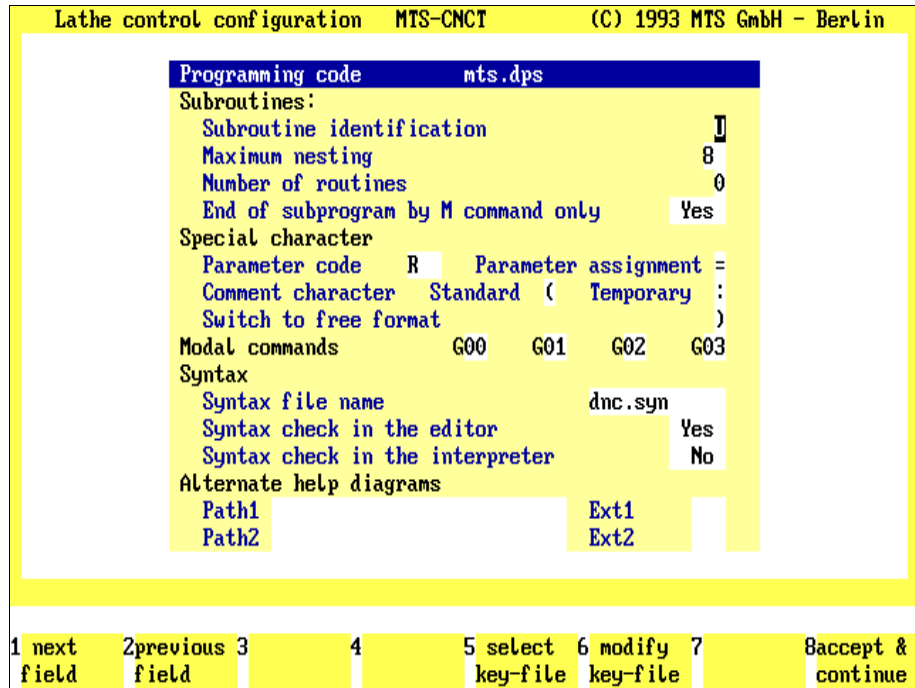


Figure 7.1-2:  
Milling Control  
configuration;  
NC Command  
Interpretation default  
settings.

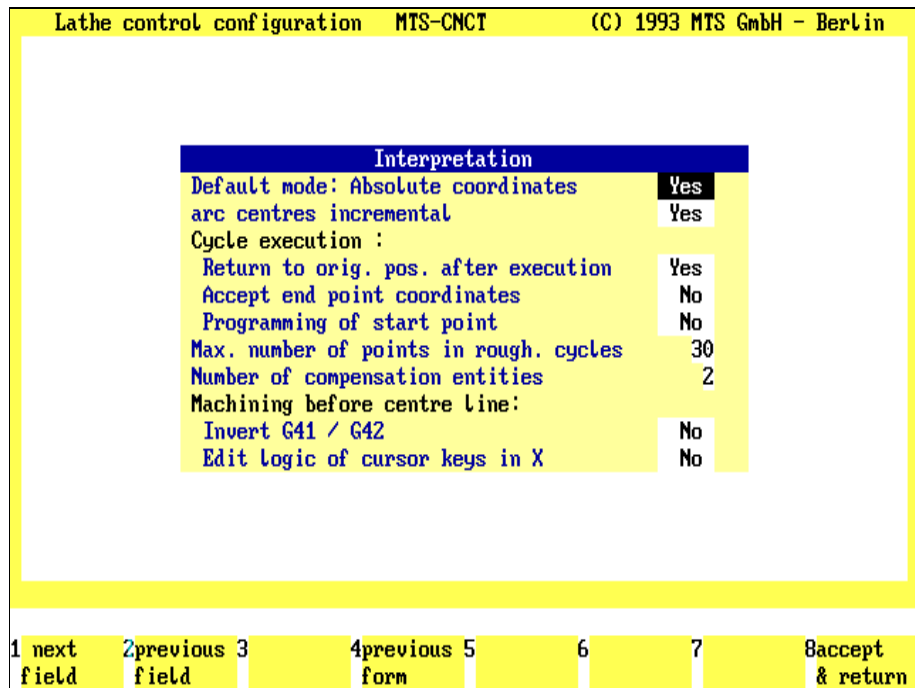
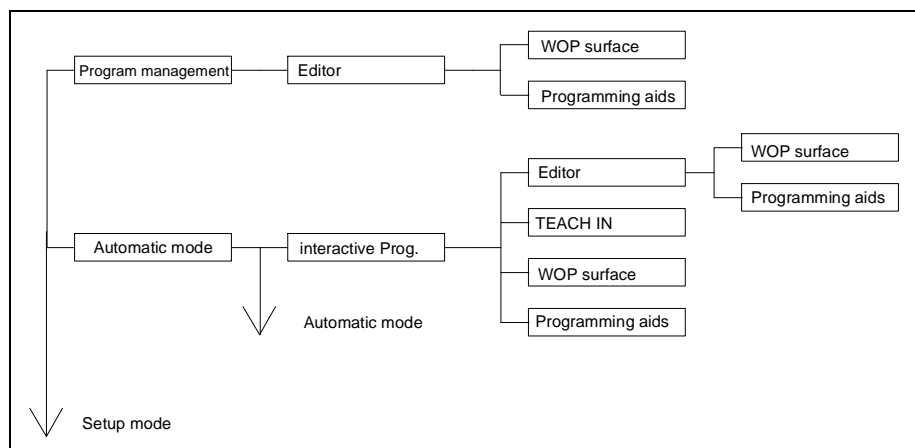


Figure 7.1-3:  
schematic of CNC  
Simulator  
programming options.



## 7.1 NC Editor

The Editor is a **programming interface** especially designed for NC programs. It allows quick and easy entry or editing of NC data blocks. Each NC word has its fixed format defining the allowable number of digits before and after the decimal point, and whether a sign may be prefixed or not. By editing the format file, these format definitions can be freely specified by the user. In addition, block numbers are assigned automatically, and it is possible to renumber NC programs or program segments, to change, delete or insert NC words, blocks and block sequences, and even to copy entire NC programs to any location of the current NC program, etc.

### Syntax Checking

During program generation, the Editor has an important function as an error detector. The NC data blocks undergo a line-by-line syntax check, i.e. a check of the formal format of the NC data blocks. If the NC data block is syntactically incorrect the message "Incorrect Syntax" appears in the header, and the corresponding line is highlighted in red.

### Value Ranges

Additional Editor functions are used for checking applied value ranges, as well as for sorting NC words.

If a range violation (according to the format file definitions) is found, it is marked with the pound sign "#" in the program line. This is a convenient function for checking and, if applicable, correcting technology data, rpm values and feed rate data.

Moreover, NC words within a program line are automatically sorted according to DIN standard 66025, or according to a user-defined default sequence.

### Free Format

In addition to the functions described above, the Editor can also be used in the so-called Free Format Mode. This mode applies no syntax checking, and you can enter any characters or character strings (see also [section 7.1.6](#) below).

The Free Format Mode is activated during configuration.

### Note

#### Selection:



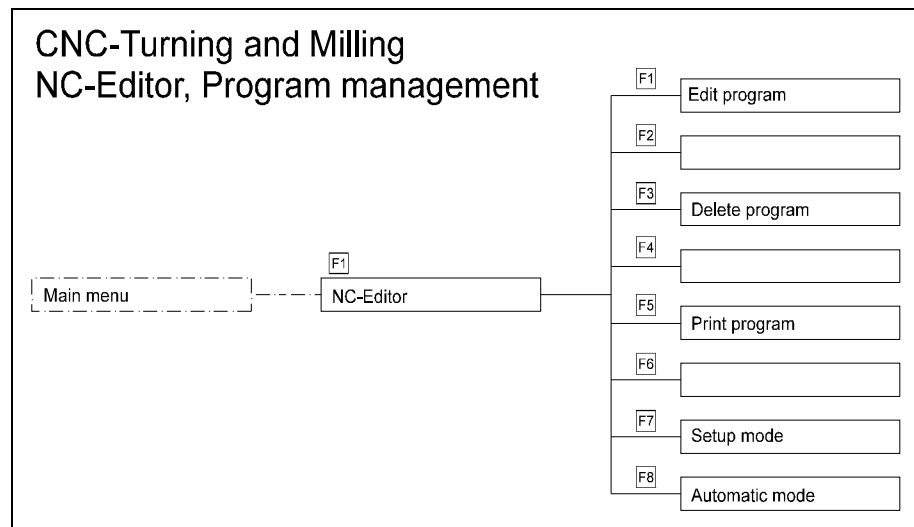
Proceeding from the main menu, the Editor is called by pressing **F1** (on the PC keyboard).

This causes the internal program management menu (see next page) to be loaded to allow NC program selection.

#### Cross Reference

Refer to the Configuration Instruction Manual for information on default values.

Figure 7.1.1-1:  
CNC Simulator,  
NC Editor; Program  
Management menu  
schematic.



Directory path

Cur. proces. funct.

Input field

File information

Available program  
files

Figure 7.1.1-2:  
CNC Simulator,  
NC Editor, Program  
Management

**Path :** dprog\\*.dnc

**PROGRAM SELECTION**

Index machine memory

**Selection:** %228.DNC

Name	1. Programming		Size
%228.DNC	10.05.94	9.10	1984
%229.DNC	8.03.94	9.34	2856
%501.DNC	10.05.94	9.14	3008
%502.DNC	2.07.94	16.46	5742
DMAX.DNC	8.03.94	9.34	853
DSTART.DNC	8.03.94	9.34	1025
HANS1.DNC	2.07.94	17.46	3018
HANS2.DNC	2.07.94	17.17	7293
U229.DNC	15.12.93	14.58	449

1 edit program
2
3 delete program
4
5 print program
6
7 setup mode
8 automatic mode

## 7.1.1 NC Program Management

As indicated, the Editor includes a simple **NC program management** function to perform the necessary steps for data management without prior knowledge of the operating system. A corresponding separate menu is available; it is loaded after activating the Editor (see figure on facing page).

### Screen Layout

The screen layout of the Program Management is slightly different from other user interfaces (see figure on facing page):

**The main screen area includes a file selection window to select the NC program. It displays all programs available on the currently active path. For better orientation, the operational function previously selected in the Program management menu, is repeated above the window.**

As always, the function keys at the bottom of the screen are labeled with their respective functions:

F1

**Edit Program:** This function allows the generation or modification of the currently selected NC program; press **F1** to load the selected NC program, and to open it for subsequent editing.

F3

**Delete Program:** With this function it is possible first to select and then to delete NC programs no longer required.

F6

**Print Program:** To print an NC program, select the desired program in the file selection window, and activate the print function by pressing **F6**.

F7

**Setup Mode:** From the NC Program management menu you can select the Setup Mode by pressing **F7**.

F8

**Automatic Mode:** Press **F8** to switch to the Automatic Mode.

ESC

Use **ESC** to return to the CNC Simulator main menu.



**To edit NC programs, you first specify the NC program to be edited, and then select the desired editing function:**

### Selecting NC Program: Selection Window

#### Directory Path

The NC programs created and modified with the CNC Simulator are saved in the directory specified for this purpose during the software configuration. During the software installation, the default directories `.DPROG` and `.FPROG` were created for turning programs and milling programs, respectively.

After opening the file selection window, one of these directory paths appears as a default (top left). A global file extension, i.e., wildcard characters "\*" or "?" is also required here. As a rule, turning programs are saved with the file extension `.DNC`, and milling programs with the extension `.FNC`.

**Examples:**

- 1.) Based on the standard conventions of the MS-DOS operating system, all files bearing the extension .FNC can be listed by adding a wildcard character, i.e., by typing \*.FNC. If we assume that .\FPROG is the path name for the subdirectory, then the correct path name for listing all milling files is **.\FPROG\\*.FNC**.
- 2.) Alternatively, if you wish to list all files stored on a floppy disk in drive A: in the list window, your correct path statement would be **A:\\*.\***.



[Directory Path]

To change the directory, press the key.

This opens the input field for the path name, and you enter the desired directory, together with a (wildcard) file extension.



If you continue by confirming the entered path name by pressing , the cursor returns to the selection field, and the programs are listed in the file list according to their pathname definition.

**Note**

**The selection of the path name can be blocked with a corresponding entry in the configuration. If such an entry has been done, a subsequent change of the directory name during program execution is not possible.**

In a similar way, the default filename extensions for the NC programs in the CNC Simulator can be defined by the user. The standard defaults are .DNC and .FNC (see also Configuration Instructions).

**Program Files**

Each program file is identified by name, date, time, and file size in byte (number of characters). Simultaneously, after entering a list criterion, the resulting list is sorted alphabetically or numerically, based on one of the above sort criteria. The selected sort criterion is highlighted.



and

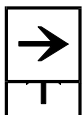


To change the sort criterion, press the + key combination. This moves the cursor into the next field, and the resulting re-sorted program list will be displayed.

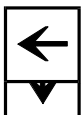
**Program Selection**

Enter the **name** of the NC program to be edited in the selection field. The selection window provides additional user functions to simplify file selection: If you know the program name, type it in the input field.

[Enter Program Name]



or



Within the selection field you move the cursor with the and cursor keys.

or

To select a program file from the file list, move the cursor to the filename by using the or cursor keys. This automatically transfers the program name marked by the cursor into the selection field.



or



If the program list is long, you can use the or keys to scroll through the file list.



Use to cancel the file selection.



**After you have entered or selected the correct program name, you can continue by selecting the desired Edit, Delete or Print function:**

## Program Management User Dialogues

### Edit Existing NC Program

Once you have chosen a program name in the selection window and confirmed it by pressing , the NC program is loaded and displayed in the Editor.

### Create New NC Program

To create a new NC program, enter the **name of the new program** in the file selection window and confirm it also by pressing the  function key. If no file with this name has been stored, the following message is displayed:

```
Program does not exist !
Create program ? (Y/N)
```

If you confirm the prompt by pressing , a new program with this name is created.

The same message also appears if you wish to select an existing program, but make a typing error when entering the file name.

If this occurs, respond by pressing  and retype the filename correctly.

### Delete NC Program

If you wish to delete an NC program, and you have selected the **desired program name** in the file selection window and confirmed it by pressing , the final control enquiry appears, for example:

```
Do you really wish to delete
.\fprog\EXAMPLE.FNC ? (Y/N)
```

or

To delete the selected program, press ; press  to cancel the operation.

### Print NC Program

If you wish to print an NC program, and you have entered the **name of the desired program** and confirmed it by pressing , you receive a corresponding message, for example:

```
Printing .\fprog\%220.FNC
... please wait
```

If the printer is not switched on or not properly connected, or incorrect information regarding interface, printer type or printer driver was entered during configuration the corresponding message is displayed, for example:

```
The printer is not ready !
Retry printing? Y/N
```

or

To restart printing, confirm the query by pressing ; use  to cancel the operation.

If no data can be sent to the printer the message: "**Printer does not respond**" is displayed. Confirm the message by pressing any key and you will return to the program management menu.

Figure 7.1.2-1:  
CNC Simulator,  
NC Editor;  
menu sequence  
schematic.

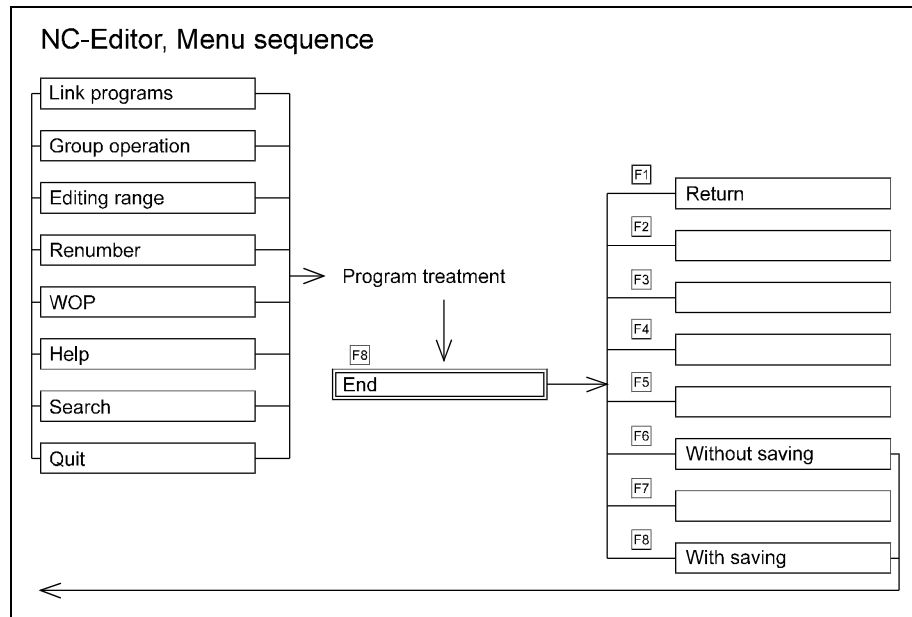
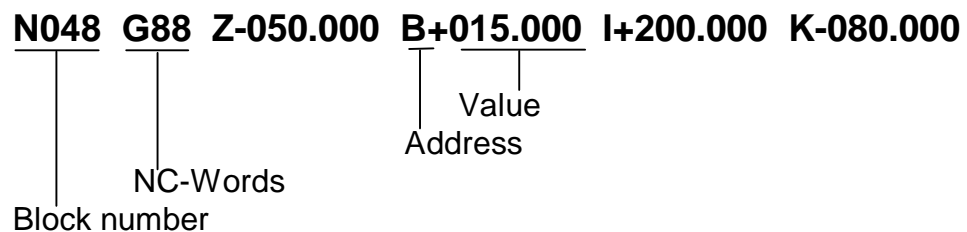


Figure 7.1.2-2:  
CNC Simulator,  
NC Editor  
programming  
interface; the two  
double lines mark the  
editing range: the  
marked program  
line(s) can be edited.  
The header contains  
also the program name  
and the syntax  
checking message.

```

M T S - GmbH Berlin
Program name : %502.DNC          Syntax : correct
=====
N025 F000.200 S1250 T1313 M03
N030 G00 X+000.000 Z+006.000 M08
N035 G01 Z-006.500
N040 G00 Z+006.000
N045 G26
N050 F000.200 S0750 T1212 M03
N055 G00 X+000.000 Z+002.000
N060 G04 Z-050.000 K+020.000 A+000.500 B+000.500 D+010.000
N065 G26
N070 F000.200 S0600 T1111 M03
N075 G00 X+000.000 Z+005.000
=====
N080 G04 Z-034.000 K+020.000 A+000.500 B+000.500 D+010.000
=====
N085 G26
N090 G92 S2200
N095 G96 S0220
N100 F000.750 S0100 T0505 M04
N105 G00 X+105.000 Z+000.500
N110 G01 X+022.000
1Link 2 group- 3 editing 4renumber 5 WOP 6 help 7 search 8 save/
Programs operation range quit
    
```

Figure 7.1.2-3:  
Structure of an NC  
data block (see also  
Programming  
Instructions):





## 7.1.2 NC Editor Programming Interface

The programming interface of the NC Editor is used to edit NC programs. It is a convenient support when entering NC data blocks: NC words are automatically formatted during the data entry. (For example, if you enter "x1", the string "'X+001.000", including the space character in front of this string, is entered in the NC program.) You can also insert, move or delete NC words, blocks and lines, and you can overwrite value statements or addresses (identification letters) of the same format without difficulty. In addition, you can insert other NC programs in any desired location of the NC program you are currently editing, etc.

### Error Checking

The **format** of the NC words (number of places before and after the decimal point, permitted signs) and the **value range** of the addresses (minimum/maximum values) are set as defaults as part of the control configuration. During programming the Editor performs a **running check** on whether these defaults have been considered. The incorrect statements are marked with a pound sign (#). With this function the RPM and feedrate can be checked and corrected if necessary. Moreover, the error function monitors the complete and correct entry form of the NC data blocks. An incorrect NC data block is highlighted in color, and the message, "**Incorrect syntax**" appears in the header.

The NC data block numbers are assigned automatically. However, NC programs or specified program segments can be easily renumbered. Finally, the NC words are sorted according to the DIN 66025 standard.

### Note

If NC programs are to be entered without syntax checking, automatic formatting, etc., the **Free Format Mode** can be used. This allows the entry of any characters and character strings. The Free Format Mode is selected either as a default for the entire edit operation, or it can be activated for single NC lines by pressing the ")" (closing round bracket) character.

### Screen Display

The illustration on the facing page shows an example of the screen display of the programming interface:

The program name and the error message are displayed in the header.

The NC program is displayed in the area below. If the NC programs are extensive, it is possible to scroll up and down the entire program NC data block by data block. If a line stretches beyond the screen display area, its continuation is indicated by a "" or "" character. In this case, the screen display can also be shifted to the right or left.

Finally, the **editing range** is framed by double line. Its position and size are set during configuration, they can, however, be changed during program editing.

Figure 7.1.2-4:  
CNC Simulator,  
NC Editor  
Programming  
Interface;  
as an alternative to the  
text labeling of the  
function keys menu  
operations can also be  
indicated with CNC  
symbols and icons.

```

M T S - GmbH Berlin
Program name : z501.DNC          Syntax : correct
N055 G96 F000.800 S0120 T0404 M04
N060 G00 X+105.000 Z+000.500
N065 G01 X+000.000
N070 G00 Z+005.000
N075 G00 X+105.000
N080 G57 X+000.500 Z+000.500
N085 G81 X+000.000 Z+002.000 I+005.000
N090 G01 Z+000.000
N095 G71 X+034.000 Z+000.000 C+002.500
N100 G71 A+175.700 Z-043.000
N105 G71 X+052.000 A+090.000 C-002.000
N110 G71 X+052.000 Z-054.000 C+003.000 P011
N115 G71 Z-054.000 X+066.000 C-002.000
N120 G71 Z-086.000 X+066.000
N125 G71 X+096.000 Z-086.000 C-002.000
N130 G71 X+096.000 Z-095.000
N135 G71 Z-095.000 X+105.000
N140 G80
N145 G26
N150 F000.200 S0200 T0202 M04
N155 G42
N160 G00 X+000.000 Z+005.000
N165 G23 P090 Q135

```

Assignment of CNC  
symbols to the  
program functions (see  
also Appendix).

CNC-Keyboard	PC-keyboard	Key-function
	<b>F1</b>	Link programs
	<b>F2</b>	Group-operation
	<b>F3</b>	Change editing range
	<b>F4</b>	Renumber
	<b>F5</b>	WOP-surface
	<b>F6</b>	Helpscreen
	<b>F7</b>	Search string
	<b>F8</b>	Quit

As usual, the function keys at the bottom of the screen are labeled with their respective menu functions:

**F1**

**Link Programs:** With this function you can copy another NC program into the current program.

**F2**

**Block Group Treatment:** If you wish to move, copy or delete one or several NC data blocks as a group, press **F2** to access the Block Group function.

**F3**

**Change Edit Range:** To change the beginning and/or end of the edit range and the number of NC data blocks which it can contain, select **F3**.

**F4**

**Re-numbering:** To change the NC data block numbering, select **F4**.

**F5**

**WOP:** Use **F5** to switch to the Workshop Oriented Programming interface.

**F6**

**Help:** To consult one of the Programming Aids functions, select **F6**. (In the Editor, the familiar help access with the <?> key is disabled.)

**F7**

**Search:** You can use this function to search for a specific character string (NC word, address, etc.)

**F8**

**End:** Use **F8** to end program editing.

### CNC Symbols

As an alternative to the text labels of the function keys, the CNC Editor can indicate the functions with CNC symbols and icons. However, depending on the processing power of the PC, the reinstatement of the screen lay-out and scrolling through the NC program may take considerably longer. This display mode is set during configuration.

### Note

To be able to describe better the program modification by means of the NC Editor, the applicable user functions are divided into two groups:

- 1.) The first group, called "**editing functions**", refers to the different editing functions for an NC program, such as entry, insertion, deletion, etc.
- 2.) The second group refers to the **supplementary program functions**, such as changing block numbers, accessing Programming Aids or search functions, etc.. These are selected with the function keys on the PC keyboard.

Since these operating function groups are used together, their application has to be learnt for the manual programming with the NC Editor.



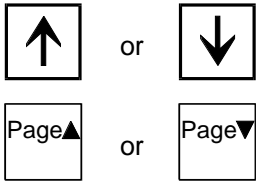
**When creating your NC program try to build a clear program structure and, if possible, to give helpful comments for the NC data blocks.**

If this is your first work with the Editor, you should try the entry functions first. Also, note that the Free Format Mode does not feature automatic character formatting, but that all characters (including spaces, etc.) need to be entered.

### 7.1.3 Editing Functions

#### Position Program Line in Editing Range

To edit a programming line, it must first be placed (scrolled) into the editing range. **The editing range is marked by the two double lines.**



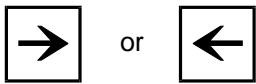
An NC data block is positioned in the editing range by using the and/or cursor keys for scrolling up and/or down through the entire program. This enables you to place any programming line into the editing range.

Since larger programs may require considerable time for scrolling, you can use the and keys for page-by-page browsing through the program.

#### Entry Routine

For improved convenience in NC programming, a new entry routine was developed that makes the entering of NC data blocks and comments faster and simpler than before. This means that two entry modes, the **New Entry Mode** and the **Overtyping Mode**, are automatically suggested:

The **New Entry Mode** is used to write new NC words or to replace value statements, while in **Overtyping Mode** single figures or addresses are changed.



The switch between both entry modes is made with the and cursor keys, and the respective entry mode is indicated by the coloured highlighting and the cursor position, respectively.

#### [New Entry Mode]

The entry routine is determined after the entry of a new address letter or, while in New Entry Mode, after moving the cursor from one address letter one column to the right. Any additional cursor movement automatically effects the switch to Overtyping Mode.

During New Entry Mode, the entire input field is marked with coloured highlighting, and you can continuously enter the desired numerical values without leading zeroes, and with optional signs. The old numerical value is deleted with the entry of the first number, and is replaced by the new entry in the format corresponding to the address character. If a decimal point (or comma) is entered after the optional sign, only the digits after the decimal point are deleted and replaced by the numbers entered.

Example: For the NC data block in Figure 7.1.2-3 the following entry is sufficient:

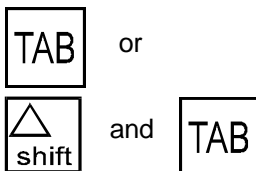
**g 8 8 i 2 0 0 k - 8 0 b 1 5 z - 5 0**

From this entry, the Editor automatically formats (and sorts) the correct NC data block:

**G88 Z-050.000 B+015.000 I+200.000 K-080.000**

#### [Overtyping Mode]

In Overtyping Mode, the cursor is located under a letter or number. To change a single value or an address letter, move the cursor to the required position and simply overwrite the present entry.



Pressing the key, or the key combination moves the cursor from word to word (and back) within an NC data block.



**To get some routine of NC word entry and Overtyping Mode, try using them!**

<b>Special Characters</b>	In addition to the entry of letters and numbers (including decimal point and positive/negative sign), the NC Editor provides various special characters for special entries:
<b>Comments</b>	The <> character (opening bracket) is used to indicate that the statement is a comment. The statements after the comment character are not interpreted as belonging to the programming sequence (exception: setup sheet, see below). A comment can be repeated as often as required. If the comment character is deleted, the entire comment is deleted as well.
<b>Setup Sheet</b>	The characters <()> (opening and closing brackets) are used to mark the start and end of the setup sheet. All other setup data is preceded by a comment character. Other setup sheet language conventions are discussed in <a href="#">Chapter 3</a> . During configuration you can specify if the setup sheets can be modified by the NC Editor. If the configuration defaults prevent setup sheet modification, this is indicated by the message, " <b>Setup sheet cannot be edited.</b> "
<b>Temporary Free Format</b>	The <> character (closing bracket) is used to select the Free Format Mode for individual programming lines. similar to the comment character, any characters may follow the 'switching' character. Although no syntax checking takes place, all characters are interpreted as belonging to the programming sequence. (see below, section 7.1.6). The line-by-line Free Format switching is used in the compact entry of NC data blocks containing many addresses. However, check that the entries make sense and can be executed by the program.
<b>Disable Interpretation</b>	The <:> (colon) character is used to disable the interpretation of the following NC words in the programming line, and can be inserted like a code letter at any location of an NC data block. A conditional disabling of the interpretation can be used to "remove" single NC words during a program test, without actually having to delete them. The character can be deleted, to enable the rest of the program line again.

### Concluding a Program Line

Conclude the entry of your NC data block as follows:



If the block is completely programmed, press to confirm it.



or



Another, even simpler way of closing a line is that of using or keys to move the cursor to the next or previous program line.

**After closing an NC data block, the NC words are automatically sorted according to the default settings saved in the format file.**






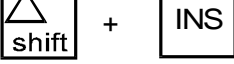














### Insert Character String

Version 5.2 of the CNC Simulator provides a total of four different insert modes: as editing functions the options of either inserting single NC words in an existing NC data block, or of adding new NC data blocks; as program functions the options of either copying several NC data blocks as a group (Block Group Handling), or of inserting complete NC programs.



To insert an NC word into a program, place the cursor on a code letter and press the key. This causes the following data to move one space to the right, and you can now enter the additional NC word.

## CNC Editing Function Symbols

CNC-Keyboard	PC-keyboard	Key-function
		Insert NC-Word / Number
		Delete NC-Word / Number
		Insert NC-Block
		Delete NC-Block
		Page down
		Page up
		Cursor down
		Cursor up
		Cursor right
		Cursor left



and



If you wish to insert an NC data block into the program, position the cursor on the desired block number and press the + key combination. This inserts a blank line which you can subsequently edit.

**Note**

To enter an NC block, the program is switched to the Insert Mode. The status message "Inserting" is displayed in the header. If you now confirm an NC block with , another NC data block is inserted, etc. To toggle the Insert Mode off, press the + key combination again.

**Delete Character String**

As in the Insert Mode, the Editor features four different Delete Modes: You either delete individual NC words or complete NC data blocks (Edit Function), or you delete several NC data blocks as a group (Block Group Handling), or a complete NC program (NC Program management).



To delete a number in an NC word, place the cursor in the desired position, and press the key.

Or, if you wish to delete a complete NC word, move the cursor to the first letter of the word and press the key.



and



To delete a complete NC data block in one step, press the + key combination.

Figure 7.1.4-1:  
CNC Simulator,  
NC Editor,  
Link Programs;  
Menu sequence  
schematic.

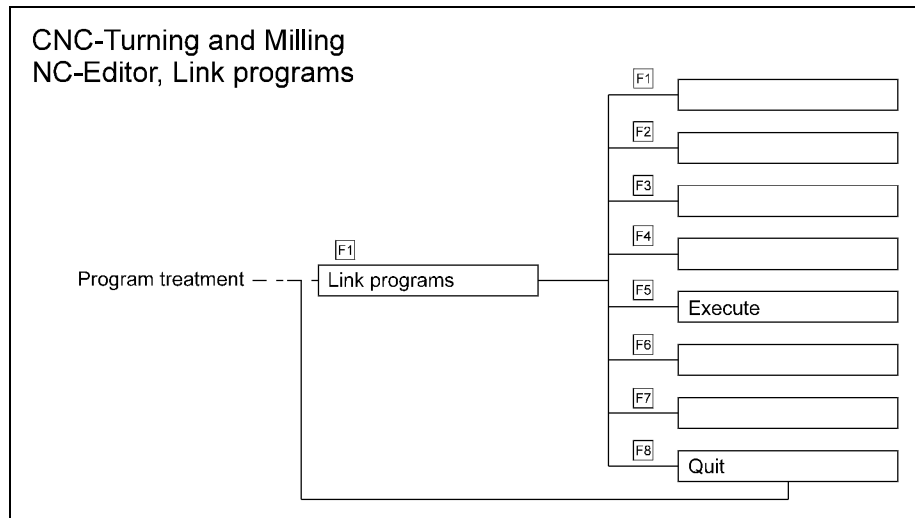


Figure 7.1.4-2:  
CNC Simulator, NC  
Editor,  
Link Programs;  
after using the cursor  
to mark the position for  
inserting the NC  
program, the Link  
Programs function is  
accessed by pressing  
[F1], and the name of  
the NC program to be  
inserted is entered.

Path : dprog\\*.dnc

**LINK PROGRAMS**

Index machine memory

Selection: %329.DNC

Name	1. Programming		Size
%228.DNC	11.03.95	17.04	1981
%229.DNC	8.03.94	9.34	2856
%328.DNC	11.03.95	17.04	1981
%329.DNC	8.03.94	9.34	2856
%501.DNC	10.05.94	9.14	3008
%502.DNC	2.07.94	16.46	5742
COMPVAL.DNC	11.03.95	17.22	1306
DMAX.DNC	8.03.94	9.34	853
DSTART.DNC	8.03.94	9.34	1025
HANS1.DNC	2.07.94	17.46	3018

N135 X+016.000 Z+001.000 F003.000 T0909 M04

1 2 3 4 5execute 6 7 8abort

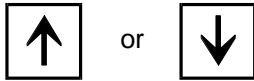


## 7.1.4 Program Functions

### Linking NC Programs



The Link Programs function can be used to insert a second complete NC program into the current program.



or

You first move the cursor prior to the line the program is to be inserted.

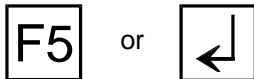


**Link Programs:** Continue by pressing **F1** to select this function.

[Enter Program Name]

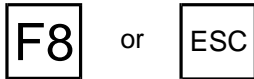
To enable you to determine which of the available NC programs to insert, the file selection window is opened again. Enter the name of the program to be inserted in the input field.

You can also use the cursor to select the desired program file (see above).



or

**Execute Insert:** Press **F5** or **←** to insert the NC program.



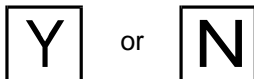
or

**Cancel:** Press **F8** or **ESC** to cancel the operation.

Before the second program is inserted, a control enquiry is displayed:

```

About to copy
"EXAMPLE.FNC"
Do you wish to proceed?(Y/N)
  
```



or

To execute the linking/inserting function, confirm it by pressing **Y**. Press **N** to cancel the operation.

Figure 7.1.4-3:  
CNC Simulator,  
NC Editor,  
Block Group Handling;  
Menu sequence  
schematic.

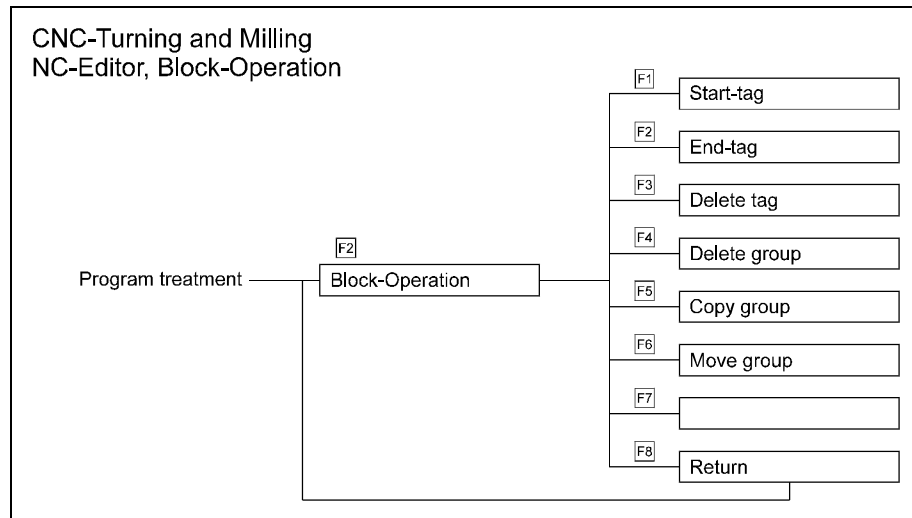


Figure 7.1.4-4:  
CNC Simulator,  
NC Editor,  
Block Operation;  
selected block in an  
NC program.

```

M T S - GmbH Berlin
Program name : %502.DNC          Syntax : correct
N050 F000.200 S0750 T1212 M03
N055 G00 X+000.000 Z+002.000
N060 G04 Z-050.000 K+020.000 A+000.500 B+000.500 D+010.000
N065 G26
N070 F000.200 S0600 T1111 M03
N075 G00 X+000.000 Z+005.000
N080 G04 Z-034.000 K+020.000 A+000.500 B+000.500 D+010.000
N085 G26
N090 G92 S2200
N095 G96 S0220
N100 F000.750 S0100 T0505 M04
N105 G00 X+105.000 Z+000.500
N110 G01 X+022.000
N115 G00 Z+002.000
N120 G00 X+105.000
N125 G57 X+000.500 Z+000.500
N130 G01 X+022.000 Z+002.000 I+005.000
N135 G71 A+180.000 Z+000.000
1 start- 2 end- 3 delete 4 delete 5 copy 6 move 7 8 quit
  tag   tag   tags  group  group  group  quit

```

## Block Group Handling



In the version 5 of the CNC Simulator it is possible to move, copy or delete NC data blocks as a group.

**F2**

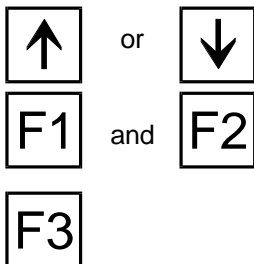
**Block group handling:** Start this editing function by pressing **F2**.

This loads a submenu with the following options:

- 1.) Define the desired NC data blocks as a block group.
- 2.) Specify the location to which the block group is to be moved or copied.
- 3.) Choose and activate the respective function (delete, copy, move) for the block group handling.

### Define Block Group

To define a block group, first mark the NC data blocks which are to be handled as a block group:



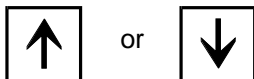
Place the cursor always on the first and last NC data block of the desired block group.

**Start/End Mark:** Confirm the respective cursor position by pressing **F1** for the start, and **F2** for the end mark of the block group. The entire block group is then highlighted in colour.

**Delete Mark:** Use **F3** to delete again the start or end mark.

### Target Position

To move or copy the block group, you then define the NC data block prior to which the block group to be moved or copied is to be inserted (this step is ignored for deletion!):



As for marking of the first and last NC data block of a block group, place the cursor again in the desired position.

### Block Group Handling

As an editing option, the block group can be deleted, moved or copied:

**F4**

**Delete Block Group:** Press **F4** to delete all selected (marked) NC data blocks. There is no control enquiry!

**F5**

**Copy Block Group:** Press **F5** to copy the block group and insert prior to the NC data block that is marked by the current cursor position.

**F6**

**Move Block Group:** Press **F6** to move the block group and insert it before the NC data block that is marked by the current cursor position.

**F8**

**Return:** Use **F8** to return to your starting position.

### Note

Should you have specified a location within the marked block group as the target/insert position, the error message, "Block group handling not possible" is displayed. It is not possible to copy or move a block group into this same group.

Figure 7.1.4-5:  
CNC Simulator,  
NC Editor,  
Modify Editing Range;  
Menu sequence  
schematic.

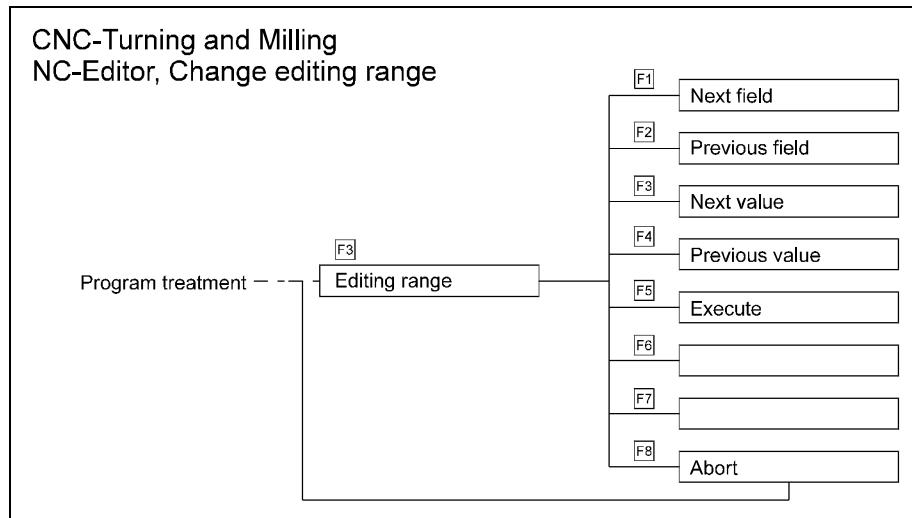
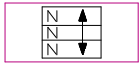


Figure 7.1.4-6:  
CNC Simulator,  
NC Editor,  
Modify Editing Range;  
enter the desired size  
of the editing range in  
the display window.

M T S - GmbH Berlin							
Program name : %502.DNC				Syntax : correct			
<pre> N060 G84 Z-058.000 K+020.000 A+000.500 B+000.500 D+010.000 N065 G26 N070 F000.200 S0600 T1111 M03 N075 G00 X+000.000 Z+005.000 N080 G84 Z-034.00 N085 G26 N090 G92 S2200 N095 G96 S0220 N100 F000.750 S01 N105 G00 X+105.00 N110 G01 X+022.00 N115 G00 Z+002.00 N120 G00 X+105.000 N125 G57 X+000.500 Z+000.500 N130 G81 X+022.000 Z+002.000 I+005.000 N135 G71 A+100.000 Z+000.000 N140 G71 A+090.000 X+044.000 C-002.000 N145 G71 Z-025.000 X+044.000 </pre>							
<b>CNC-editor, configuration</b> Editing range beginning line: 12 No. of lines : 1 UP, DOWN: edit value; TAB: switch field							
1	2	3	4	5	6	7	8execute

## Modify Editing Range



On the screen, the editing range is marked by the two horizontal double lines; it identifies the part of the NC program that can currently be edited. Even if the size of the editing range is defined by a configuration default, it can also be changed during NC programming:

**F3**

**Editing Range:** To redefine the editing range, press **F3**.

This opens a dialogue window (see figure) in which you can enter new specifications to define the size and screen position of the editing range. The current configuration defaults specifying the editing range are already present in the dialogue window:

**Starting Line:** The number entered here determines the screen position for the first line of the editing range.

**Number of Lines:** This is the number of lines included in the editing range on the screen.

**TAB**



or



The entry which can now be changed, is highlighted in colour. Use the **TAB** key to select an input field.

Use the  or  cursor keys to change the field values.

**F8**

**Accept:** Press **F8** to accept your modifications.

**ESC**

Press **ESC** to cancel the operation.

Figure 7.1.4-7:  
CNC Simulator,  
NC Editor,  
Renumbering;  
Menu sequence  
schematic.

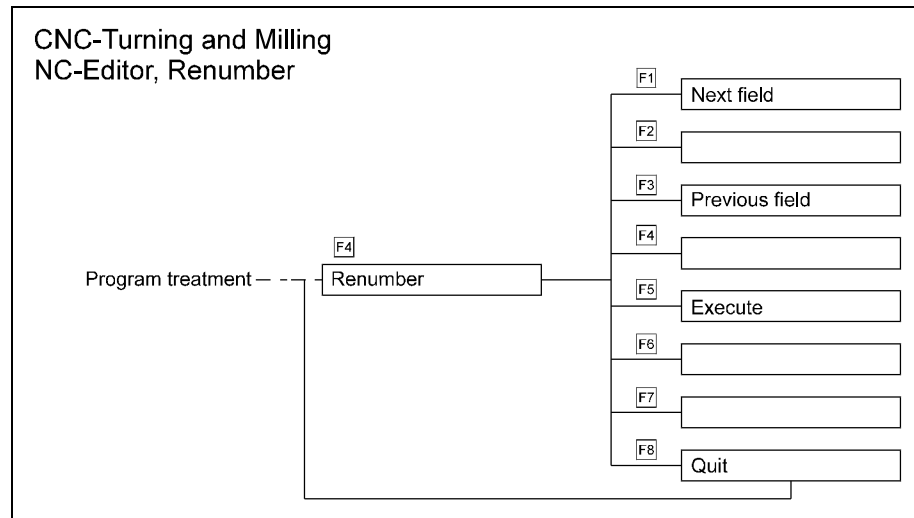


Figure 7.1.4-8:  
CNC Simulator,  
NC Editor,  
Renumbering;  
In the dialogue window  
you enter the range,  
the new starting block  
number and the  
numbering increment.

M T S - GmbH Berlin

Program name : %502.DNC Syntax : correct

---

```

N060 G84 Z-058.000 K+020.000 A+000.500 B+000.500 D+010.000
N065 G26
N070 F000.200 S0600 T1111 M03
N075 G00 X+000.000 Z
N080 G84 Z-034.000 K
N085 G26
N090 G92 S2200
N095 G96 S0220
N100 F000.750 S0100
N105 G00 X+105.000 Z
N110 G01 X+022.000

N115 G00 Z+002.000

N120 G00 X+105.000
N125 G57 X+000.500 Z+000.500
N130 G81 X+022.000 Z+002.000 I+005.000
N135 G71 A+100.000 Z+000.000
N140 G71 A+090.000 X+044.000 C-002.000
N145 G71 Z-025.000 X+044.000
        
```

renumber

from : 10

to : 999

start number : 10

increment : 1

TAB: switch field; ESC: abort

---


1      2      3      4      5      6      7      8execute

## Renumbering NC Data Blocks



The block numbers are automatically assigned during programming. You can change the numbering, of course. This is helpful especially when one or more NC data blocks are inserted into, or moved within a program.



**Renumbering:** Press  to define a new block numbering sequence:

This opens a dialogue window in which you can specify the desired sequence:


**From:** This number indicates the first block of the current program to be renumbered.

**To:** This number indicates the last block of the current program to be renumbered.

**Increment:** Numbering progress, i.e., with an increment value of "3", it is: 3, 6, 9, 12 ...




The entry which can currently be changed is highlighted in colour.

Use the  key to select an input field.

[Enter Value]

Enter the desired value in the active input field.



**Execute:** Having changed all values as desired, confirm them by pressing . This causes the renumbering to proceed from the first identified NC data block number.



Press  to cancel the operation.



The Renumbering function might be necessary if NC data blocks or block groups were moved, inserted or copied. These functions may cause duplications of NC data block numbers, possibly requiring one or more renumbering runs, if the NC data blocks must be numbered in linear succession.

In the case of duplicated block numbers, renumbering takes place from the first starting number found to the first ending number found.

## Switch to WOP Interface



As discussed earlier, if the Workshop Oriented Programming module was installed in your system, the WOP Interface can also be accessed during programming operations with the Editor. This change of mode facilitates the programming of complex contours, without any prior contour calculations.



**WOP:** Press  to switch to the Workshop Oriented Programming interface.

### Cross Reference

Detailed information on the WOP Interface is given in a separate chapter.

Figure 7.1.4-9:  
CNC Simulator,  
NC Editor,  
Find Character String;  
Menu sequence  
schematic.

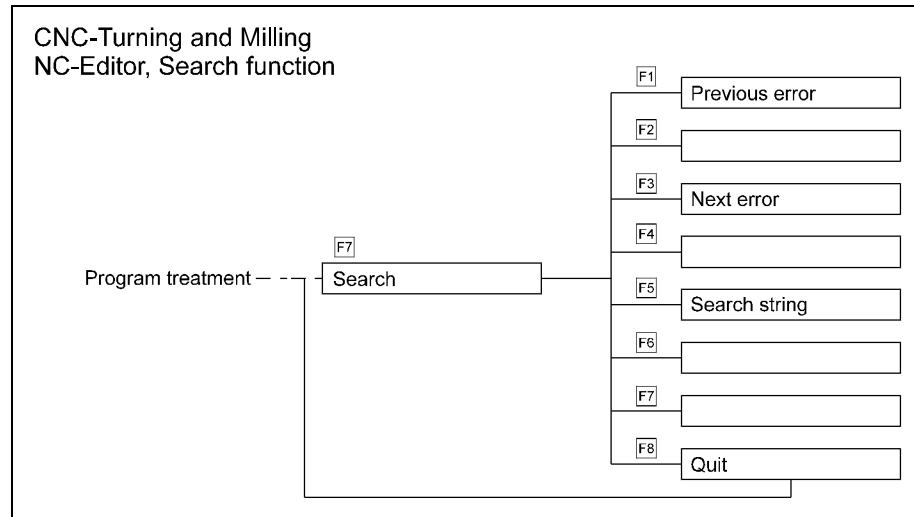


Figure 7.1.4-10:  
CNC Simulator,  
NC Editor,  
Find Character String;  
enter in the dialogue  
window the desired  
character string for  
automatic search.

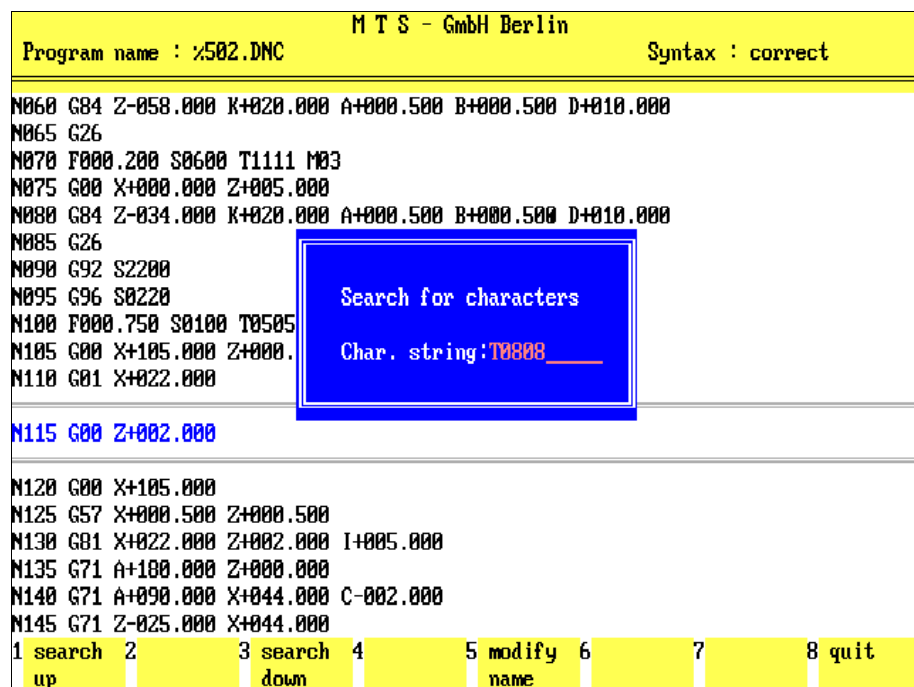
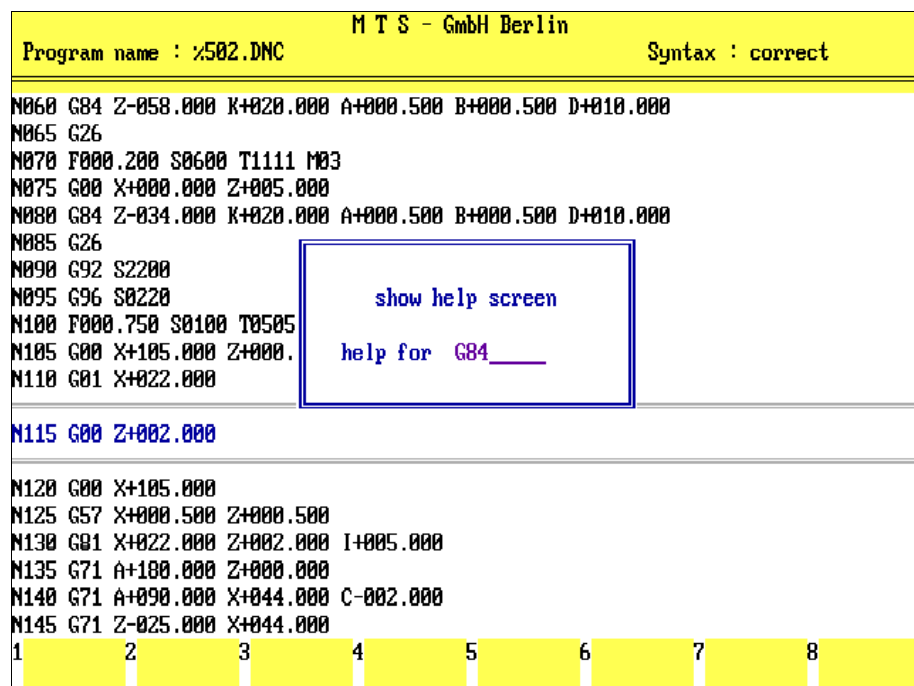


Figure 7.1.4-11:  
CNC Simulator,  
NC Editor;  
Accessing  
Programming Aids.






## Programming Aids



As in the other operating modes, all Programming Aids are also available during working with the Editor.



**Aids:** In the NC Editor, the Programming Aids are accessed by pressing the **F6** key (and not the  key!). Use the dialogue window to enter the name of the desired Aid or select it by means of the registers (see also [Chapter 11](#)).



or

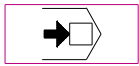


Press **F8** or **ESC** to cancel the help function.

**Note**

**In the same manner as the Programming Aids can be accessed from the Editor, an NC data block can be modified by calling the Editor when using Programming Aids.**

## Find Character String



A program modification often requires the changing of one single item only. With the search function you can quickly find such items. The search function checks whether a certain character string (such as one or several NC words, an NC data block or merely code letters or values) is included in the program, and displays it automatically. However, the maximum length of the specified character string is 10 characters.



**Find:** Press **F7** to start the search function.

This opens an intermediate menu with several search options:



or



**Previous/Next Error:** To search for a "tagged" error (see [Figure 7.1.5-1](#) next page) in the NC program, select **F1** or **F3**. If an error is found, it is displayed automatically.



**Find String:** Press **F5** to enter any type of "search string" (see below).



or



**Return:** Press **F8** or **ESC** to return to your starting situation.

### Find String

To check whether the specific string is included in the NC program, activate the Find String function to open a dialogue window in which you can enter the character string for the automatic search.

[Search String]

Enter the desired character string in the dialogue window, and check the correct spelling of the Search String.



or



**Search Up/Down:** If you confirm your entry by pressing **F1** or **F3**, the program searches for the search string before or after the current editing line, respectively. If the string is found, it is automatically displayed. If it is not found the message "Search string not found" is displayed

If the character string occurs several times in the program, repeat this function.



**Change String:** If you wish to search for a further string, select **F5** and repeat the procedure described above.



**Return:** Press **F8** to return to your starting situation.

Figure 7.1.4-12:  
CNC Simulator,  
NC Editor;  
Ending Programming.

M T S - GmbH Berlin							
Program name : %229.DNC				Syntax : correct			
N165 X+035.000 Z+005.000							
N170 G31 X+030.000 Z-026.000 A+030.000 D-002.147 F003.500 S0005							
N175 G00 X+150.000 Z+100.000							
N180 T0800 M04							
N185 X+052.100 Z-043.600 F002.000							
N190 G57 X+000.000 Z+000.000							
N195 G03 X+052.100 Z-063.600 K+010.000							
N200 G79 X+047.000 Z-064.000 I+001.000 K+001.000 A+001.000 W+001.000 P110 Q110							
N205 G00							
N210 (							
N215 G00 X+100.000 Z+050.000 M30							
N225							
1 quit	2	3	4	5	6 abort	7	8 save/ quit

Figure 7.1.5-1:  
NC Editor  
Syntax Checking and Range  
Violations:  
NC data block "N110"  
contains two incorrect  
statements: By mistake  
feedrate was programmed  
F:000.000 and toolchange  
"T0000". The not acceptable  
values are automatically  
tagged with the "#" character.  
Also, NC data block "N135"  
contains an incorrect word  
combination. The syntax  
checking function highlights  
the line with colour, and also  
displays the message,  
"Incorrect syntax" in the  
header.

M T S - GmbH Berlin								
Program name : %228.DNC				Syntax : incorrect				
N020 (								
N025 G54 Z+251.500								
N030 G90 S1500								
N035 G96 S0300								
N040 G00 X+100.000 Z+050.000								
N045 (								
N050 ( PREPARING THE PART -								
N055#F000.000#T0000 M04								
N060 G00 X+110.000 Z-000.500								
N065 G01 X-002.000								
N070 G00 X+100.000 Z+050.000								
N075 X+000.000 Z+002.000 T1313 M03								
N080 G97 F000.800 S3000 G01 Z-005.000								
N085 G01 Z-000.000								
N090 G00 Z+160.000								
N095 (								
N100 ( DEEP BORING CYCLUS								
N105 X+000.000 Z+001.000 T1515 M03								
1Link	2 group-	3 editing	4renumber	5	WOP	6 help	7 search	8 save/ quit
Programs	operation	range						

## Conclude NC Programming



You can conclude your programming session with or without saving your data:

Press **ESC** to return directly to the Program Management menu, and to save simultaneously your programming results.

**With the **F10** function key you return directly to the Program Management without saving your programming data.**

**End of Job:** At the same time it is possible to conclude your work via program dialogue: press **F8** to activate this dialogue sequence:

**Return:** If you have mistakenly started the End of Job function, you can return to program editing by pressing **F1**.

**Without Saving:** Press **F6** to cancel program editing, without saving the most recent editing results, and you return to the Program Management menu.

**Caution!** After cancelling, the previous editing results are no longer available for you.

**Save Data:** Normally, you conclude your NC programming session by saving your data. Press **F8** to conclude your session and to save your data. **You** then return to your starting point, the Program Management menu.

## Return to Setup or Automatic Mode

From the Program Management menu, you can access the Setup Mode by pressing the **F7** key, or the Automatic Mode by pressing **F8** (see above).

### 7.1.5 Syntax Check and Range Violation

As already described, the Editor automatically performs a syntax and value range check during programming.

If you enter and confirm an NC data block which violates the set file format the incorrect data block is highlighted in color, and the message, "Incorrect Syntax" is displayed in the header.

Also, in case of value range violation the incorrect value is tagged with the character "#".

The figure on the facing page shows examples of these functions.

#### Procedure

As soon as one of these error messages occurs, check the program, and correct the incorrect value statements or addresses, etc.

#### Note

Refer to your programming instructions for information on correction procedures. You can also consult a Programming Aid.

Figure 7.1.3-1:  
CNC Simulator,  
NC Editor;  
programming in Free  
Format Mode.

M T S - GmbH Berlin	
Program name : %229.DNC	Syntax : OFF
<pre> N10 G54 N20 G90 F4. S1600 M4 N30 G0 X120. Z120. N40 G0 X98. T4 D4 N50 G0 Z0 N60 G96 S150 N70 G1 X-2. N80 G0 Z5. N90 G0 X120. Z50. N100 G0 X120. Z120. N110 G0 X95. Z0.5 T12 D12  N120 R20=229 R21=30. R22=0.5 R24=1 R25=0.5 R26=5. R27=0 R28=4  N130 R29=31 L95 P1 N140 G0 X100. Z20. N150 G0 X120. Z120. N160 F0.2 S1500 T5 M4 D5 N170 G96 S300 N180 G0 X35. Z0 </pre>	
1Link	2 group-
3 editing	4renumber
5 WOP	6 help
7 search	8 save/
Programs	operation
range	quit

## 7.1.6 Free Format Mode

In addition to the functional modes described the Editor can also be used in the so-called Free Format Mode. **This mode uses no syntax checking, and you can enter any character or character strings.** The Free Format Mode is switched on during configuration (see also Configuration Instructions).

### Procedure

Basically, the programming functions featured in the Editor (Editing Range, End-of Line, Find Character String, etc.) are handled in Free Format Mode in the same way as under the MTS Programming Code.

To program (editing) NC data blocks you enter directly addresses and values; to correct them you overwrite them and/or insert new data. At the same time, the cursor can be freely positioned within the editing range, and existing characters as well as empty input fields are overwritten by the entries.

However, keep in mind that the Free Format Mode does not provide automatic character formatting, that all characters (including empty spaces) must be entered separately, and that all editing functions must be executed individually.

For example: The NC data block

**G88 Z-050.000 B+015.000 I+200.000 K-080.000**

can be programmed in Free Format Mode as follows:

**G88 Z-50 B+15 I+200 K-80**

### Note

Also consider, that an NC program made in Free Format Mode cannot be loaded by the Editor during standard editing or in PAL Mode. Due to the fact that the code letter formats are, as a rule, different, a correct program run is not guaranteed. Therefore, prior to starting the CNC Simulator, it is recommended that you either check the configuration defaults, or adjust the data formats of the NC program to correspond to the MTS Programming Code by means of the Format Conversion utility (see above).

## Temporary Free Format

As described earlier in this chapter, the Free Format Mode can also be selected line by line with the <> (closing bracket) character for the purpose of single-line editing, in order to allow condensed entry of NC data blocks containing multiple addresses. In the same way as after the comment character this mode allows the entry of any character string, and even if no syntax check is made, all data is interpreted for program execution (see section 7.1.3 above).

For example:

```
...
N055 (PARTIAL CIRCLE CYCLE COMMAND
N056 ) G77 X+170 Y+50 A+0.0 B+22 D+60 S6
N057 G00 X+162.000 Y+050.000 Z-023.000 T0909 M03
...
```



Please, note that the entries made in Free Format must make sense, and that the program must be able to interpret them.



## 8. Interactive Programming

Interactive programming is a reciprocal "supplement" to the **Editor** and **Automatic Mode**. NC programs can be created, tested and optimized with it. You first program an NC block or machining cycle; the corresponding machining simulation is then performed. In the subsequent dialogue you decide whether you accept the program sequence as it is or whether you want to correct or change it.

### Collision Monitoring

Like in the automatic mode, exact **collision monitoring** is made in interactive programming as well. If there is a collision it is displayed. After the error message has been confirmed the incorrect machining cycle is canceled. The NC block can then be corrected and tested again by simulation. Parallel programming and workpart machining shows which "information" the NC commands contain for the control system and the opportunity to make immediate corrections supports error-free NC programming.

### Note

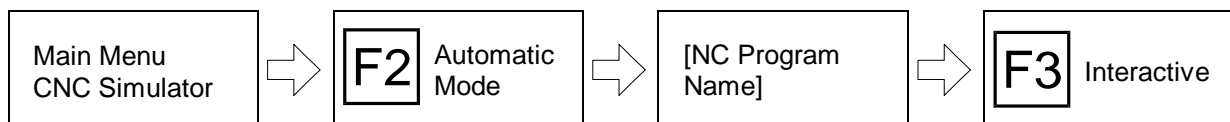
The presettings of the configuration and the previously defined machine setup apply for interactive programming.

Since error and collision monitoring do not consider the programmed feedrates and speed rates, these should be tested prior to the transfer of the NC program to a CNC machine tool.



It is the aim of this manual to provide a complete description of the program functions in each chapter they occur to avoid unnecessary cross-referencing. However, repeating the same function again and again may also get boring. Since many subroutines of interactive programming - in particular loading an NC program, defining the range, modifying the graphical display, etc., - are the same as for the automatic mode and other subroutines, these should be read first.. Please refer to the programming instructions ([Programmers Guide for Turning and Milling](#)) and the configuration instructions for further information.

### Selection:



Interactive programming is integrated in the Automatic Mode: proceeding from the main menu first select Automatic Mode with **F2**, then load an NC program and if necessary enter the editing range. The interactive programming mode is not selected (with **F3**) until when selecting the simulation mode.

### Setup

Please, check that the CNC Simulator has been **set up** correctly before starting the Automatic Mode (cf. above, [Chapter 4](#)).

### Setup Sheet

In addition to the option of setting up the machine tool manually, this can be done automatically with the setup sheet. **Provided the setup sheet is included in the program and the interpreter is switched on, the machine tool is set up according to the set-up sheet on starting the NC program.**

Figure 8.-1:  
CNC Simulator,  
Interactive  
Programming  
Input of an NC block  
(Deep Hole Drilling  
Cycle).

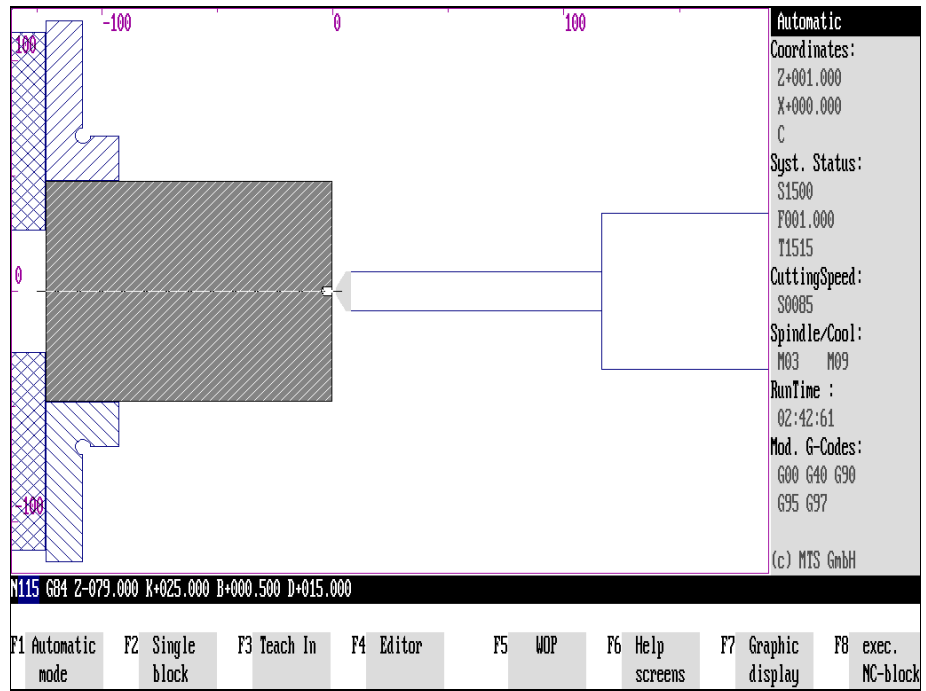
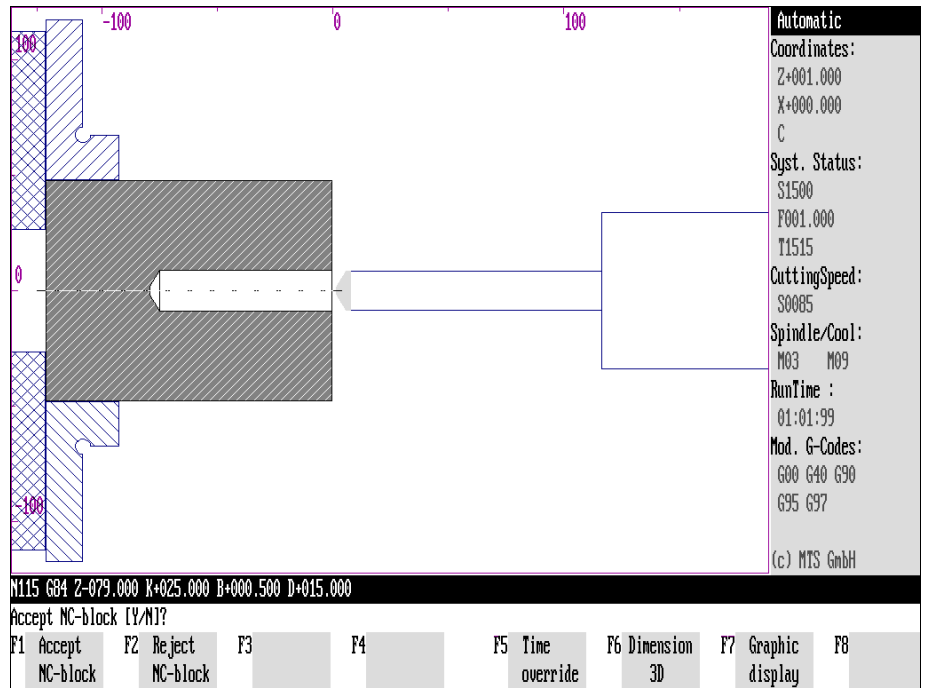


Figure 8.-2:  
CNC Simulator,  
Interactive  
Programming  
Dialogue to confirm an  
NC block.





## Screen Layout

The **screen display** has an analogous layout to that of the Automatic Mode screen: In the upper screen area part of the work space of the machine tool with blank or workpart, clamping fixture and - according to its position - tool is displayed. A graphic-dynamic workpart machining is displayed during the simulation. The information column on the right side of the screen contains information on the current system status.

### Dialogue Guidance

The dialogue guidance of interactive programming is in the two information lines under the work space window:

The top line always displays the **current NC block** of the program generation. **Error and collision messages** and the **program dialogue** are displayed in the line below it.

To use interactive programming is simple and clear: You enter the NC block in the input menu Interactive Programming and then either confirm or cancel the programmed NC block with the dialogue menu.

As a special feature, it is also possible to switch over to the other programming modes of the CNC Simulator, without having to interrupt program processing. In addition, the programming aids as well as other additional functions of the Automatic Mode are available:

### Input Menu

In the dialogue line you enter the next data record.

F1

**Automatic Run:** With **F1** you end interactive programming and test the remaining NC blocks in the automatic run. This function is not available for a program to be generated.

F2

**Single Block:** With **F2** you end interactive programming and test the remaining NC blocks in the single block mode. This function is not available for a new program.

F3

**Teach-In:** With **F3** you switch to the Teach-In (cf. below).

F4

**Editor:** With **F4** you switch to the NC editor.

F4

**WOP:** With **F5** you switch to workshop-oriented programming - provided this program is available.

F6

**Help Images:** With **F6** you call up the programming aids.

F7

**Graphical Display:** To change the display, press **F7**.

F8



**Execute Block:** You accept the programmed NC block with **F8** and the machining simulation is performed.

### Dialogue Menu

After machining simulation, you can take the NC block into the program:

F1



**Accept Block:** With **F1** you take the last executed NC block/blocks into the NC program.

F2



**Reject Block:** With **F2** you release the last executed NC block/blocks for correction.

F5

**Override/Times:** To change the feed override, to switch on the test run or to define the tool usage times, press **F5**.

F6

**[Measuring]/3D view:** With **F6** you switch to the "Measuring" function (only CNC turning) or 3D view.

F7

**Graphical Display:** To change the display, press **F7**.

Figure 8.1-1:  
CNC Turning,  
Schematic of the  
menu sequence for  
Interactive  
Programming.

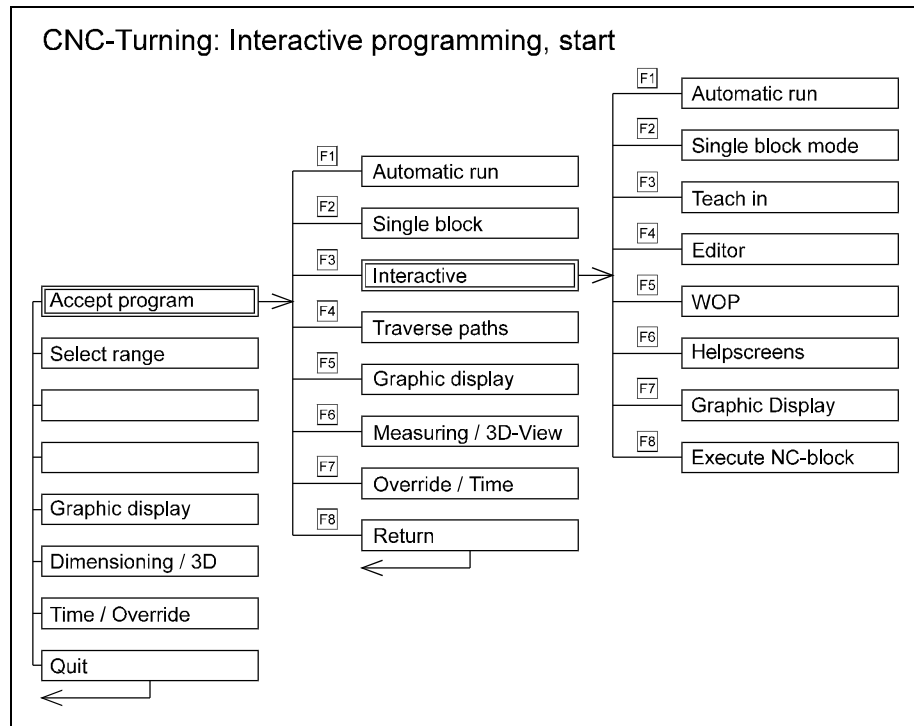
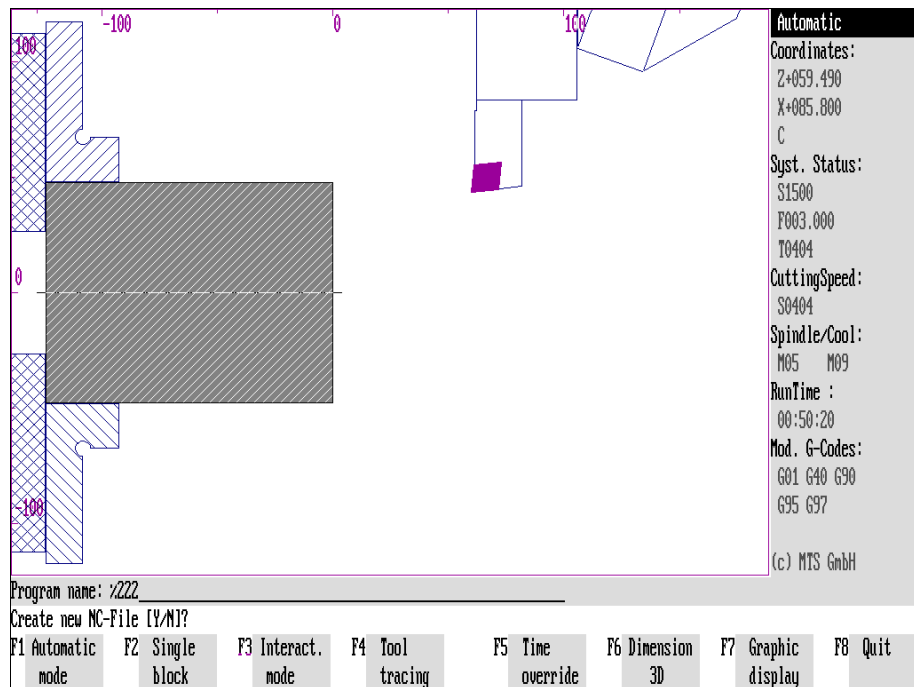


Figure 8.1-2:  
CNC Turning,  
Automatic mode;  
Enquiry of the file  
name of the desired  
NC program:  
after a new program  
name is stated, there  
is a control enquiry:  
"Create new program  
?". If this is confirmed  
with [Y], the CNC  
Simulator creates a  
new program file.

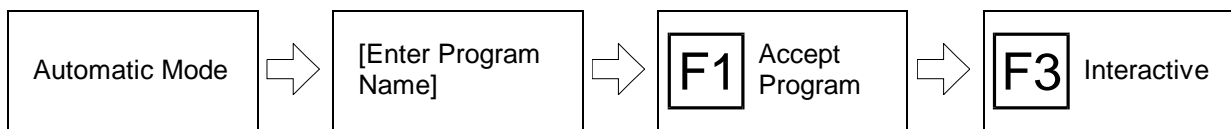


## 8.1 Activate Interactive Programming

### Load an Existing NC Program

Interactive programming is activated from the **Automatic Mode**. Please check that the CNC Simulator has been set up correctly (cf. above, [Chapter 4](#))

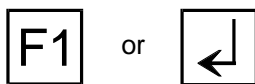
#### Selection



The Automatic Mode is called up from the main menu of the CNC Simulator with **F2**, from the program manager of the editor with **F8** (indicated by the message: "Automatic Mode").

#### [Program Name]

Enter the name of the desired NC program in the dialogue line. If a program is already loaded in the program memory (e.g. if it was edited previously) its name appears as a default entry in the dialogue line. To load another program overwrite this statement.



**Accept program:** Confirm the statement of the program name with **F1** or **←**.



**Interactive:** You choose the simulation mode in the following menu: with **F3** you switch to interactive programming.



If you have selected this processing mode by mistake, cancel it with **ESC**.

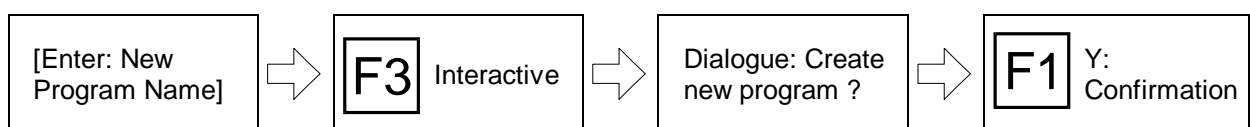
#### Cross Reference

The stated NC program must be stored in the currently selected subdirectory (standard: ".\DPROG" or ".\FPROG", cf. above, [Chapter 7](#)). When in doubt you can check via the file manager of the NC editor which NC programs are available and if necessary also change the path statement.

### Create and Load a New NC Program

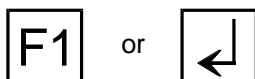
It is also possible to start interactive program processing with a **new NC program**. The starting point is again the main menu of the Automatic Mode:

#### Selection:



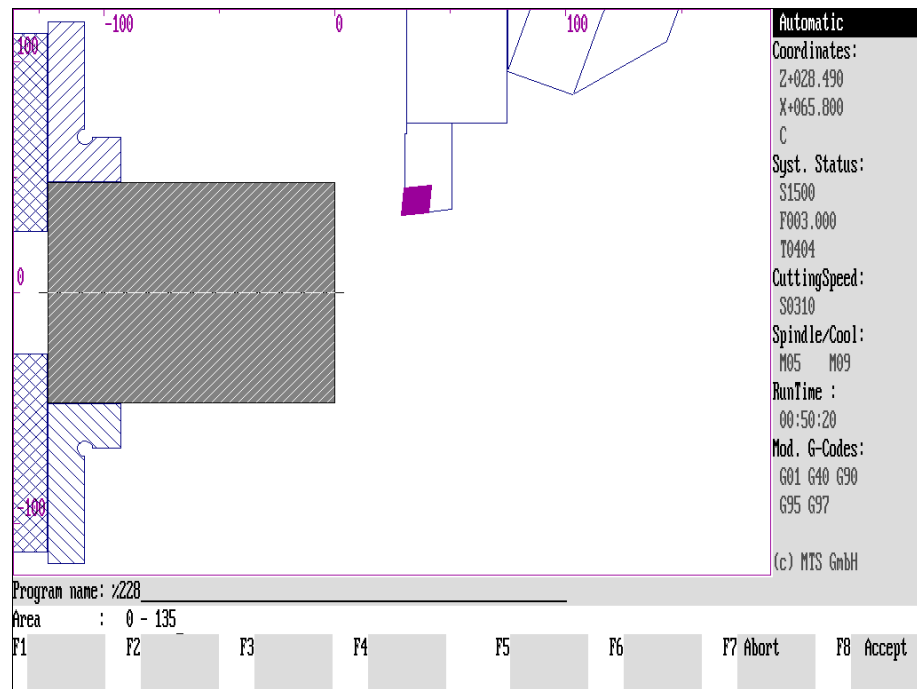
#### [Program Name]

Enter the name of the **new program file** in the dialogue line. Existing entries can be overwritten.



Confirm the program name with **F1** or **←**.

Figure 8.1-3: CNC Turning, Interactive programming, Range definition; After the program name has been entered, it is also possible to load only a part of the NC program. The corresponding NC block numbers are entered in the range definition.



#### Range definition, example:

```

...
N039 (CYCLE INVOCATION ON A STRAIGHT LINE
N040 G78 X+015.000 Y+072.000 A+000.000 D+022.000 S0006 Z+005.000
N041 X+020.000 Y+020.000
N042 Z+005.000
N043 (RECTANGULAR POCKET CYCLE
N044 G87 X+030.000 Y+032.000 Z-040.000 I-050.000 K+025.000
N045 (CYCLE INVOCATION ON A STRAIGHT LINE
N046 G78 A+000.000 D+050.000 S0003          desired processing
N047 (CIRCULAR POCKET CYCLE
N048 G88 Z-050.000 B+015.000 I+200.000 K-080.000
N049 (CYCLE INVOCATION ON A POINT
N050 G79 X+170.000 Y+050.000
N051 G00 Z+005.000 T0404 M03
N052 (DEEP DRILLING CYCLE WITH CHIP BREAKING
N053 G83 Z-050.000 K+030.000 D+010.000
N054 X+192.000
N055 (CYCLE INVOCATION ON A DIVIDED CIRCLE
N056 G77 X+170.000 Y+050.000 A+000.000 B+022.000 D+060.000 S0006
N057 G00 X+162.000 Y+050.000 Z-023.000 T0909 M03
...

```

A bracket on the right side of the code blocks N045 through N051 is labeled "defined range".

**F3**

**Interactive:** In the subsequent menu select the simulation mode: switch to interactive programming with **F3**.

**In the case of a new program name, there is a control enquiry: "Create new program?" before the CNC Simulator creates a new program file.**

**Y**

or

**↵****N**

or

**ESC**

If you want to create a new NC program, confirm this message with **Y** or **↵**, otherwise cancel the process with **N** or **ESC**.

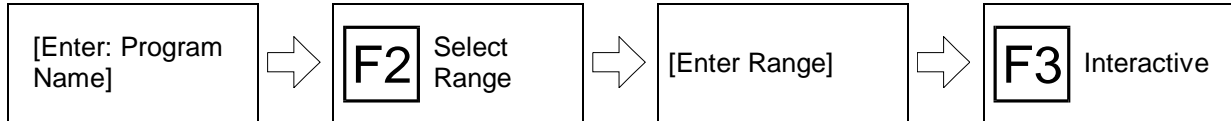
## Load Part of an NC Program

In general, the program lines of an NC program are numbered continuously with the letter "N" and a subsequent three digit block number at the beginning of each line. If you want to use the interactive mode only for a certain part of the program define the desired program part after entering the program name.

### Note

Please note that the start of the NC program at a specified point frequently presupposes certain workpart geometry. Therefore, save the prefabricated workpart (cf. above, [Chapter 3](#)) first and set up the CNC Simulator accordingly before the program start.

### Selection:



The Automatic Mode needs to be called up first.

**[Program Name]**

First, enter the name of the desired NC program in the dialogue line.

**F2**

**Select range:** To process the range definition section by section press **F2**.

**[Entry of the lower and upper Block Number]**

To define the range of the NC program, enter the last or first block number **before or after** the desired range in the dialogue line.

**F8**

or

**↵**

**Accept:** In addition, you should in each case confirm the two block numbers with **F8** or **↵**. If no entry is made, then "N000" is taken as the first block number and "N999" as the last.

Subsequently select interactive programming as described above. The NC program is then read in starting from the selected partial range.

**↵**

or

**ESC**

After the selected range has been processed, there is a dialogue as to whether the remaining NC blocks should be executed or not.

### Note

It is natural enough that the range definition makes sense only for an existing program. If you apply the range definition for a new program, then the program file is created but processing is interrupted with the message "Inconsistent conclusion of a program part" and the program is created without NC data.

Figure 8.2-1:  
CNC Simulator,  
Interactive  
programming;  
Schematic of the  
menu sequence.

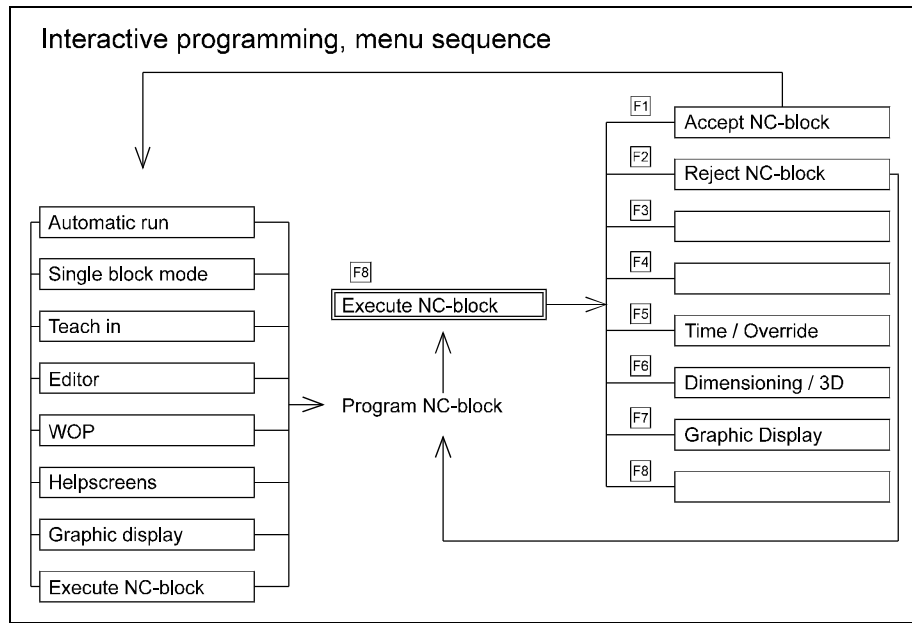
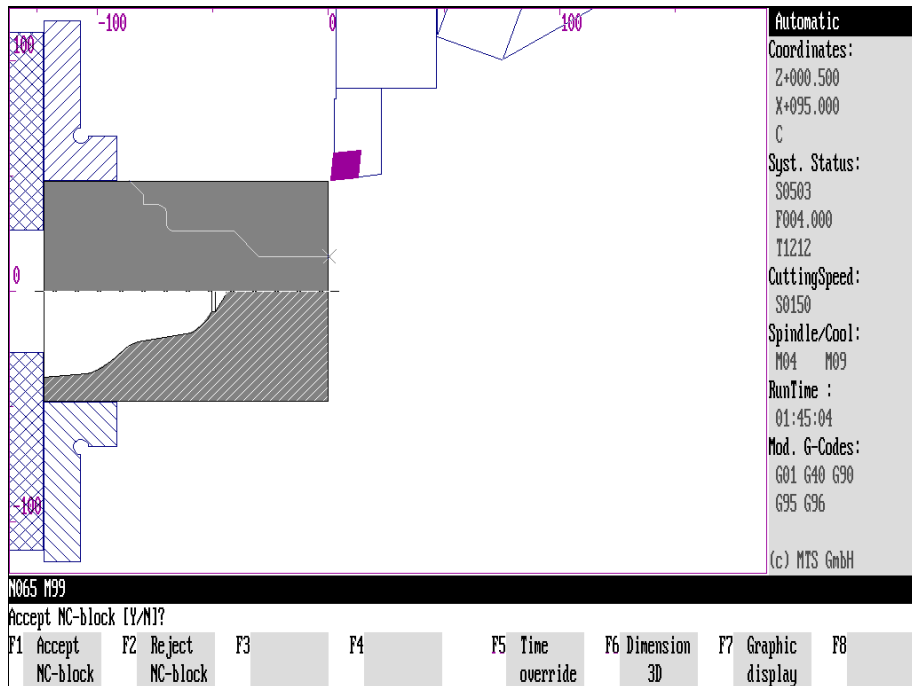


Figure 8.2-2:  
CNC Turning,  
Interactive  
programming;  
Processing a cycle:  
Simulation of  
machining does not  
start until the entire  
cycle (here rough  
turning cycle and  
contour definition in a  
subprogram) is  
confirmed.  
The contour is  
displayed for checking.



## 8.2 Programming NC Blocks

### NC Block Input


In contrast to the Automatic Mode, the NC block displayed in the dialogue line can be edited in interactive programming. After one NC block has been confirmed, the simulation is performed. Finally, different correction possibilities are available to change an NC program.

### [Display of the NC Block]

After confirming the program name the NC program is loaded and the first NC block is output in the programming line. If a new NC program is created, only the first block number (e.g. "N10") is available.

### [Creating or Changing an NC Block]

The NC block which has been read into the dialogue line can be then processed. **In this case you can use, even in interactive programming, all editing functions which the editor provides:** in addition to a letter and number input, address and value data can be overwritten as well and you can delete or insert characters. To overwrite individual data, position the cursor at the desired position.

For the input you just state the individual letters and digit values of the immediately following NC block. **Do not confirm the individual NC words with** !

For example: enter the NC block of Figure 8.-1 as follows (assuming a "blank" program line is available):

**N115 g 8 4 k 2 5 b 0 . 5 d 1 5 z - 7 9**

This line is then automatically formatted into the following NC block:










**N115 G84 K+025.000 B+000.500 D+015.000 Z-079.000**

Unlike in NC editor, the NC words are not automatically sorted in interactive programming.



You can try out the input functions with a "Test program".

### Overview of Editing Functions

[Character Input]	:	Input / Overwrite an NC block
 or 	:	Position cursor
 or 	:	Insert or delete a character or NC word
 and 	:	Insert an NC block <sup>1</sup>
 and 	:	Delete an NC block
	:	Position cursor (word by word)

<sup>1</sup> The insert mode of an NC block remains activated until it is switched off with the same key combination.

Figure 8.2-3: CNC Simulator, Interactive programming, recessing cycle: since the recessing cycle should be performed twice, the distance of the recesses from each other is first defined with G83 and then the recess with G79.

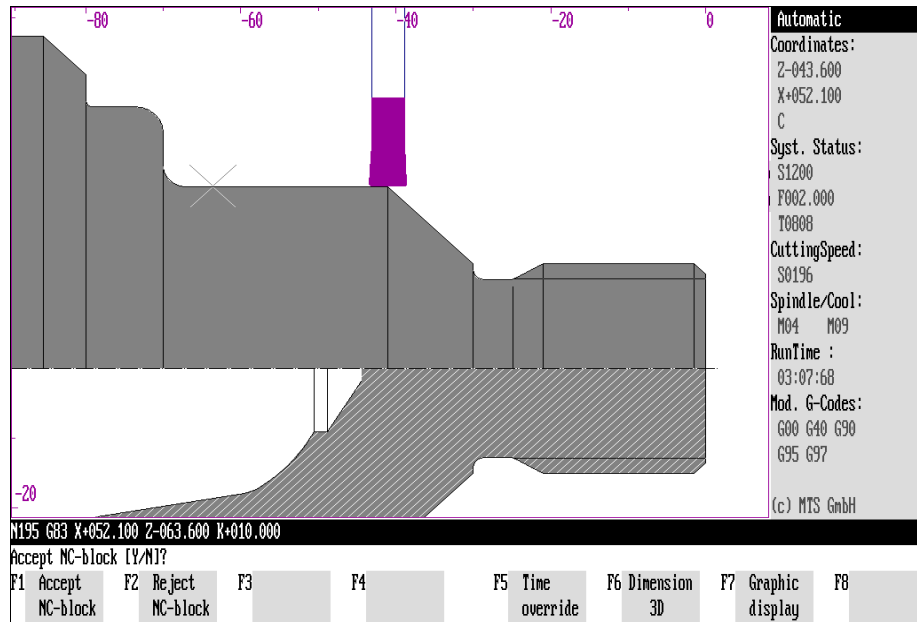


Figure 8.2-4: CNC Simulator, Interactive Programming, before machining the recessing cycle, its geometry has to be defined first.

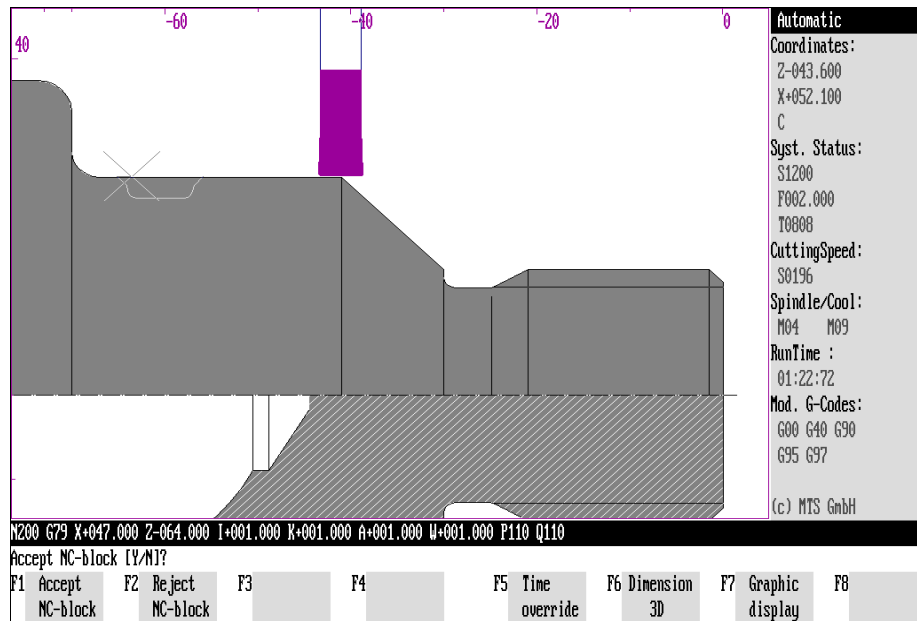
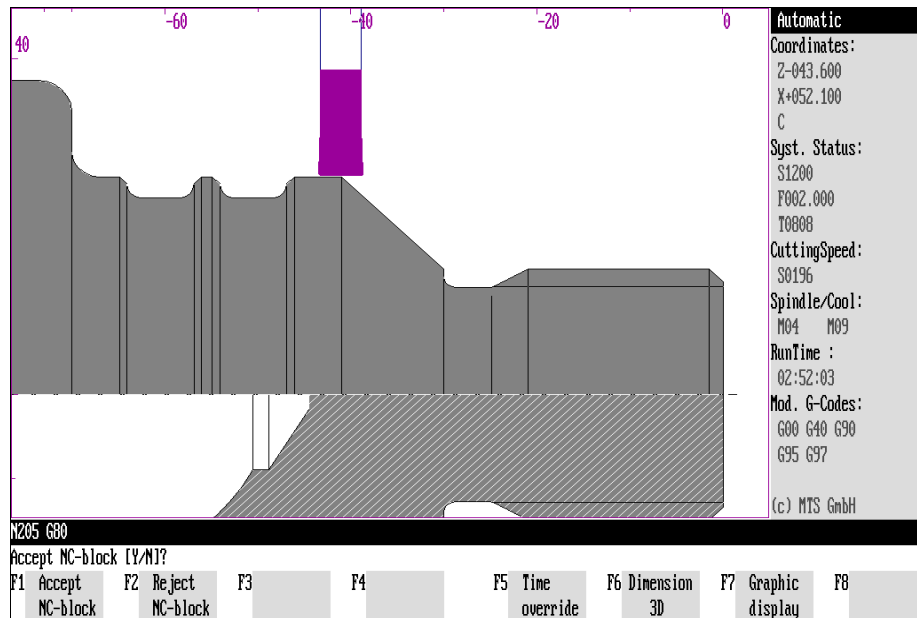


Figure 8.2-5: CNC Simulator, Interactive Programming; after the workpart has been machined, you can decide in a dialogue whether you want to confirm or change the programming.






[Completion of an NC Block] Having entered the NC block completely, machining can be simulated:

**F8**

or



**Execute block:** Execution of the NC block is started with **F8** or . When **programming a "simple" NC block** the corresponding machining simulation is then performed immediately.

#### Radius Compensation

Note that **with radius compensation switched on, each NC block is performed offset by at least one NC block**, since the end point of a section or of an arc can only be computed by the subsequent NC block (cf. Configuration instructions, Control configuration: Interpretation).

#### Cycle

In **programming a machining cycle**, the cycle is first defined, then it is invoked and finally the simulation is performed.

#### Note

In interactive programming the addresses are not sorted because of memory capacity.

In addition to the editing options, you have the choice of the following **program functions**:

**F1**

**Automatic Run:** With **F1** you end interactive programming and test the remaining NC blocks in automatic run. This function is not available for a new program.

**F2**

**Single Block:** With **F2** you conclude interactive programming and test the remaining NC blocks in single block mode. This function is not available for a new program.

**F3**

**Teach-In:** With **F3** you switch to Teach-In Mode (cf. below).

**F4**

**Editor:** With **F4** you switch to the NC editor.

**F5**

**WOP:** With **F5** you switch to workshop-oriented programming - provided this program is available.

**F6**

**Help Images:** With **F6** you call up the programming aids.

**F7**

**Graphical Display:** To change the display, press **F7**.

**F8**

As in Automatic Mode, it is also possible in Interactive Programming to **interrupt** machining simulation temporarily with **F8**, in order to enlarge the display representation of the turning operation for example, to call up a 3D view, etc. **The temporary interruption is possible only during machining.** Machining simulation is then continued with **F8**.

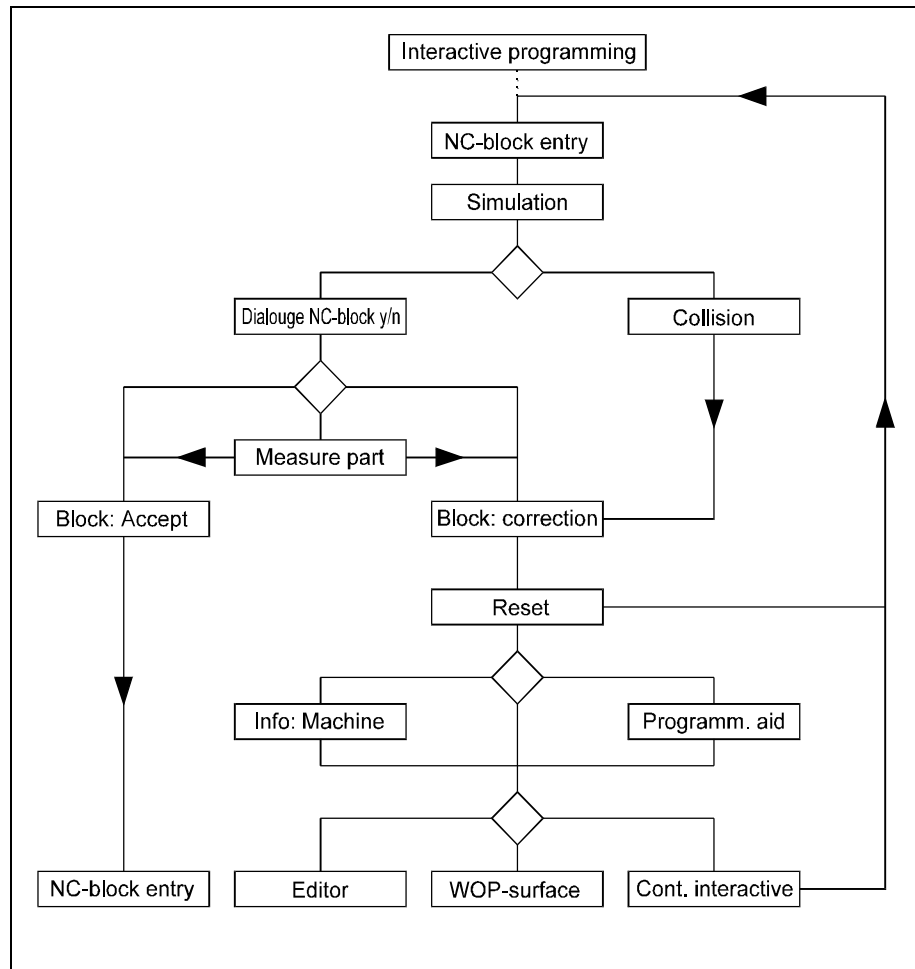
**ESC**

To **cancel** the programming of an NC block, press, **ESC**. The message "Please press **ESC** once again for program cancel" is then displayed. If you now press the **ESC** key once again, you cancel the entire program processing. After operation of any other key, the NC block is released again for editing.

#### Cross Reference

[Chapter 7](#) contains detailed discussion of the **editing functions**.

Figure 8.2-6:  
CNC Simulator,  
Interactive  
Programming;  
processing schematic.



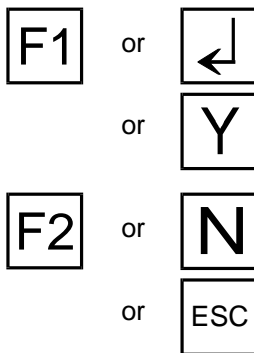
## Interactive Check of NC Blocks

Provided no processing errors occur, the CNC Simulator performs the programmed NC block or cycle. You thus get the control information contained in the NC commands displayed and can create and test NC programs block by block.

### Dialogue

When the processing of the current NC block or machining cycle is completed, a **program dialogue** follows before the next block is edited. This serves the purpose to control the NC command with the message: "**Accept NC Block [Y/N]?**" (cf. Fig. 8.2-3/4/5).

After processing has been completed, the NC block can be taken into the program:



**Accept block:** If you agree with the processing result, confirm with **F1**, **↵** or the **Y** key. The NC block or cycle is added to the program and either the next NC block can be entered or the next program line is read.

**Reject block:** If the execution did not bring the desired result, reject the block with **F2**, **ESC** or the **N** key. The machining process specified in the last block is then canceled (if it is a cycle it can be extensive) and the current NC block (or a cycle call) is proposed again for programming.

You can repeat this process until the desired result has been achieved.

### Note

After having negated the program dialogue with radius compensation switched on, you return to the starting position of radius compensation just like when processing a cycle.

Apart from this dialogue you have further options to modify and check the machining execution:



**Override/Times:** To change the feed override, to switch on the test run or to calculate the tool operation times, select **F5**.

**[Measuring]/3D view:** You switch to the "Measuring" (only CNC turning) or 3D view functions with **F6**.

**Graphical display:** To change the display, press **F7**.

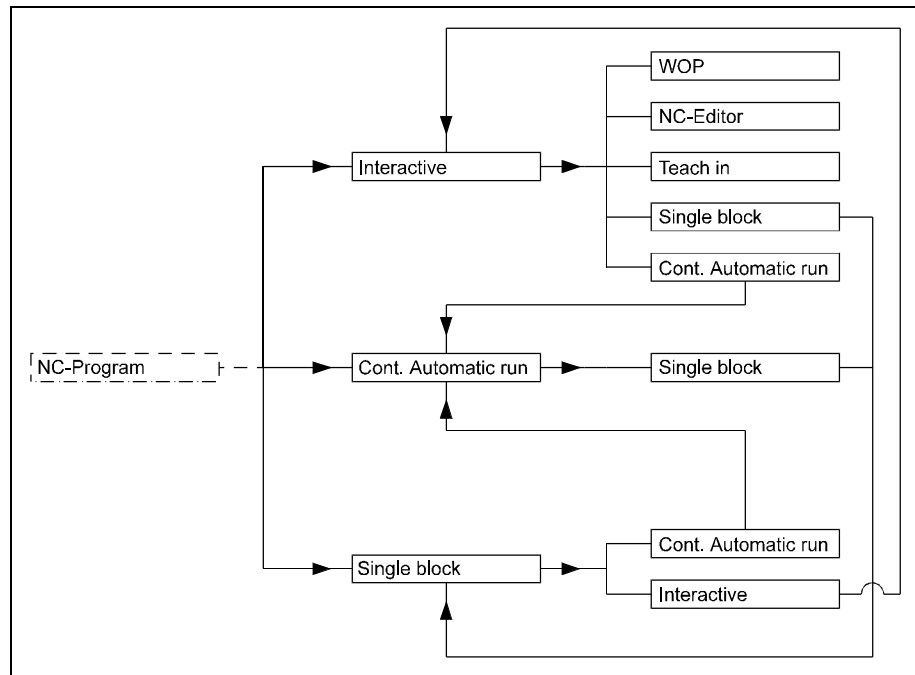
## Collision Monitoring

A program error or collision during simulation is reported in the dialogue line and the execution is interrupted. The error message is then to be confirmed with any key. The processing of the incorrect NC block is then canceled and the program block is released for correction. After modification the block can be tested again.



**An incorrect NC block cannot be taken into the NC program.**

Figure 8.3-1:  
Overview for the  
switch between the  
simulation and  
programming modes;  
simplified schematic  
for the operation  
sequence.



## 8.3 Switching between Programming Modes

As a special feature the CNC Simulator allows you to switch to other programming modes during interactive programming without having to interrupt and to restart the processing of the NC program. The user can change as necessary between:

- **Interactive Programming,**
- the **Editor,**
- **Workshop-Oriented Programming (WOP)** and
- the **Teach-In Mode.**

and continue program processing (cf. schematic opposite page). Furthermore, the programming aids can be called up; or you can subsequently return to automatic run - after having changed already existing NC blocks - in order to have only the subsequent NC blocks executed in the simulation.

With these parallel operating possibilities, the CNC Simulator allows convenient applications of NC programming and of testing or optimizing NC programs, to be used practice-oriented in production and in CNC training.

### Cross Reference

In this section only the switch-over possibilities are discussed: cf. the relevant chapters of these instructions for operating the programming modes.

### Changing over from Interactive Programming to the WOP

With the "WOP interface", contour strings are generated with the help of user guidance by transferring the dimension data of the workpart drawing into the relevant "entity menus" (straight line, arcs, tangentially adjoining entities etc.). The corresponding NC block and the graphic representation of the programmed contour are both displayed.

### Note

The WOP is not included in the standard scope of the CNC Simulator. It is, however, available as a software extension and needs to be ordered separately. If the WOP is not available in your computer, the below described selection option is **not** possible.

It is assumed that you are editing an NC block in interactive programming mode. To continue programming in workshop-oriented programming mode, call up the WOP interface without confirming your entries prior to it:

**F5**

**WOP:** With **F5** you switch to workshop-oriented programming.

**F8**

The main menu of this program part is loaded and from there you can select the further processing operations required.

To continue programming in interactive programming mode, end WOP processing and switch from the main menu of the WOP interface back to interactive programming mode with the **F8** key.

### Note

All NC blocks programmed in WOP are **inserted prior to the NC block** which you had last processed before switching over. If you now continue interactive programming, the first (inserted) NC block that you had generated with WOP is loaded: thus you can now test and if necessary modify these NC blocks in interactive programming mode.

### Change from Interactive Programming to the Editor

Assuming you are editing an NC block in interactive programming mode. To obtain an overview of the entire NC program, call up the editor without confirming your entries prior to it:

**F4**

**Editor:** With the **F4** key, you can switch directly over to the programming interface of the editor: the entire current NC program is displayed and, in addition to this, all editing and program functions of the editor are available: You can thus change, delete, insert etc. any NC block.

**ESC**

To continue programming in interactive programming mode, switch back with the **ESC** key (PC keyboard). You return to the same processing situation from which you had called up the editor.

### Note

All changes made with the editor, are **saved**. However, if you change NC program lines located **prior to** the NC block which is displayed in interactive programming, these changes are no longer valid and they cannot be tested when continuing the "current" interactive machining simulation (therefore, pay attention to changes located prior to the current NC block!). On the other hand, it is possible to insert further program blocks, which can also be tested when continuing interactive programming, **after** the current NC block of interactive programming.

### Calling Programming Aids

In all working situations of the CNC Simulators, programming aids can be activated. They clarify the use of motion commands or cycle calls and are grouped according to their technical function. For better orientation and as an introduction overviews of contents and representations are also available. The user can consequently make use of programming instructions even during processing.

Also, in interactive programming a help window can be called up during the creation/modification of a block as well as during interrupted machining simulation:

**F6**

**Help Window:** Programming aid is called with **F6** or **<?>** during NC block generation. After calling it enter the name of the help needed or choose a group register from which you can select the desired programming aid.

**Also, parallel to the programming aid you can have the current NC block displayed in order to enter changes and corrections.**

**F8**

With **F8** you close the aid function again. You then switch back to the starting situation from which you have called up the aid.

### Change from Interactive Programming to Teach-In Mode

Teach-In Mode is a special operating mode of interactive programming. As in setup mode, the tool is moved manually and the workpart is machined. The traverse paths resulting from the manual mode are in this case converted automatically into simple provisional functions of the command syntax and inserted in the NC program.

If you are editing an NC block in interactive programming call up Teach-In Mode directly without confirming your entries:

**F3**

You switch into Teach-In Mode with the **F3** key. You can now machine the workpart "manually". The simple G-commands generated for this are displayed in each case in the first information line and inserted **prior to** the NC block from which you called up the Teach-In Mode.

**F8**

To continue in interactive programming mode, switch back with **F8**.  
The next NC block is then released for interactive programming.

### Switching Back into Automatic Mode

It is possible to terminate program processing in interactive programming mode at any place and - provided this is not the last NC block of the program - to test the remaining program in automatic run or in single block mode. This switch-over function is ideal for optimizing certain program parts. In this way it is not necessary to work through the entire NC program in interactive mode but you can select for each section the most efficient processing method (cf. above, [Chapter 6](#)).

You can conclude the processing of an NC block and switch directly over into continuous or block-by-block Automatic Mode:

**F1** or **F2**

**Automatic Run / Single Block:** During block generation you call up continuous automatic run with **F1**. Or if you want to test the remaining program blocks in block-by-block machining simulation press the **F2** key.

### Program End or Interrupt Processing

As a rule a program ends with the machine command "M30". The CNC Simulator then switches back into Automatic Mode.

Should the program end with some other command the message "Unexpected program end" is displayed, and after confirmation with any key, you switch back into Automatic Mode.

It is also possible to interrupt program processing directly:

**ESC**

If you press the **ESC** key during programming, the entire program processing is interrupted.

The message "Please press **ESC** once again for program interruption" is then displayed. If you press **ESC** the second time, the program lines created by then are stored, however, the program processing is concluded immediately.



If you press any other key in response to the above message you can continue the programming done prior to the interruption.

## Special Functions

### Cross Reference

The functions for **changing the Graphical Display, Zooming (CNC turning), Measuring (CNC turning), 3D views, changing the override and switching on the test run** are discussed in detail in [Chapter 9](#) and [10](#). Please, refer to these chapters if necessary.

Figure 8.4-1: CNC Turning, Teach-In Mode; as in Setup Mode, the tool is moved by manual control and corresponding to this a simple G-command is created as an NC block. The last NC block is displayed in each case, the dialogue line beneath this is reserved for the error messages etc.

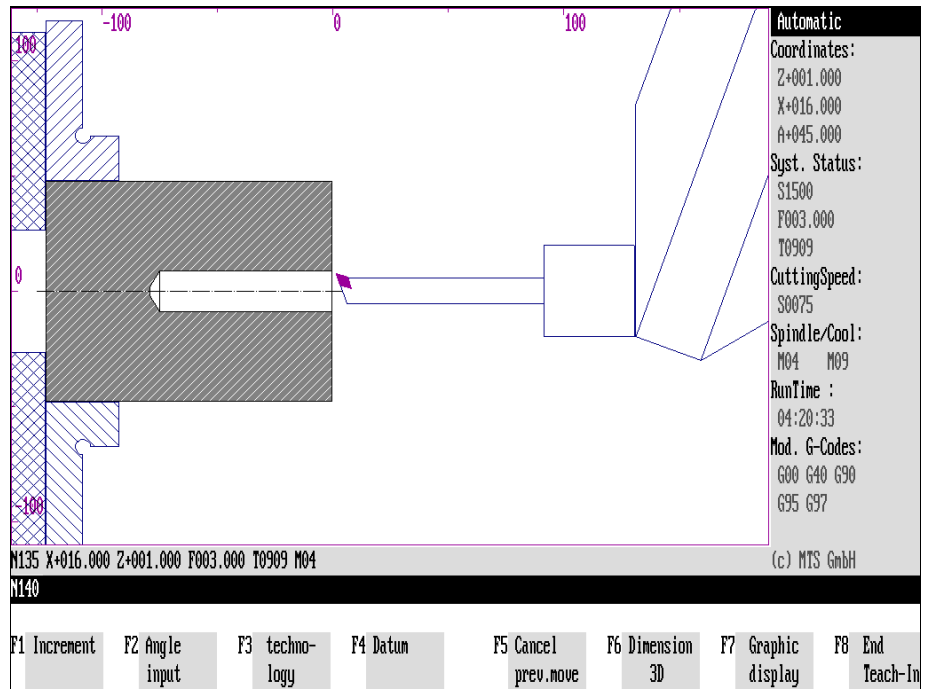
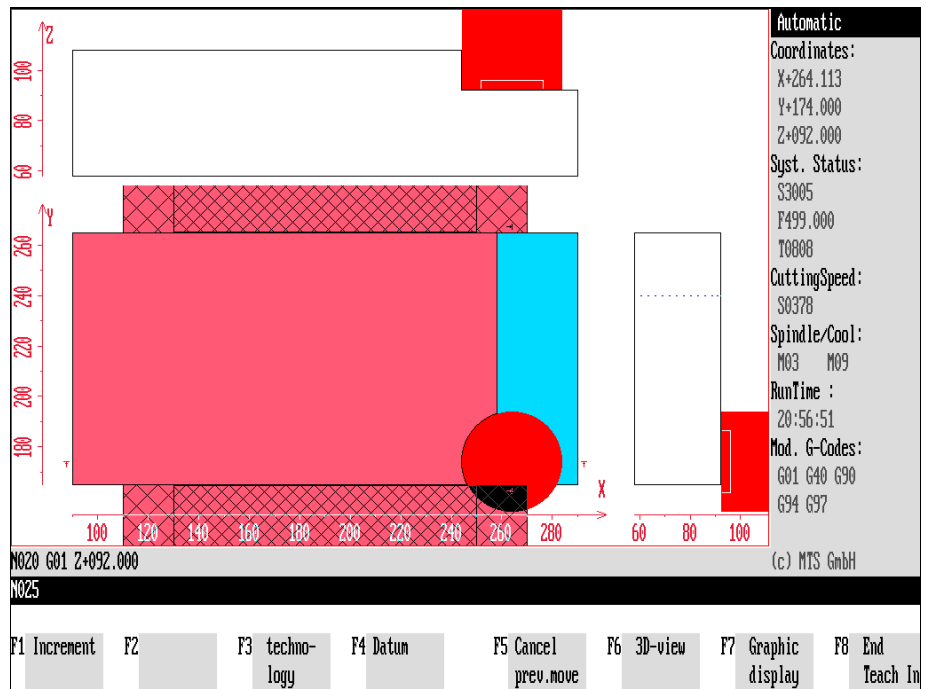


Figure 8.4-2: CNC Milling, Teach-In Mode; in contrast to turning, the "Angle statement" function is not available.





## 8.4 Teach-In Mode

Teach-In Mode offers a special form of interactive programming. In this programming mode, the workpart is machined by manual control like in setup mode. The program automatically converts the resulting traverse paths into simple G-commands (G00 or G01) and inserts them in the NC program.

### Teach-In Concept

To make Teach-In Mode a complete programming mode, the following processing options were included in it:

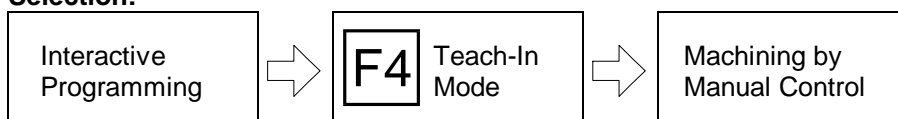
- Reading the contour of the finished part as "nominal contour" to avoid any breach with the final contour ("gouging") (only CNC turning).
- Programming of the additional and switching functions such as speed, feedrate, tool change and M commands
- Programming the zero point.
- Angle statement for the motion commands G00 and G01 (only CNC Turning).

With these functions, a complete NC program including all technology data can be created in Teach-In Mode without entering any command code. The NC program blocks are created by the system in the background and displayed on the screen - and can thus be identified by the user at any time. Therefore, the Teach-In Mode is suitable as "the first stage" of CNC training and Simulator.

### Display

The **screen display** is structured analogously to interactive programming: In the upper part, a section of the work space is shown and a graphic-dynamic representation of the workpart machining is displayed during manual control. The lower dialogue lines give **information** on current machining: in the upper lines, the last NC block is displayed and error and collision messages are displayed below it.

### Selection:



### TEACH-IN

Teach-In Mode is called up with the **F3** key during interactive programming. The main menu of the Teach-In mode then loads in automatically:

**F1**

**Increment:** As in setup mode, you can also set under "increment" the tool movement unit applied when pressing the traverse key in Teach-In programming.

**F2**

**[Angle statement]:** The turning tool can be moved at an angle of 45° (or 135°, 225° and 315°) with the **1**, **9** and **3**, **7** keys on the numerical keyboard (PC keyboard). You can change this angle setting with the "Angle statement" function.

**F3**

**Technology:** In this submenu you can program speed, feedrate, tool change and M commands.

**F4**

**Zero point:** In this submenu you can program the workpart zero point.

**F5**

**Reject traverse motion:** With **F5** you can delete the last generated NC block again.

**F6**

**[Measuring]/3D view:** With **F6** you switch to the measuring (CNC turning) and 3D view functions.

**F7**

**Graphical display:** To change the display, press **F7**.

**F8**

**End Teach-In Mode:** With **F8** you finish Teach-In programming and switch back to Interactive Programming mode.

Figure 8.4.1-1: CNC Turning, Teach-In Mode; Reading in the nominal contour: if a nominal contour has been defined, the area limited by it can no longer be machined. Consequently, it is not possible to violate the finished part geometry.

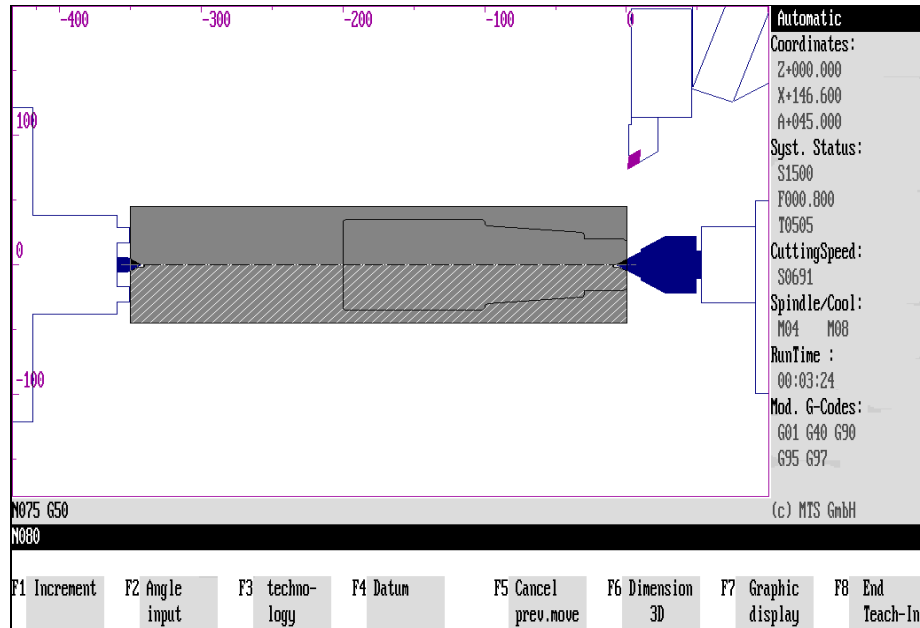


Figure 8.4.1-2: CNC Turning, Workshop-Oriented Programming; the easiest way to define a nominal contour is to use the WOP interface.

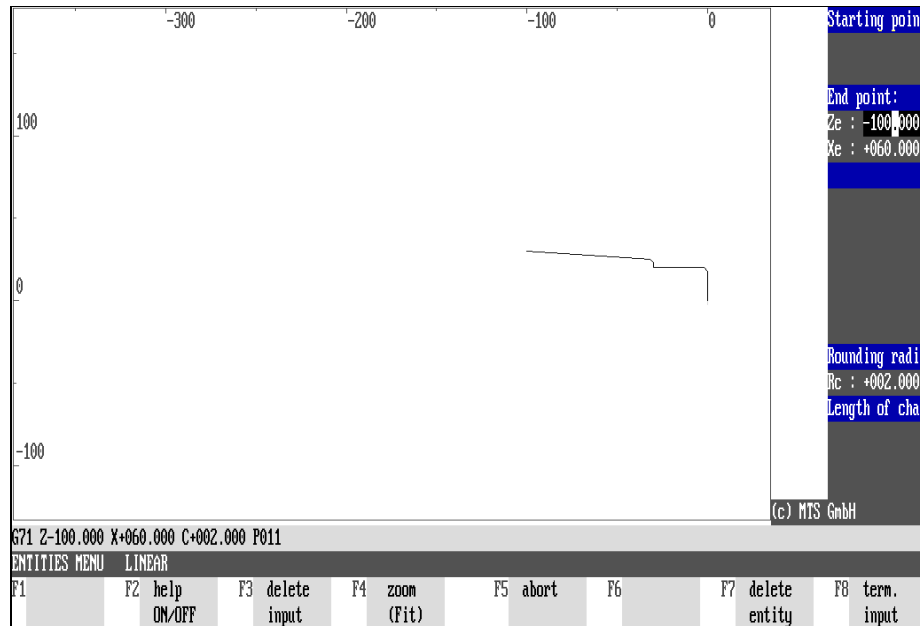
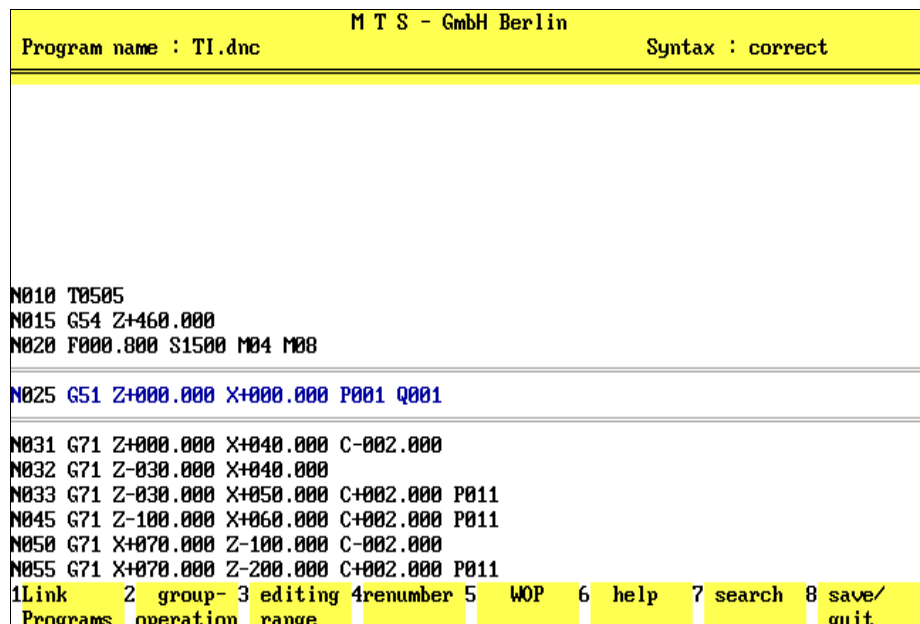
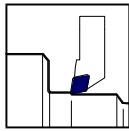


Figure 8.4.1-3: CNC Turning, NC program for the nominal contour; The "nominal contour" is defined in an NC program by the commands "G51" (start) and "G50" (end).



### 8.4.1 CNC Turning: Nominal Contour



Reading-in and displaying a nominal contour is available only for turning. It can be used especially in Teach-In programming:

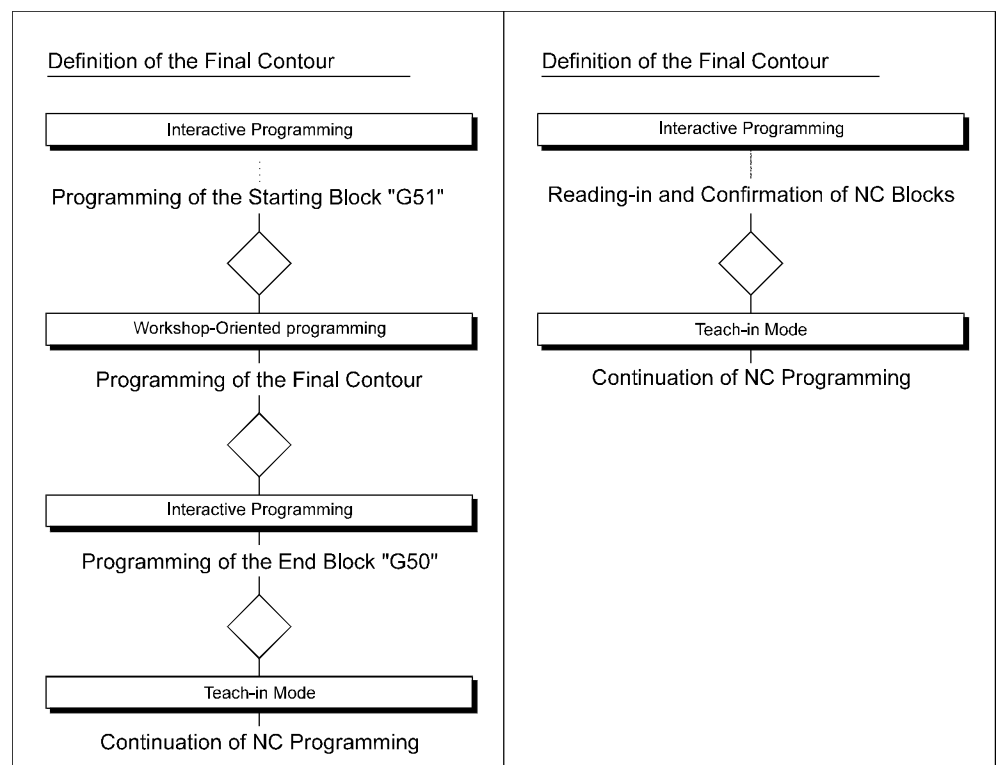
A defined contour which should not be violated during tool movement - similar to limiting the clamping jaws, the tailstock etc. - is called "nominal contour". Machining beyond this line is not possible and causes "collision".



**It is reasonable to define the geometry of the finished part as "nominal contour" to avoid any damage of the workpart during manual machining.**

### Defining Nominal Contour

The nominal contour is specified with the NC commands "G51" and "G50" as well as with the normal contour description according to DIN or the segment contour programming (cf. Programming Instructions). For convenient handling we would recommend you the following procedure:

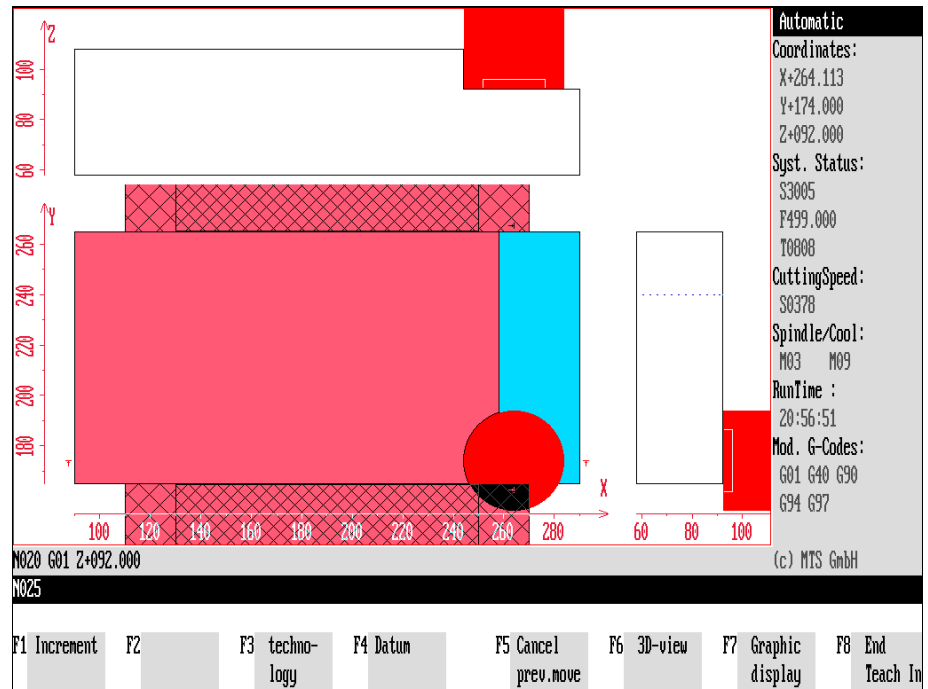


1. Enter the start and end block (G51/G50) in Interactive Programming,
2. the nominal contour is specified with Workshop-Oriented Programming,
3. then switch to the Teach-In Mode and continue your NC programming.



In CNC training the trainer can specify the start/end block and the path of the nominal contour, whereas the Teach-In programming should be done by the trainee. Furthermore, with reference to the nominal contour it is possible to demonstrate in a simple way machining deviations with/without nose radius compensation operative.

Figure 8.4.2:  
CNC Turning,  
Teach-In Mode;  
Menu for entering the  
Technology Data.



## 8.4.2 Input of Technology Data

TECHNO

### 1st Menu:

F1

**F, feedrate mm/rev:** To enter the wanted feedrate in mm/rev, press **F1**.

F2

**F, feedrate mm/min:** To enter the wanted feedrate in mm/min, press **F2**.

F3

**S, speed:** Machining speed is programmed with **F3**.

F4

**S, cutting speed:** To program constant cutting speed, press **F4**.

F5

**[S, maximum speed:]** The maximum speed for turning is defined with **F5**. The command "G92" (speed limit) is generated for this in the background.

F6

**Spindle/coolant:** You switch to the spindle and coolant controls with **F6**.

F7

**T, tool:** With **F7** you can program a tool change including the statement of the corresponding compensation value register.

F8

**Return:** After you have specified the technology data as desired, return to the main menu of the Teach-In Mode with **F8**.

### Procedure

Each function is processed analogously:

F1

or

F7

Call up the desired menu item with a function key.

[Value Entry]

The corresponding data field is then activated in the information column on the right side of the screen and you can now enter the data.

F7

or

F8

**Cancel/Accept:** You confirm your entry with **F8** or **↵** and cancel the operation with **F7** or **ESC**.

### 2nd Menu:

F1

**Spindle clockwise:** Press **F1** to switch on the spindle (clockwise/M03).

F2

**Spindle counter-clockwise:** Press **F2** to switch on the spindle (counter-clockwise/M04).

F3

**Spindle off:** Press **F3** to switch off the spindle (M05).

F5

**Coolant M08:** Press **F5** to switch the coolant pump (M08) on.

F6

**Coolant M07:** Press **F6** to switch the coolant pump (M07) on.

F7

**Coolant off:** Press **F7** to switch the coolant pump off (M09).

F8

**Return:** Press **F8** to return to the starting situation.

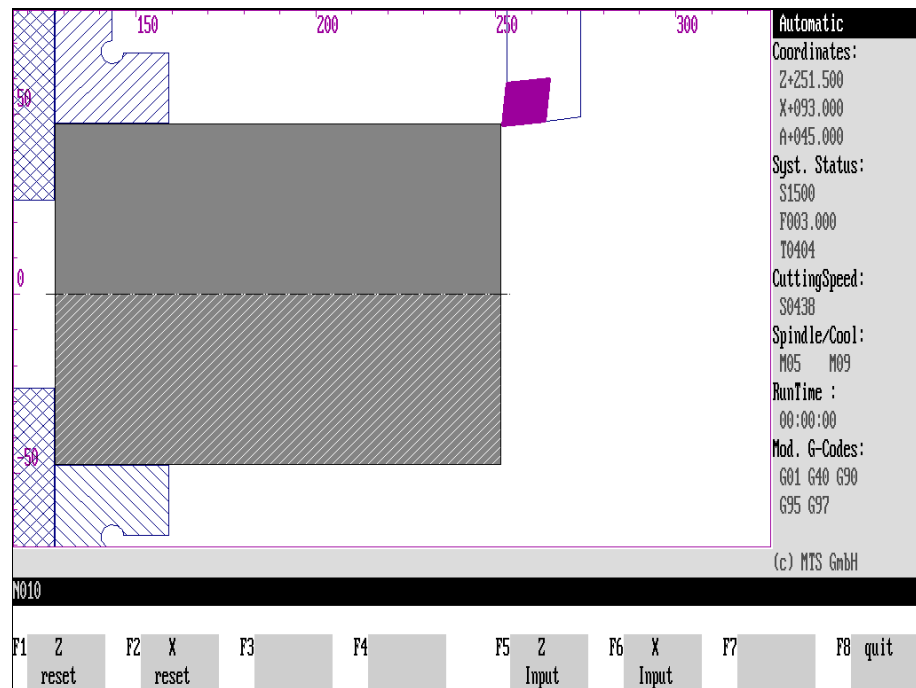


Before you switch back to the main menu, enter all technology data as required. The corresponding NC block is not generated and displayed until during the generation of the next NC block.

Cf. also [Chapter 4](#), Setup Mode.

### Cross Reference

Figure 8.4.3:  
CNC Turning,  
Teach-In Mode;  
Menu for entering the  
Workpart Zero Point  
(by touching).

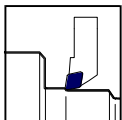


### 8.4.3 Programming Workpart Zero Point



To program the workpart zero point with the command G54, call the "Zero Point" function with **F3** proceeding from the main menu of the Teach-In Mode:

#### CNC Turning



To determine the workpart zero point for turning, you can either enter the Z and the X coordinates manually or you determine them by touching the workpart:

**F1**

**Z zeroing:** Press **F1** to place the Z value of the zero point on the current tool position. First move the tool to the right position by touching.

**F2**

**X zeroing:** Press **F2** to place the X value of the zero point on the current tool position. Move first the tool to right position by touching.

**F5**

**Z entry:** To enter the Z value manually, press **F5**.

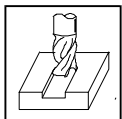
**F6**

**X entry:** To enter the X value manually, press **F6**.

**F8**

**Return:** Press **F8** to return to the main menu of the Teach-In Mode. The corresponding NC block (G54 ...) is not generated and displayed until when generating the next NC block.

#### CNC Milling



Analogous to turning, the workpart zero point is determined in milling either by a manual entry of the coordinates or by touching the workpart:

**F1**

**X zeroing:** Press **F1** to place the X value of the zero point on the current tool position. Move first the tool to the right position by touching.

**F2**

**Y zeroing:** Press **F2** to place the Y value of the zero point on the current tool position. Move first the tool to the right position by touching.

**F3**

**Z zeroing:** Press **F3** to place the Z value of the zero point on the current tool position. Move first the tool to the right position by touching.

**F5**

**X entry:** To enter the X value manually, press **F5**.

**F6**

**Y entry:** To enter the Y value manually, press **F6**.

**F7**

**Z entry:** To enter the Z value manually, press **F7**.

**F8**

**Return:** Press **F8** to return to the main menu of the Teach-In Mode. The corresponding NC block (G54 ...) is not generated and displayed until when generating the next NC block.

#### Note

After touching, the message: "**Note: After touching move first back to the right starting point!**" is displayed. Its purpose is to remind you to move to the starting point before you switch back to the main menu. If you do not reset the tool until you are in the main menu, the corresponding G commands for these movements are automatically generated as NC blocks!



The following section describes how the tool is moved.

Figure 8.4.4-1:  
CNC Turning,  
Teach-In Mode;  
Generating simple  
Motion Commands.

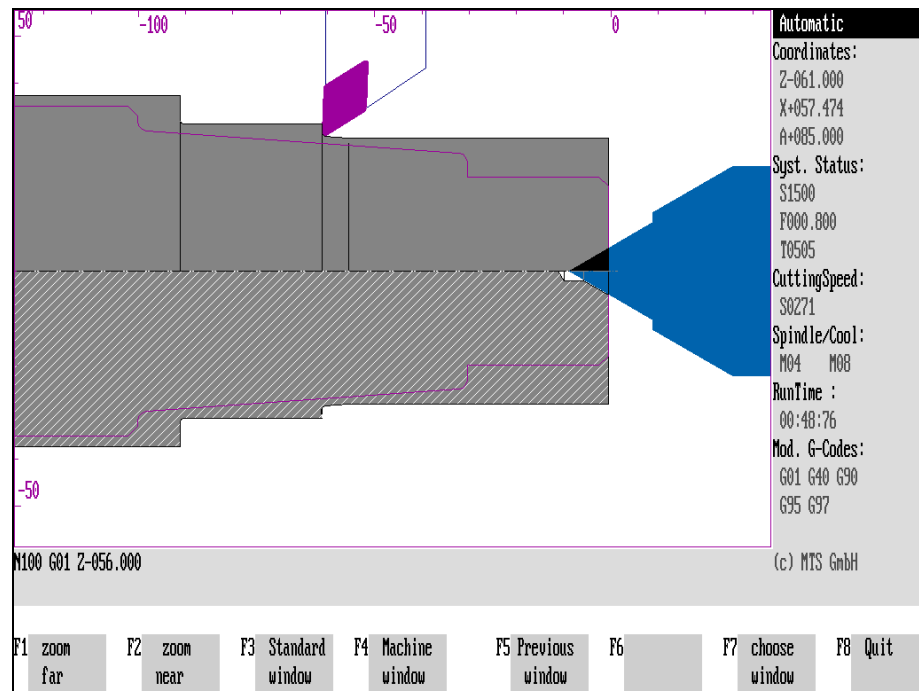
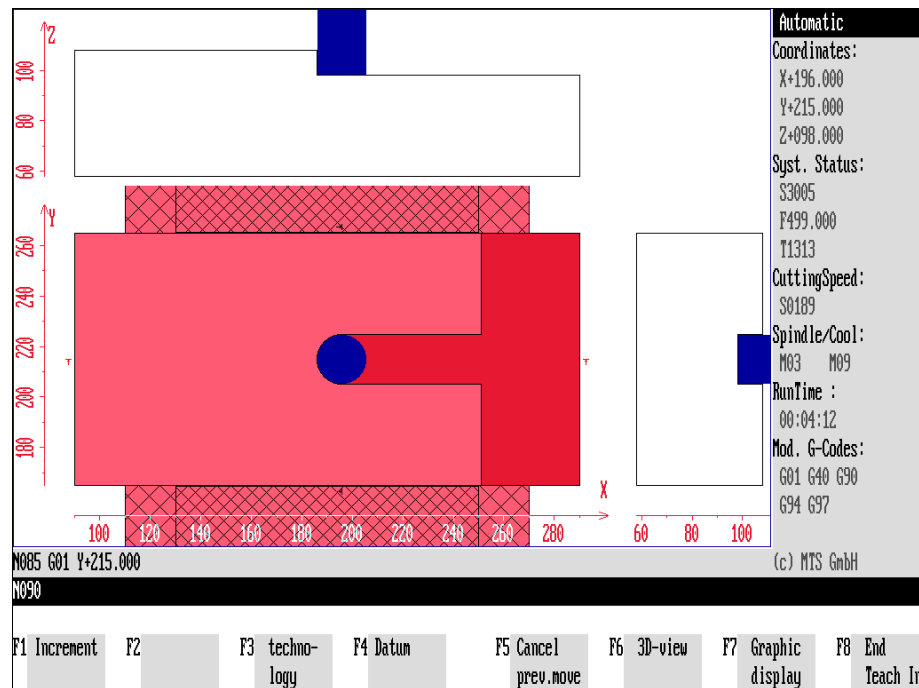


Figure 8.4.4-2:  
CNC Milling,  
Teach-In Mode;  
Generating simple  
Motion Commands.

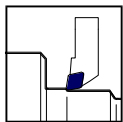




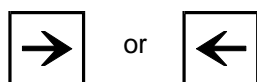
### 8.4.4 Machining with Manual Control

After the necessary switching commands have been determined, you can machine the workpart "manually". The corresponding preparatory functions generated are displayed in each case in the first information line and inserted **after** the last processed NC block.

#### CNC Turning

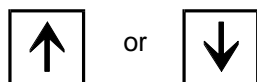


In the turning CNC Simulator, the tool slide can be moved in **fast speed** or in **feed motion**. Note that in the Teach-In Mode, you can also enter machine commands (tool change, spindle, feedrate, speed).

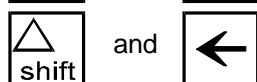


The tool slide is moved in the Z direction with the cursor keys or . If the spindle is switched on, the workpart can be machined and cutting is shown on the screen.

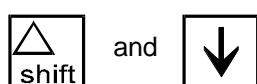
**Collisions** with the toolholder etc. are indicated by error messages.



The tool slide is moved analogously in the X direction with the cursor keys or .

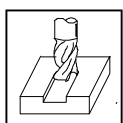


Movement in fast speed is analogous to feed movement. For this press the key and at the same time the corresponding cursor key for the wanted movement direction.



Fast speed movement is defined in the configuration and can be between 1000 and 10000 mm/min. **Collisions** and other errors are displayed in fast speed as well; however, it is natural enough workpart machining is not possible in this case.

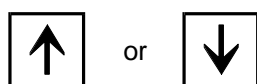
#### CNC Milling



With the milling CNC Simulator, the spindle head or machine table can be moved in **fast speed** or in **feed motion**. Please note that you can also enter machine commands in the Teach-In Mode (tool change, spindle, feedrate, speed).

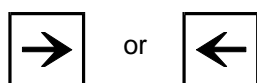


For the sake of clarity, we refer to one tool movement only in the following.

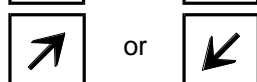


The tool is moved in the Z direction (according to G17) with the cursor keys or . If the spindle is switched on, the workpart can be machined and cutting is shown on the screen.

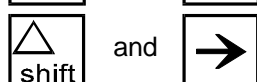
**Collisions** with the toolholder etc. are displayed as error messages.



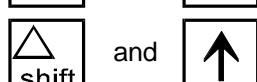
The tool is moved in the X direction (according to G17) with the cursor keys or .



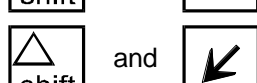
The tool is moved in the Y direction (according to G17) with the or key **on the numerical keypad** of the PC keyboard.



Movement in fast speed is analogous to the feed movement. Press the key on the PC keyboard for this and at the same time the cursor key or the corresponding key on the numerical keypad.



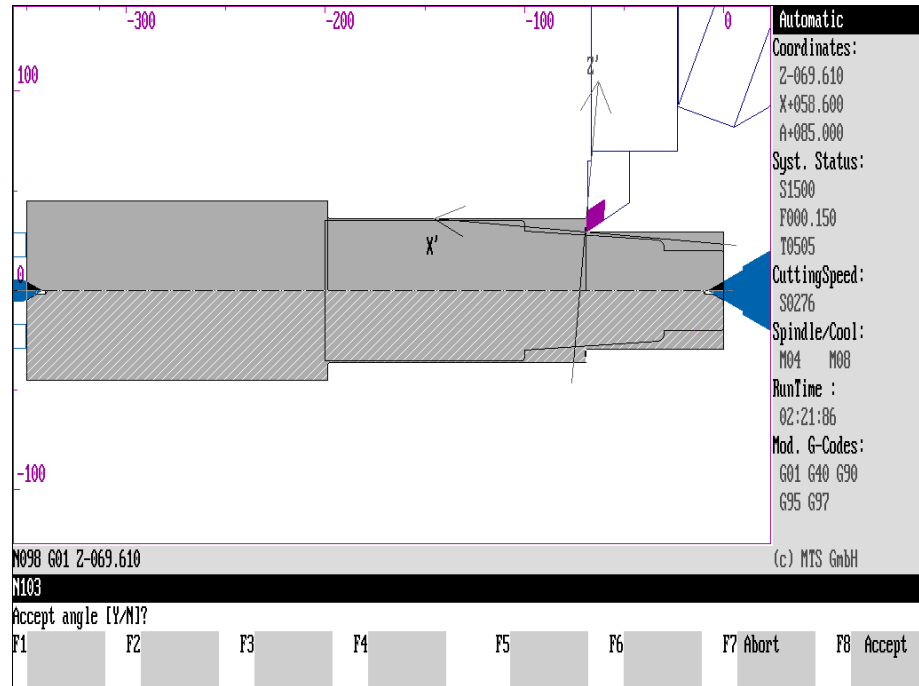
The fast speed is defined in the configuration and can be between 1000 and 10000 mm/min. **Collisions** and other errors are also displayed in fast speed; however, workpart machining is not possible.



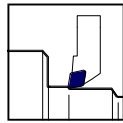
#### Note

Incorrect operations frequently result from spindle not being switched on, the spindle direction being wrong or from cutting edge collisions or the selected feed rate being too low.

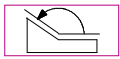
Figure 8.4.4-3:  
CNC Turning,  
Teach-In Mode;  
Angle Input for Manual  
Machining.



## CNC Turning: Angle Input



In the CNC Simulator turning, the tool slide can be moved not only parallel to the axes but also at a certain angle in **fast speed** or in **feed motion**. In this case 45° (or 115°, 225°, 315°) is set as standard for the angle, or you can also enter some other angle value. You then move the tool slide to the target position.



From the main menu of the Teach-In Mode you call the "Angle Input" submenu with **F2**. A second coordinate cross is then displayed at the tool tip based on the current entry under "**Angle**", indicating the current movement angle for the tool slide.

The angle input field in the information column is activated parallel to this and you can enter the angle size as desired:

[Angle Input]

Enter the angle for the the tool slide movement.



Then confirm this entry with **↵**. Based on your entry the new tool slide movement is displayed.

If you want to change the angle, repeat the entry.



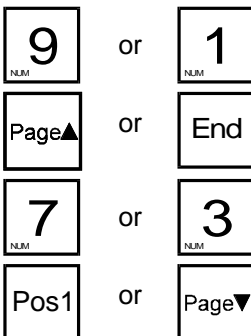
**Cancel:** If you have selected the function by mistake, you can cancel processing with **F7**.



**Accept:** To confirm the angle entry, return to the main menu with **F8**.

## Tool Movement Based on Set Angle

The keys of the switched-off numerical keypad (PC keyboard) are the movement keys for the tool slide which moves either at the standard angle or at the entered angle setting:



The tool slide is moved in the Z' direction with the **9** or **1** keys on the numerical keypad or **End** or **Page▲**. If the spindle is switched on, the workpart can be machined and the cutting is shown on the screen.

The tool slide is moved analogously in X' direction with the **7** or **3** keys on the numerical keypad or the **Pos1** or **Page▼** keys.

## Cancel Last Movement Command



If you make a mistake in manual machining, it is possible to cancel the last generated movement command in the main menu of the Teach-In Mode:

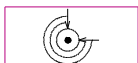


**Reject movement:** The last NC block is deleted with the function key **F5** and the displayed workpart machining is canceled.

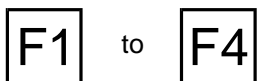
## Change Incremental Value

The step length (in mm) for the tool or machine table movement after pressing a direction key once is called "incremental value". The increment therefore defines the **movement unit** for machining in manual mode.

**Increment:** The CNC Simulator offers a choice between four incremental values: 1.0 mm, 0.1 mm, 0.01 mm or 0.001 mm. It is possible to switch between these incremental values in any machining situation.

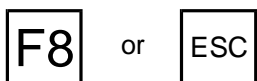


An increment is switched on either with the corresponding "increment key" - or via menu guidance with **F1**.



**Increment 1.0/0.1/0.01/0.001:** You choose the increment with the function keys **F1** to **F4**.

The CNC keys are marked with the corresponding values.



**Return:** Press **F8** or **ESC** to return to the starting situation.

### Note

The change of the incremental value is displayed in the information line.

## Special Functions

### Cross Reference

The functions for **changing the graphics display, zooming (CNC Turning), measuring (CNC Turning), 3D views** are discussed in detail in [Chapters 9](#) and [10](#). Therefore, for details refer to those chapters.

## Return to Interactive Programming

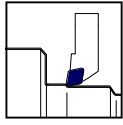


**End Teach-In Mode:** To continue in the interactive programming mode, switch back with the **F8** key.

You can then continue NC programming in the Interactive Programming Mode or in one of the other programming modes.

## 9. Measuring

### CNC Turning



For a **quality test** prior to production, **workparts to be machined by turning or contours programmed with the WOP system can be measured at any time**: either the contour elements are entered with all geometrical data (starting and end points, circle center points, lengths, transition angles etc.) or the contour points are measured from

the zero point with an accuracy of 1 µm. **Threadings** are also recognized and displayed.

#### Peak-to-Valley Heights

To test the surface quality, the theoretical **peak-to-valley height** of roughing is also calculated and displayed graphically for the machining passes. The maximum peak-to-valley height and the mean roughness value of the surface can be determined in accordance with DIN 4768.

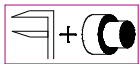
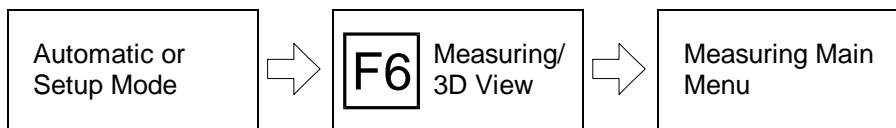


Apart from this quality control, the "Measuring" program function can also be used in CNC training to teach some principles of NC programming: for instance, contour deviations due to lack of radius compensation or radiusing of inner corners can be made transparent and plausible with the tool nose radius.

#### Note

The zoom function is available at any time during measuring as an additional support.

#### Selection:



You call up the measuring program function

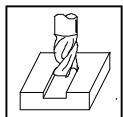
- in the **Setup Mode**,
- in the **Automatic Mode**,
- during **Interactive Programming** or
- during **Teach-In Programming**

with **F6**.

During **Workshop-Oriented Programming**, the measuring function is offered directly in the main menu, provided one geometry element was specified already. If you currently use some other function of the CNC Simulator, switch to the Automatic or Setup Mode menu first.

#### Note

### CNC Milling



Unlike the Turning Simulator, **only the contours programmed with the WOP can be measured in milling.**

Operation of the measuring function in Workshop-Oriented Programming is analogous to the measuring function of the Simulator for Turning described below.

Figure 9.-1:  
CNC Turning,  
Schematic of the  
Measuring Main Menu.

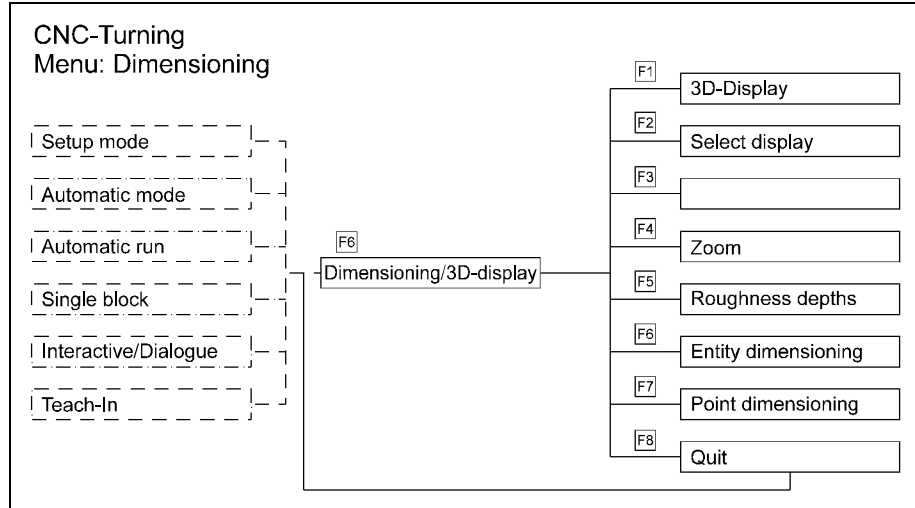


Figure 9.-2:  
CNC Turning,  
Automatic Mode,  
Processing in the  
zoom section;  
you call up the  
"Measuring/3D display"  
function with **F6**.

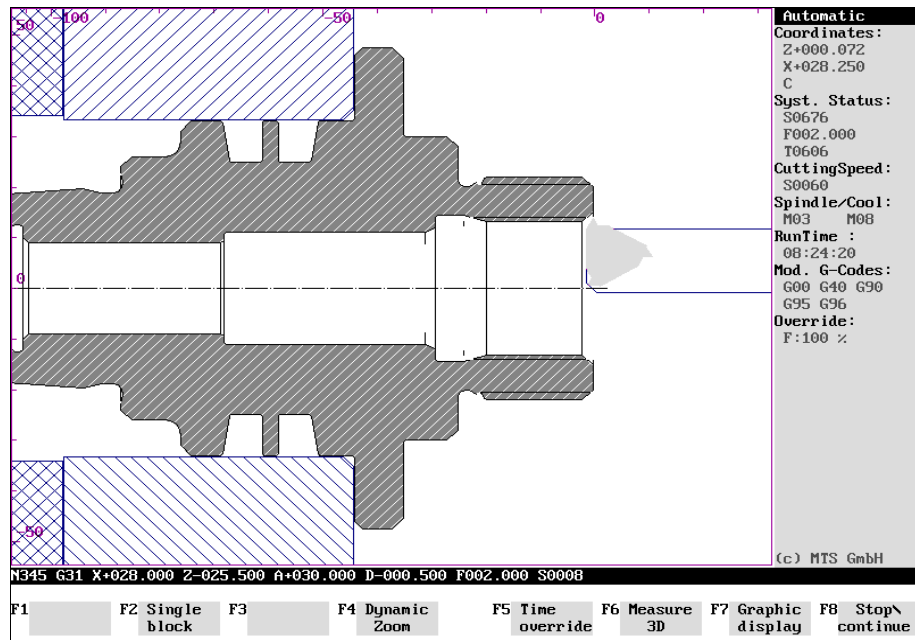
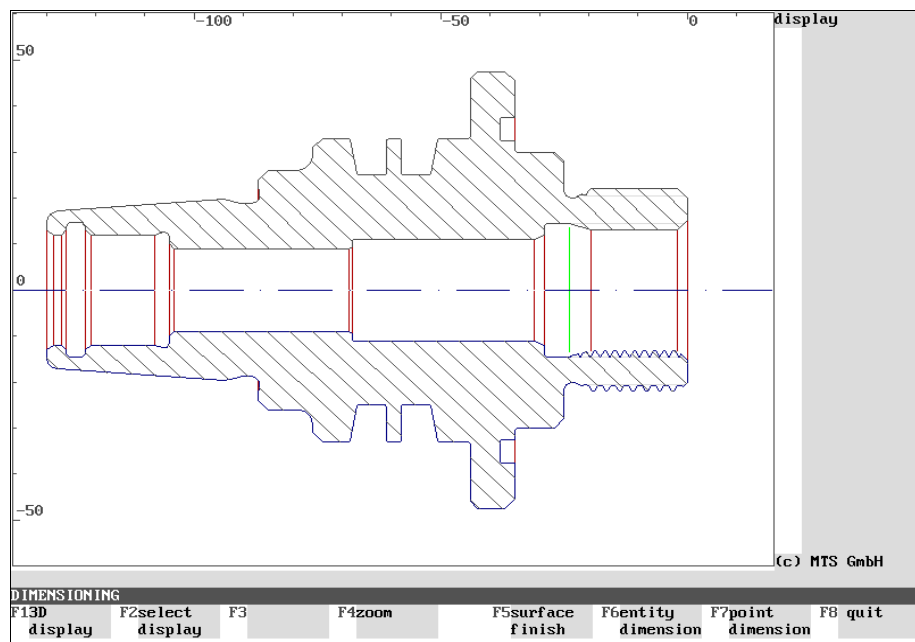


Figure 9.-3:  
CNC Turning,  
Measuring main menu;  
Center: Display of a  
workpart produced with  
the CNC Simulator  
(optionally as a cut-out  
and as different  
sectional views);  
Right: information  
column for dimension  
data;  
Below: function keys  
and dialogue lines.



## Menu: Measuring

### Display

In the measuring menu (cf. Figure), a **graphical display** showing the workpart contours, similar to a design drawing, is displayed in the screen window. The dot-and-dash line indicates the axis of rotation; workpart edges are marked as visible edges, internal corners which are rounded with the tool nose radius are marked as so-called "clearance edges" (lines which end in front of physical edges). The other screen areas are divided into an **information column** and a **function key menu** like in the Set-Up or Automatic Mode.

Proceeding from the "Measuring" menu, you can choose one of the following operation functions:

F1

**3D display:** You switch to the 3D view function with **F1**.

F2

**Select display:** You can choose between the display modes: **external view, full section, half section below, half section above** as well as **display of the thread** as symbol or as thread contour. Press **F2** to switch to the setting for these display modes (cf. below, Chapter 10).

F4

**Zoom:** If you want to enlarge or reduce the size of the displayed section, switch to the "Zoom" function with **F4** (cf. below, [Chapter 10](#)).

F5

**Peak-to-valley height:** The function "Calculate the peak-to-valley heights" is called with **F5**.

F6

**Element dimensioning:** In **element dimensioning**, the relevant geometrical data (starting and end point, transition angle, lengths, circle center points etc.) of the contour elements (lines or circular arcs) are output.

F7

**Point dimensioning:** In **point dimensioning**, the contour points (including the center points for existing circular lines), intersections or vertices are measured in relation to a predetermined dimensioning zero point.

F8

**Return:** With **F8** you return to the starting situation.



Now, try to use the different measuring subfunctions.

Figure 9.-4:  
Graphical symbols  
used for element and  
point dimensioning.

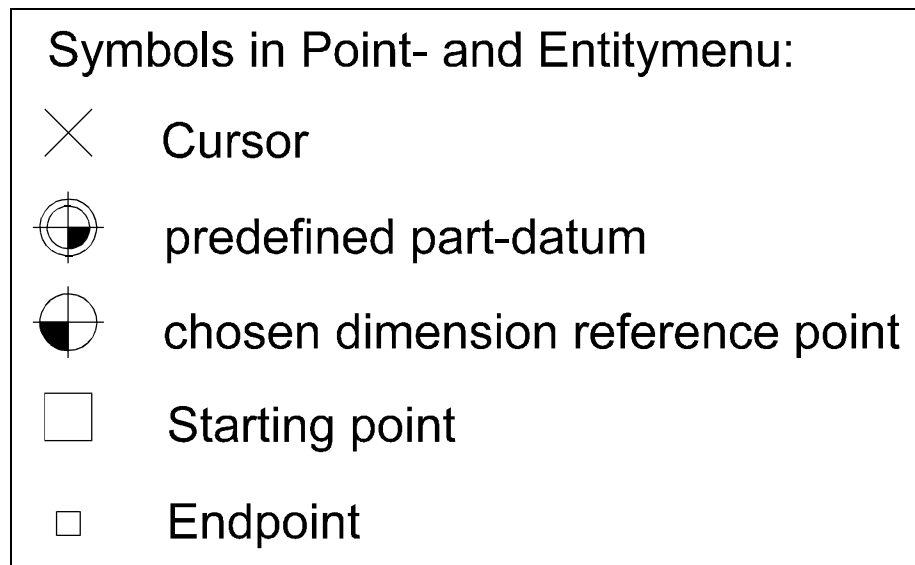


Figure 9.-5:  
Graphical marking of  
a contour segment in  
element dimensioning.

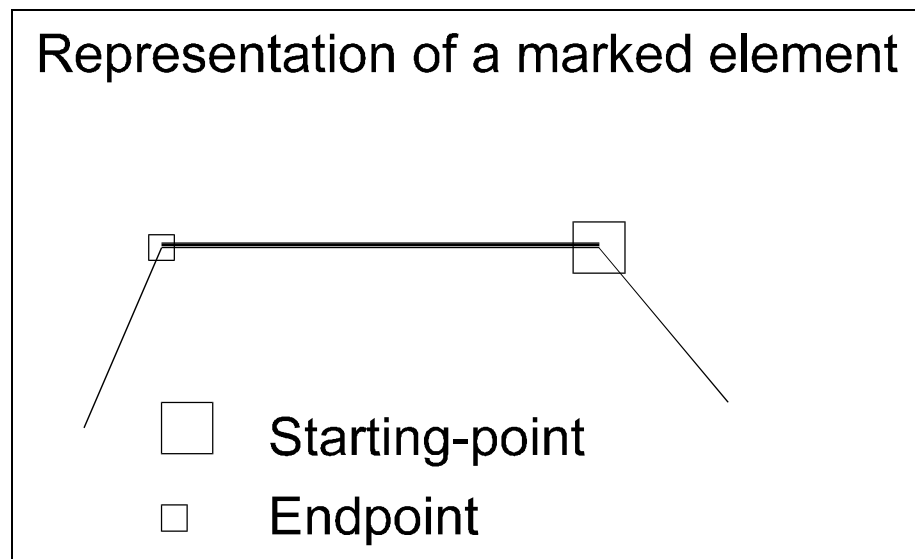
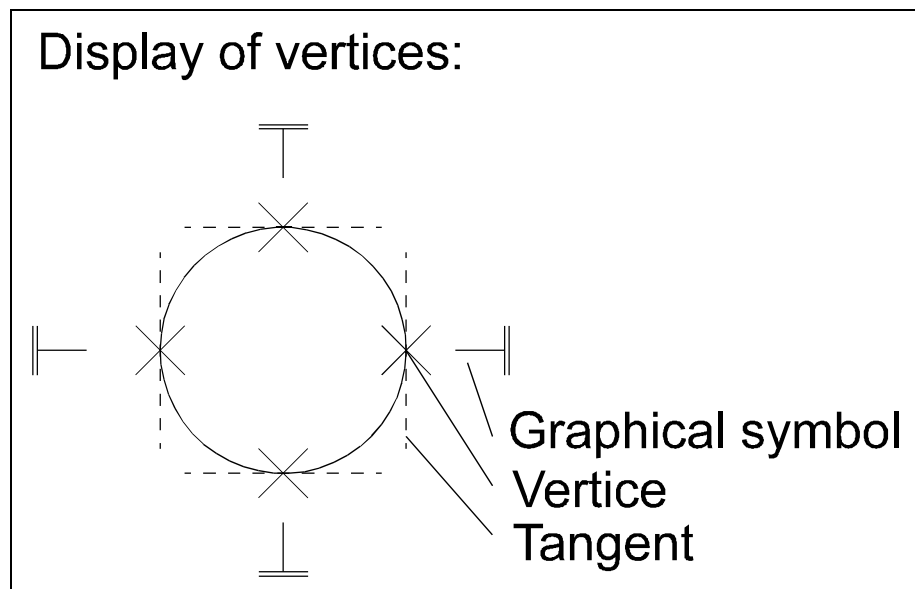


Figure 9.-6:  
Display and graphical  
symbols of the vertices  
in point dimensioning.





## Special Graphic Symbols and Markings

For the sake of clarity, additional graphic markings to emphasize the reference points and selected geometrical elements are used for element and point dimensioning of the screen display. The following symbols are used:

### Point Marking

The cursor position is indicated with " X " and marks the selected point. This can be within a geometry element or is freely selected with the cursor keys.

### Default or User-Defined Workpart Zero

The reference system of the measuring program is defined by two reference points: After program invocation, **the workpart zero point is pre-set** and corresponds to the programmed or manually set workpart zero point (cf. Setup, [Chapter 4](#)). This point is identified by the standardized symbol and also forms the origin of the displayed system of coordinates.

Should the pre-set workpart zero point differ from that of the pattern (drawing or NC program), it is possible to set a **dimension reference point**. In order not to distinguish it from the default workpart zero point, the symbol used (cf. Figure) is emphasized in color. **The dimension reference point is the starting point for all dimensioning data.**

#### Note

If the predetermined workpart zero point and dimension reference point are identical, they are placed on one another in the screen display.

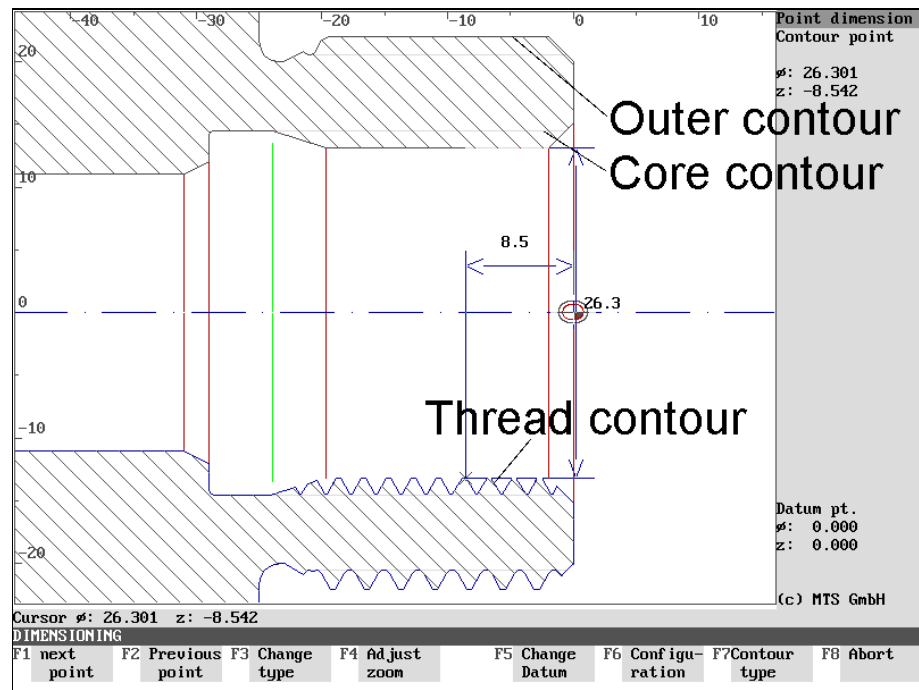
### Marking Elements

In element dimensioning, the selected element is emphasized in color and its starting and end point are indicated (cf. Figure). This "orientation" of the contour segment is arbitrary and does not depend on the production method of the actual workpart.

### Displaying Vertices

Circular lines are sometimes given as "vertices" in a drawing, and can, in this form, be analyzed in point dimensioning. The contact points of a horizontal or vertical tangent are vertices. The four assigned symbols (cf. Figure) appear in the text column and identify the corresponding vertex.

Figure 9.-4:  
CNC Turning,  
Point Dimensioning  
menu;  
Display of threads.





## Thread Contours

If there is no thread in the workpart, it is drawn as a closed contour. On the other hand, if there is one, then **three** closed contour lines exist:

- the thread contour,
- the outer contour and
- the core contour.

The course of these contour lines differs only in the area of the thread. During dimensioning, these contour lines can be activated one after another with the "Contour Type" function (see below).

## Active Contour for Dimensioning

The contour displays show the workpart as a mirror-image above and below the rotation axis. During dimensioning, either of these two sections is active and can be "tracked" with the cursor. To change the active contour, position the cursor with the cursor keys  or  above or below the rotation axis and continue dimensioning. The currently active contour is highlighted in color.



Since the **Figures** in this manual are monochrome, it is not always possible to reproduce the color distinctions available on the screen.

Figure 9.1-1:  
CNC Turning;  
Schematic of the  
Element Dimensioning  
menu.

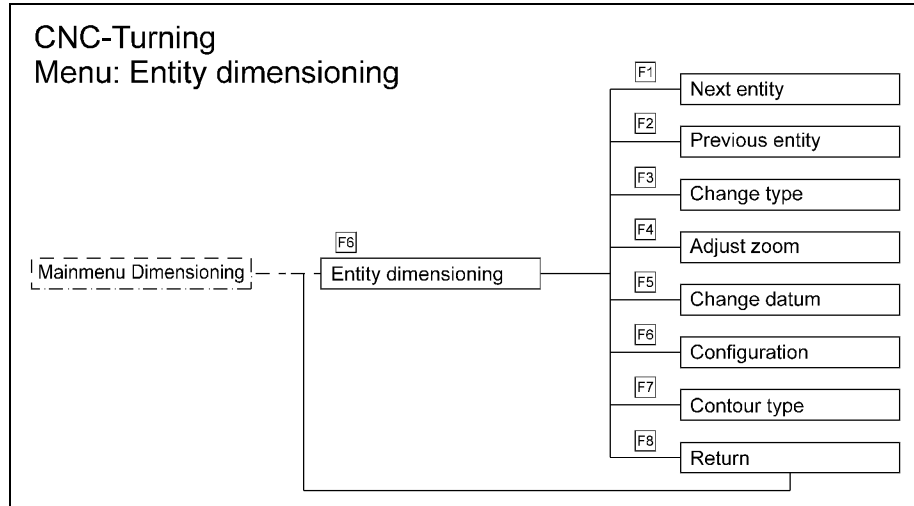


Figure 9.1-2:  
CNC Turning,  
Element  
Dimensioning;  
Measuring a straight  
line in enlarged  
representation,  
Half Section;  
in this Figure,  
reference is also made  
to the length and the  
gradient angle.

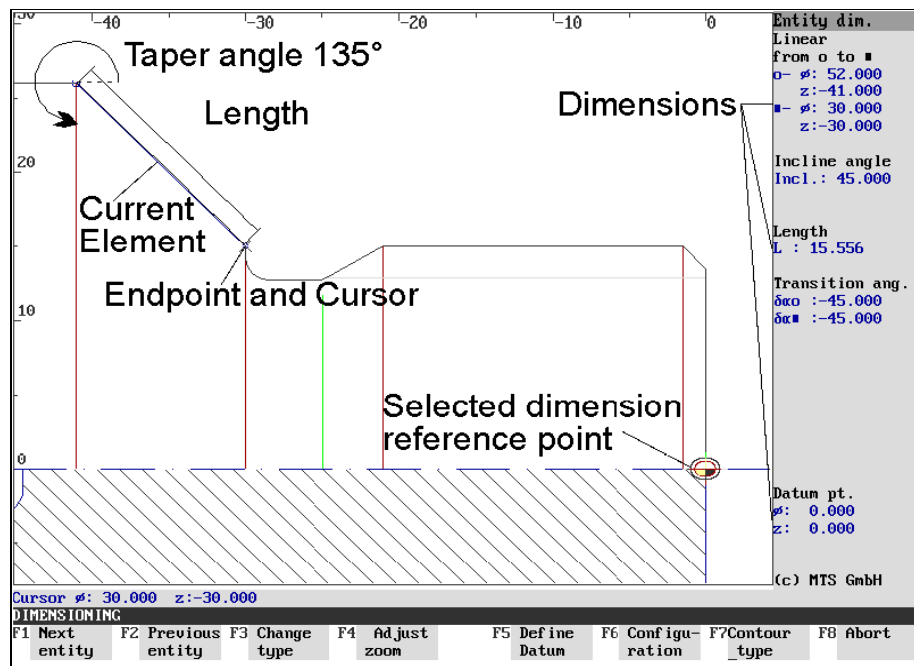
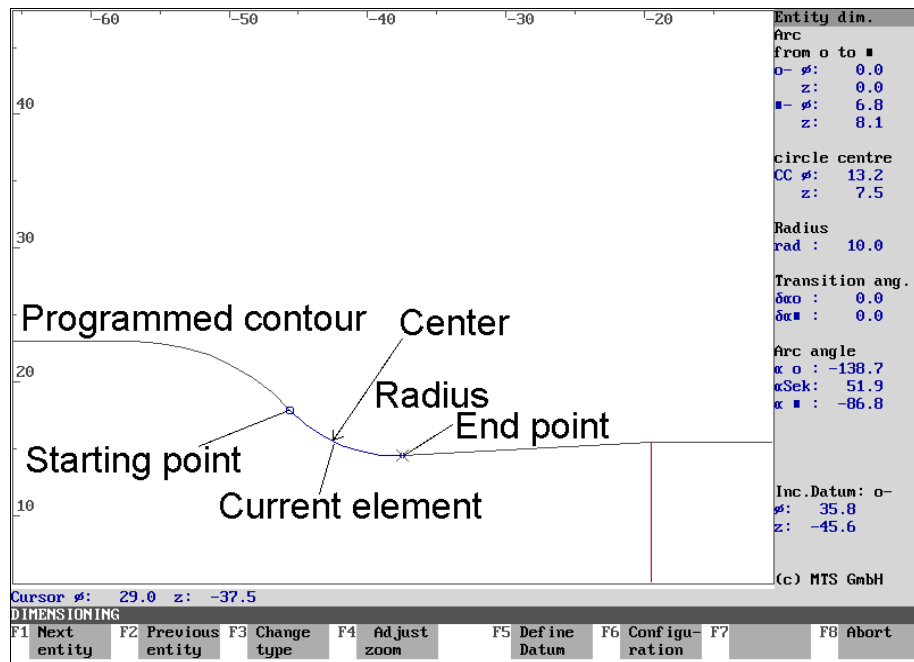
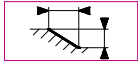


Figure 9.1-3:  
CNC Turning,  
Element  
Dimensioning;  
Incremental  
dimensioning of a  
circular line  
programmed with the  
WOP, enlarged; in this  
Figure, reference is  
made additionally to  
the center point, the  
radius and the  
specified zero point.



## 9.1 Element Dimensioning



To start dimensioning contour segments, it is necessary to display at least one geometrical entity (contour element).

### **Straight lines and/or circular lines**

can then be measured as elements with the accuracy of 1  $\mu\text{m}$ . Dimensioning can be effected in the **absolute** or **incremental** system (by shifting the dimension reference point).

In the graphic representation of the workpart or of its contour, the currently selected element is marked in color and its starting and end point are marked by the corresponding symbols (cf. Figure).

In the text column next to it, the relevant dimensions are displayed:

- Line:**
- Starting and end point** (in each case according to the diameter and Z coordinates),
  - Gradient angle** (related to positive rotation axis and orientation)
  - Length,**
  - Transition angle:** Angle between the extension of the preceding element and the current element counter-clockwise;
  - :** Angle between the extension of the current element and the following element counter-clockwise, (note orientation); tangential connection:  $0^\circ$ ),
  - Zero point** (coordinates of the dimension reference point);
- Arc:**
- Starting and end point** (in each case according to the diameter and Z coordinates),
  - Center point** (diameter and Z coordinates),
  - Radius,**
  - Transition angle** (between the preceding and subsequent element (note orientation); tangential connection:  $0^\circ$ ),
  - Circular angle** (1. Starting point: Angle between the positive rotation axis and the line from the circle center point to the starting point; 2. Angle of the circular sector; 3. Angle between the positive rotation axis and the line from the circle center point to the end point),
  - Zero point** (coordinates of the dimension reference point).

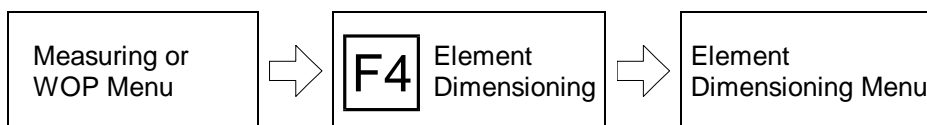
### **Note**

Angles are shown counter-clockwise in degrees, values exceeding  $180^\circ$  are shown as complements of  $360^\circ$ . Correspondingly, for instance, the value  $-10^\circ$  is displayed instead of  $350^\circ$ .



**When entering angle values consider the orientation of the selected element.**

### **Selection:**



First select the appropriate display mode of the workpart so that the dimensioning required can be performed (cf. above). Then select Element Dimensioning with the function key **F6**; the menu for Element Dimensioning is then loaded.

Figure 9.1-4:  
CNC Turning,  
Element  
Dimensioning;  
Dimensioning a line in  
a zoomed detail  
window; in this Figure  
the definition of the  
transition angles is  
emphasized.

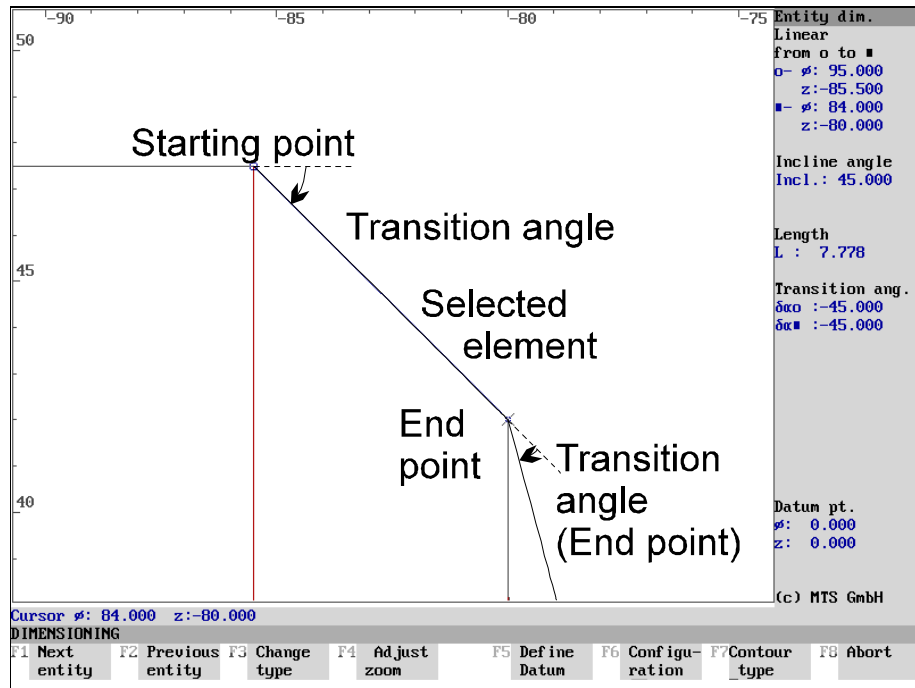
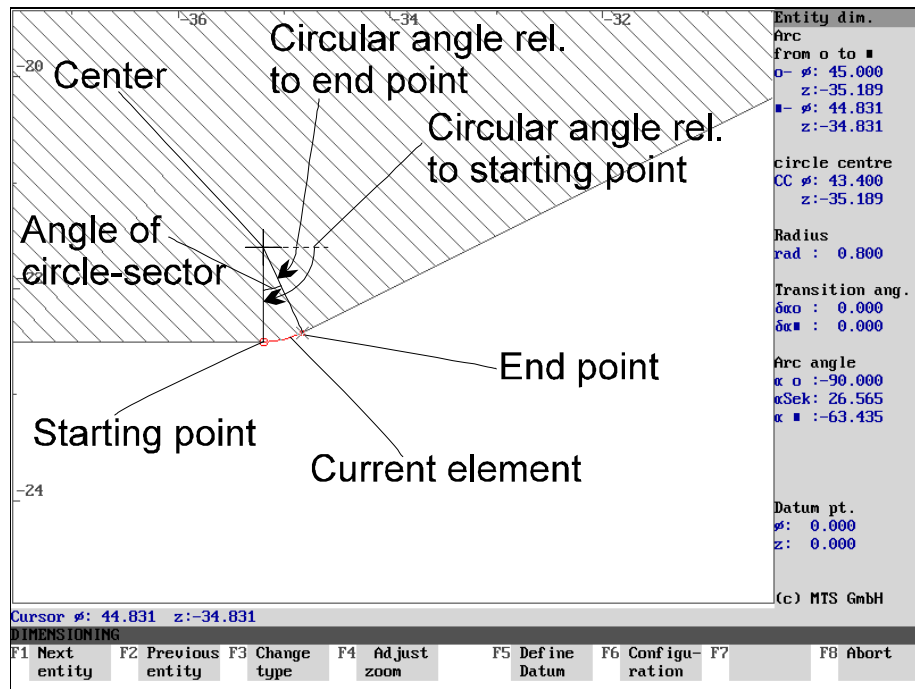


Figure 9.1-5:  
CNC Turning,  
Element  
Dimensioning;  
Example for  
dimensioning a circular  
line in a zoom section  
window; in this Figure  
reference is made  
additionally to  
determining the  
circular angle.



## Menu: Element Dimensioning

After the first invocation of this menu, the dimension reference point is identical with the default workpart zero point.

**F1**

**Next element:** Depending on the selected "element type", the element closest to the cursor is measured after the initial invocation. After that the elements coming next in the contour line are measured. These values specify the sense of rotation.

**F2**

**Previous element:** To select the preceding element, press **F2**.

**F3**

**Change type:** Either **only straight lines** or **circular lines** or **both straight lines and circular lines** can be measured with Element Dimensioning. You select the "element type" with **F3**; the corresponding selection criterion is then displayed in the dialogue line.

### Note

If the selected element type is not available, this is indicated by the message: "**Region empty**". Confirm this message with any key and then select another selection criterion.

**F4**

**Adapt zoom:** The workpart display can be modified even during dimensioning. A detailed presentation of this function is given in [Chapter 10](#).

**F5**

**Define zero point:** To define the dimension reference point anew, call up the corresponding submenu with **F5**, cf. below.

**F6**

**Configuration:** In this configuration, colors of the screen display as well as the number of decimal places for the dimension values can be changed. A detailed description of this function is provided in [Section 9.5](#).

**F7**

**Contour type:** This function is available only if there is a thread available. You switch between the thread, surface and core contour with **F7**.

**F8**

**Return:** Return to the main menu with **F8**.

To be able to select elements in complex contour segment strings quickly, it is possible to position the cursor manually as well, as an alternative to the steps discussed above as follows:



or





Move the cursor vertically with the cursor keys  or .



or






Move the cursor horizontally with the cursor keys  or .



or



Move the cursor diagonally with the ,  or  keys.

**Pos1**

### Note

As already mentioned, dimensioning of an already programmed workpart takes normally place above the rotation axis. Should you, however, wish to measure points below this axis, position the cursor below the Z axis first.

Figure 9.2-1:  
CNC Turning;  
Schematic of the Point  
Dimensioning menu.

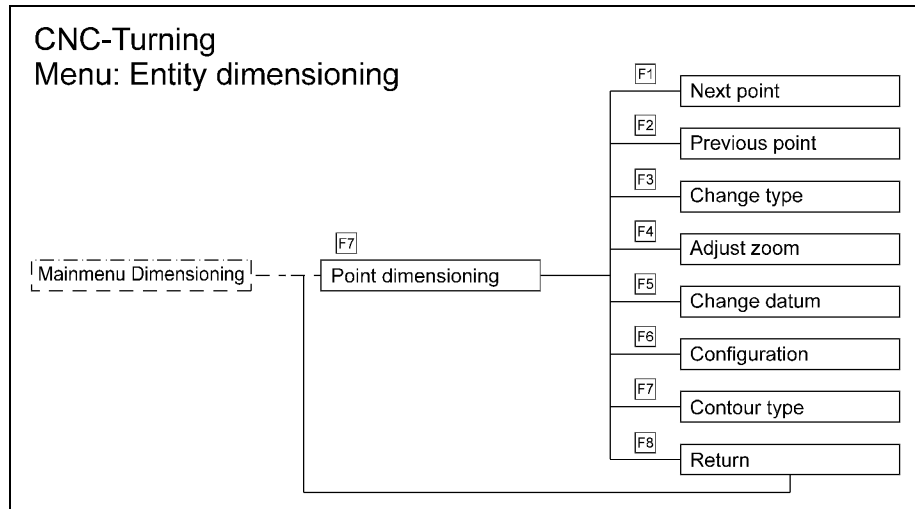


Figure 9.2-2:  
CNC Turning,  
Point Dimensioning;  
Dimensioning a thread,  
sectional display. To  
define the thread pitch,  
a temporary dimension  
reference point has  
been set.  
In the point  
dimensioning, a  
contour point is  
marked, a  
dimensioning frame  
is displayed and the  
values are displayed in  
the text column.

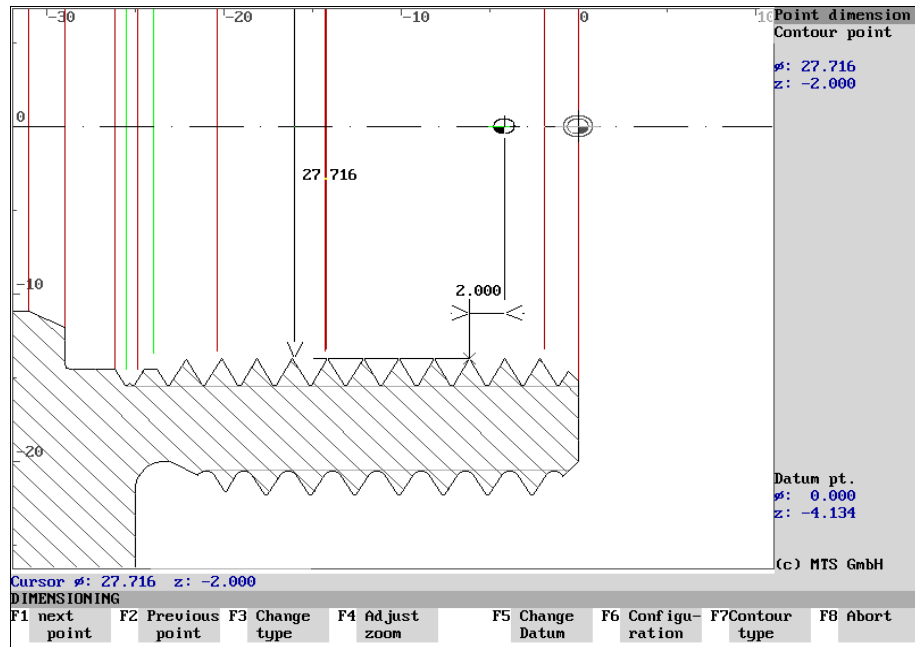
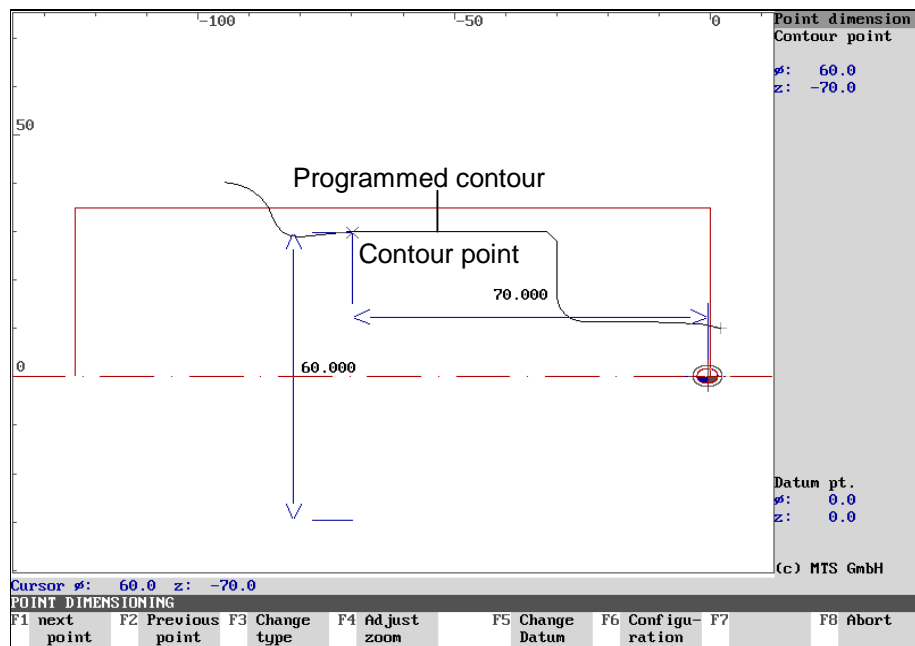
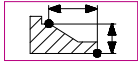


Figure 9.2-3:  
CNC Turning  
Point Dimensioning;  
Dimensioning a  
contour point  
proceeding from the  
WOP interface; in this,  
Figure reference is  
made additionally to  
the programmed  
contour line.





## 9.2 Point Dimensioning



For point dimensioning it is necessary to have at least one contour point displayed. After that, points such as:

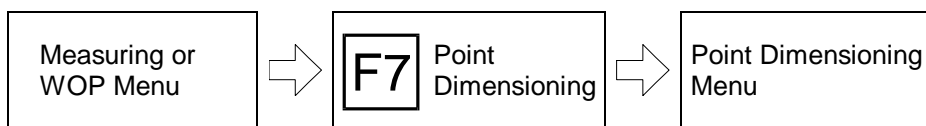
**contour points, intersections or vertices**

can be measured with the accuracy of 1  $\mu\text{m}$ . The default dimensioning mode is **absolute**, and it is possible to perform **incremental** dimensioning by shifting the dimension reference point correspondingly.

In the graphical display of the workpart or of the programmed contour, the cursor location is identified with a " X " and the dimensioning frame is displayed (cf. Figure). In addition to this, the complete dimension data is displayed in the text column next to it:

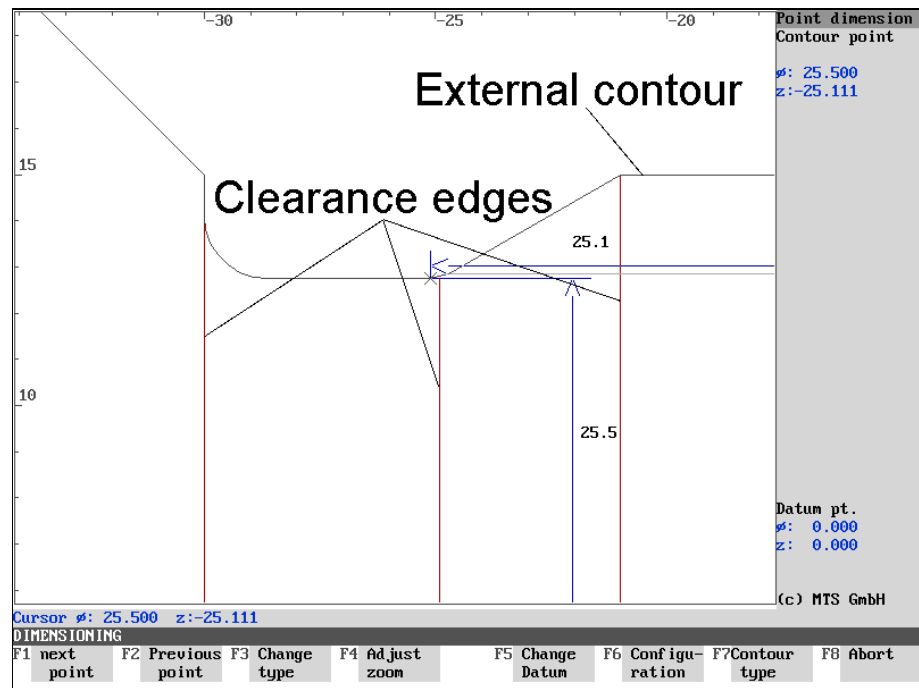
<b>Contour point:</b>	<b>Coordinates</b> (diameter and Z coordinates), <b>Zero point</b> (coordinates of the dimension reference point);
<b>Intersection:</b>	"Intersections" are called the intersection points of two lines connected by a third element (e.g. a chamfer or radiusing) (dimensions as above) in a section;
<b>Vertex:</b>	"Vertices" are called the contact points of a horizontal or vertical tangent (Dimensions as above). The symbols indicate the position of the tangent as well (cf. above)

Like in Element Dimensioning, there are the functions for changing the graphical display or stating the dimension values (decimal places) available, in addition to the option to select the type of the point and the definition of the zero point selection:



Proceeding from the current main menu, select Point Dimensioning with the function key **F7**. Before you call up Point Dimensioning, select first, if necessary, the appropriate display mode to be able to perform the dimensioning as required (cf. above).

Figure 9.2-4:  
CNC Turning,  
Point Dimensioning;  
Example for point  
dimensioning in a  
zoomed detail window.



## Menu: Point Dimensioning

After the first invocation of this menu, the dimension reference point is identical with the default zero point of the workpart.

F1

**Next point:** Depending on the selected "point type", the point closest to the cursor is activated for dimensioning after the initial invocation, and after that the elements coming next in the contour line. These values define sense rotation.

F2

**Previous point:** To select the preceding point, press **F2**.

F3

**Change type:** as already mentioned, contour points, intersections and vertices can be measured. Select the "point type" with **F3**; the current selection criterion is displayed in the dialogue line.

### Note

If the selected point type is not available, the message: "Section empty" is displayed. Confirm this with any key and select a further selection criterion.

F4

**Adapt Zoom:** during dimensioning, the zoomed section of the display can be changed. A detailed description of this function is given in Chapter 10.

F5

**Define Zero Point:** to re-define the dimension reference point, call up the corresponding submenu with **F5**. A detailed description of this function is provided below.

F6

**Configuration:** in this configuration, the dimensioning frame can be shifted, the number of decimal places for the dimensions as well as the colors of the screen display can be changed (cf. below).

F7

**Contour Type:** This function is available only in case of threading. You can switch between the thread, surface and core contour with **F7**.

F8

**Return:** You return to the main menu with **F8**.

In addition to the operation steps given above, it is possible to position the cursor manually. This is necessary, for instance, for shifting the dimensioning frame or defining the zero point.

↑

or

↓

Move the cursor vertically with the cursor keys **↑** or **↓**.

→

or

←

Move the cursor horizontally with the cursor keys **→** or **←**.

↗

or

↘

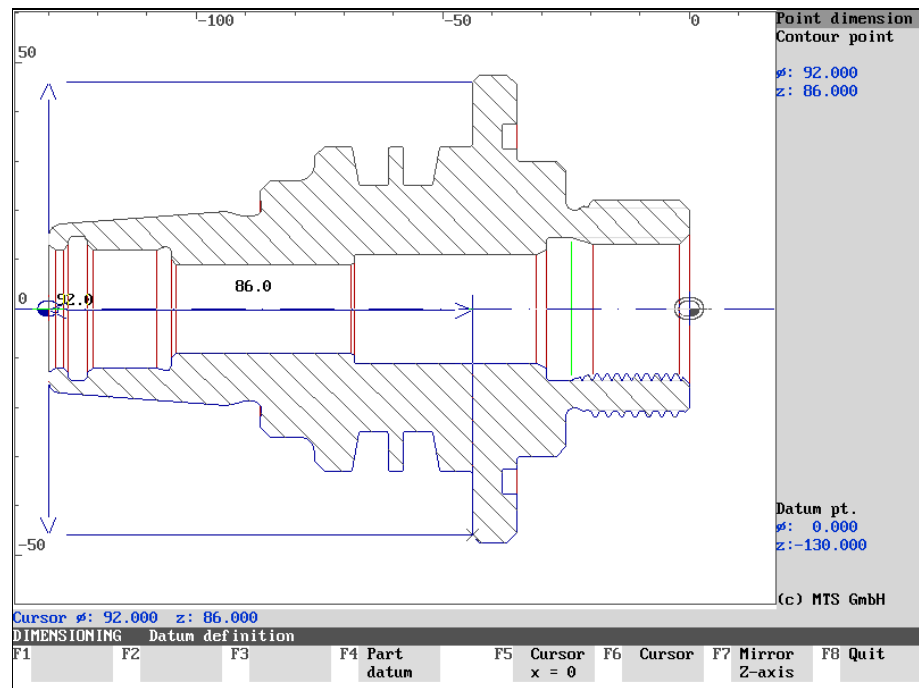
Move the cursor diagonally with the **PageDown**, **PageUp** or **<HOME>** keys.

Pos1

### Note

As already mentioned, a programmed workpart is normally dimensioned above the rotation axis. Should you, however, want to measure the points below this axis, place the cursor below the Z axis first.

Figure 9.3-1:  
 CNC Turning,  
 Menu for shifting the  
 Dimension Reference  
 Point;  
 the location is selected  
 with the function keys.  
 In this example, the  
 Dimension Reference  
 Point was set with  
 <F5> to the point X=0,  
 Z=-130.



## 9.3 Defining Dimension Reference Point

In each dimensioning menu there is a **default workpart zero point** which is identical with the programmed or manually set zero point. If the reference system of the drawing or of the NC program is not identical with this specified zero point (or if you want to dimension a detail, e.g. a thread, separately), a new **dimension reference point** can be determined during measuring. This then defines the zero point for the following measured data.

The specified workpart zero point is marked with a standard symbol and it is the origin of the displayed coordinate system. The symbol of the dimension reference point (cf. above) is displayed in color offset from it.

### Procedure

Determining a new dimension reference point is identical in Element and Point Dimensioning and consists of three operation steps:

**F1** or **F2**

First, set the cursor in the Element or Point Dimensioning menu on the point which is to be defined as the new dimension reference point. For this purpose move the cursor - either with the function keys **F1** or **F2** or the cursor keys - to the corresponding position.

**F5**

Then press **F5** to call up the submenu with which you redefine the dimension reference point:

### Menu: Zero Point Definition

You select the zero point in this menu:

**F4**

**Workpart Zero Point:** When **F4** is operated, the workpart zero point and dimension reference point coincide (again).

**F5**

**Cursor X = 0 :** With **F5** the Z value of the cursor position is taken over as Z coordinate of the dimension reference point and the X value is set to zero.

**F6**

**Cursor:** With **F6** the dimension reference point is set to the current cursor position.

**F7**

**Mirror in Z axis:** With **F7** the current dimension reference point is mirrored in the Z axis.

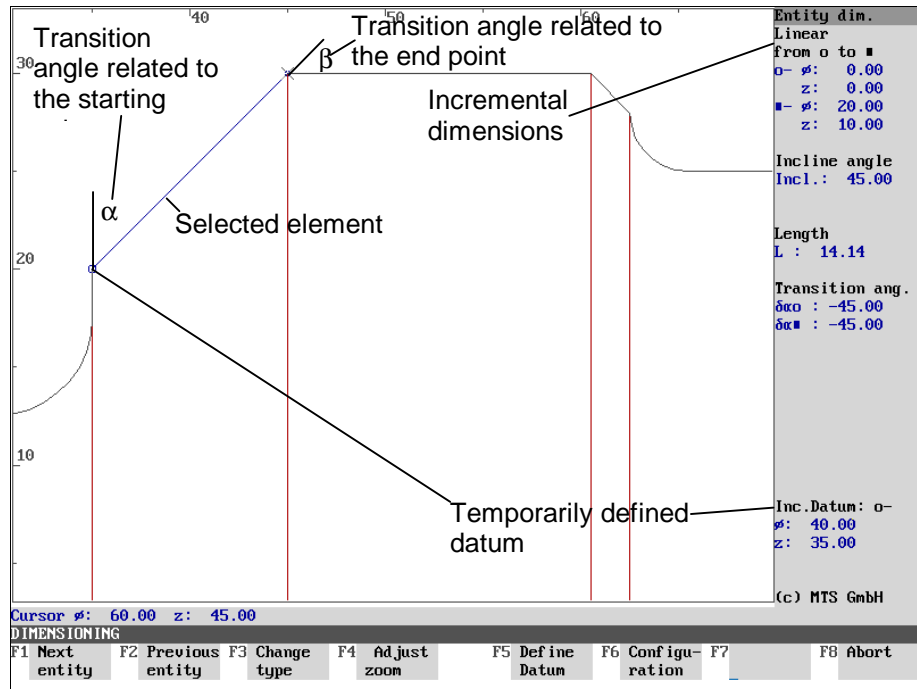
**F8**

**Return:** Press **F8** to return to Element or Point Dimensioning without change.

### Note

There are three further functions in Element Dimensioning to shift the zero point for incremental dimensioning:

Figure 9.3-2: CNC Turning, Incremental Dimensioning with provisory zero point. The workpart has been reclamped previously. The calculation of the transition angles considers this orientation; they are given as negative values here (clockwise).



## Incremental Dimensioning

Before doing increment measuring in **Element Dimensioning**, select the element which is to be dimensioned incrementally. If you then call up the zero point definition, the following three possibilities are available in addition to the above described assignments:

F1

**Starting point:** (this function is available only if an element is currently selected) if you confirm with **F1**, the zero point is placed temporarily onto the starting point of the marked element. This zero point shift can be performed only once.

F2

**End point:** (this function is available only if an element is currently selected) with **F2** the zero point is placed temporarily on the end point of the marked element. This zero point shift can be performed only once.

F3

**Center point:** (this function is available only if a circular line is currently selected) with **F3** the zero point is placed temporarily on the center point of the marked circular line. This zero point shift can be performed only once.

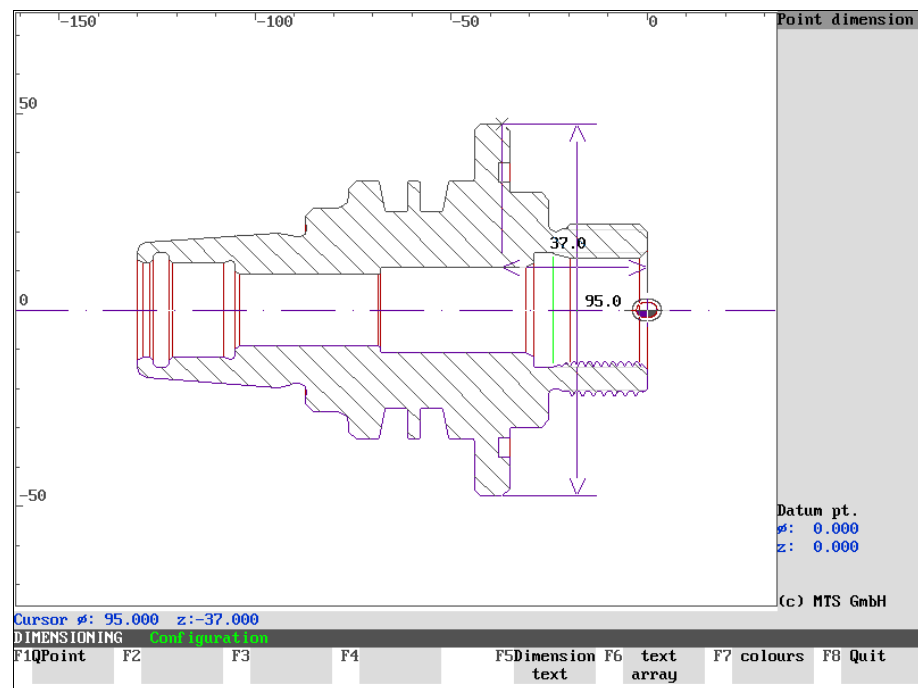
### Note

If you want to dimension several elements or points incrementally, you need to redefine the zero point in each case.



To exchange the starting and end point of the dimensioning, it might be necessary to reclamp the workpart in the Setup Mode first.

Figure 9.4-1:  
Menu for the configuration of the program functions of measuring and Workshop-Oriented Programming:  
In the configuration you can modify the dimensioning frame in Point Dimensioning ("Q point"), the statement of the decimal places ("dimension text" and "text column") as well as the color settings.





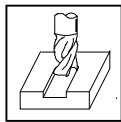
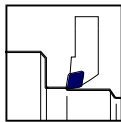
## 9.4 Measuring Configuration: Change Displays

The **Measuring** program function as well as the **Workshop-Oriented Programming** have the same **configuration**, which allows to change certain presettings of the graphical or text display.

### Password

After concluding the configuration menu, the changes are stored in a configuration file and then considered in the subsequent processing. Since the configuration program of the CNC Simulator can be secured with a "**password**", the password prompt is displayed after of the configuration has been concluded. After entering the correct password, the changes are protected for all times, otherwise they apply only for the current program run.

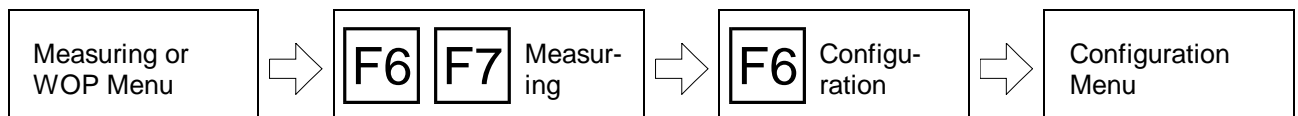
### Note



The configuration described below concerns the graphic display and the text/value entries of the Measuring program function as well as Workshop-Oriented Programming. It is separated from other configuration procedures of the CNC Simulator (please refer to the [Configuration Instructions of the CNC Simulator for Turning and Milling](#)).

**Should the turning and milling CNC Simulators both be installed on your computer, the changes made apply for the WOP interface in both systems.**

### Selection:

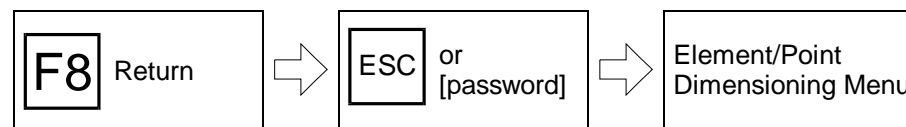


Element or Point Dimensioning is first called up with **F6** or **F7** from the main menu of Workshop-Oriented Programming as well as from the Measuring main menu. You then switch to the Configuration menu with the function key **F6**.

### Note

In Workshop-Oriented Programming, the dimensioning functions are not available until at least one geometrical element has been defined.

### Return:



**F8**

**ESC**

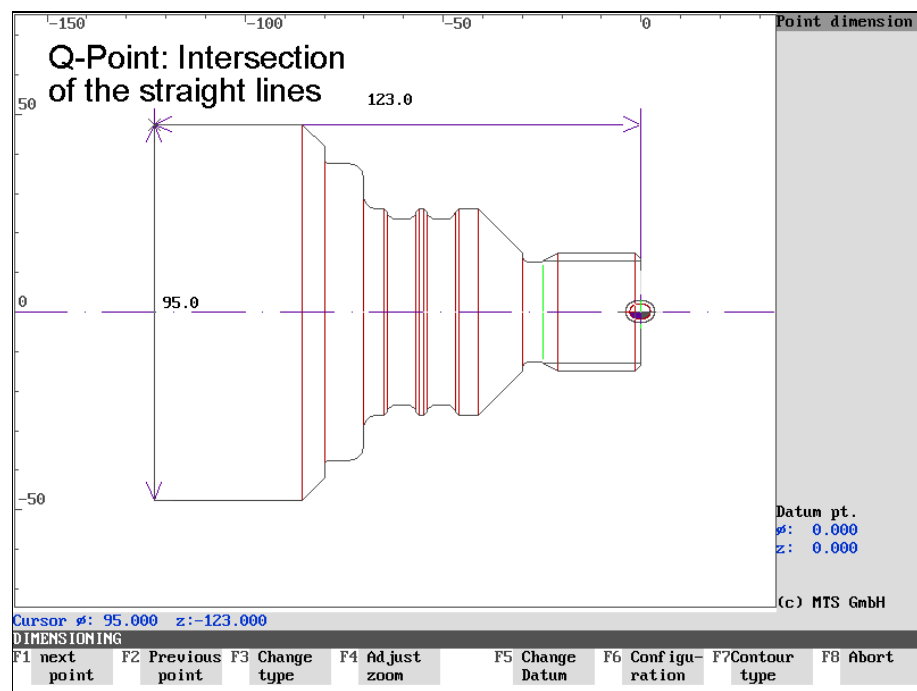
or [Password]

With **F8** you end the configuration. In the subsequent dialogue you either **enter the password** or confirm it with **ESC** or **F8** (cf. above).

### Note

The changes of the configuration are not considered in full extent until you have returned to Element or Point Dimensioning. Color changes, however, are valid immediately.

Figure 9.4-2:  
The so-called "Q-Point" is the intersection of the straight lines running through the two dimension lines in Point Dimensioning. This Figure shows the situation after repositioning the Q-Point.



## Positioning the Dimensioning Frame: "Q Point"

In **Point Dimensioning**, **dimensioning frames** are displayed in the graphic window, in addition to the dimension data displayed in the text column indicating the cursor position relative to the dimension reference point. As this data might overlap other contour elements, the dimension lines can be shifted as required. The so-called "Q Point" is decisive for the position of the dimensioning frame.

The intersection of the two straight lines going through the two dimension lines is called "**Q Point**". To shift the dimensioning frame a new Q Point must be defined.

### Procedure



or



or



**Before** you call up the configuration to define a new Q Point, first move the cursor with the cursor keys or or or to the the desired new intersection of the dimension lines.

Then call up the configuration with .

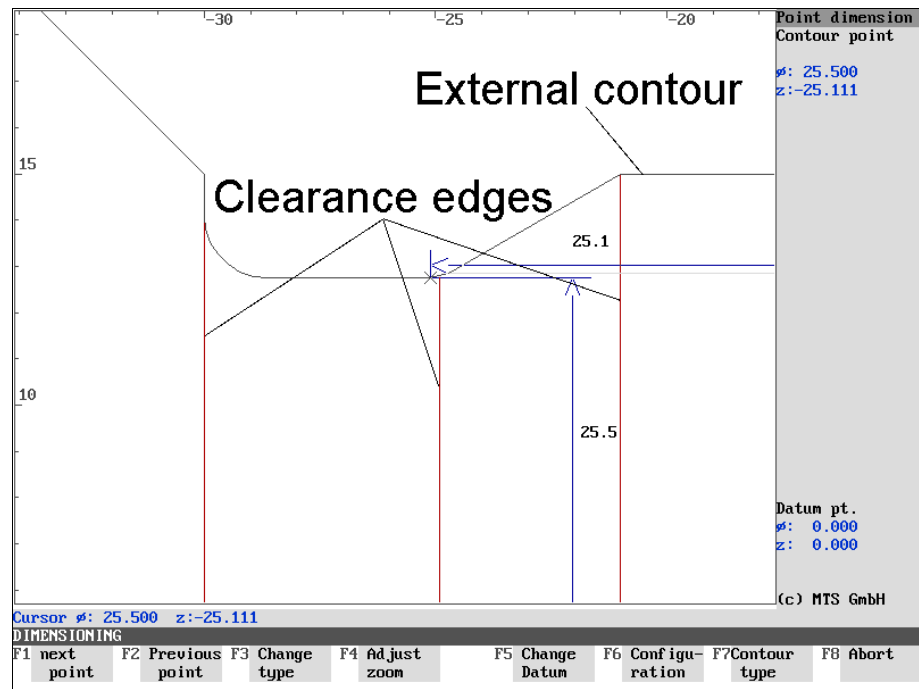
In the configuration menu, the Q Point is placed on the cursor position with the function key . The message: "**Intersection of the dimension lines set on cursor**" is displayed for this.

You can then make further changes in the configuration or you can return to the starting situation by pressing the key by entering the password or .

### Note

Positioning the Q Point is relevant only for Point Dimensioning.

Figure 9.4-3:  
CNC Turning,  
Measuring;  
Dimension data with  
the accuracy of  $10^{-3}$   
mm.



## Places after the Decimal Point of the Dimension Values: Dimension Text and Text Column

In dimensioning, the dimension values can be displayed without any places after the decimal point or with the accuracy of a tenth, hundredth or thousandth of a millimeter. You define the "format" of the dimension data for the dimensioning frame under the menu item "dimensioning text" and that for the text column under "text column".

### Procedure

**F6**

If the configuration is not loaded yet, call it up with **F6**.

**F5**

**Dimension text:** To change the format of the value data in the dimensioning frame, continue with the function key **F5**.

**F6**

**Text column:** To change the format of the value data in the text column, continue with **F6**.

In both cases a further menu is then loaded with the following options:

**F1**

**0 Dec \$:** With **F1** the values are displayed with no places after the decimal point.

**F2**

**1 Dec \$.\$:** With **F2** the values are displayed with one place after the decimal point.

**F3**

**2 Dec \$.\$\$:** With **F3** the values are displayed with two places after the decimal point.

**F4**

**3 Dec \$.\$\$\$:** With **F4** the values are displayed with three places after the decimal point.

### Note

As soon as you have defined the format, the program returns to the configuration menu and confirms the made definition with the message: **"The dimension text (or the text block) now has 0 (1) (2) (3) places after the decimal point"**.

**F8**

If you decide not to make any format changes, return with **F8** to the configuration menu.

You can then make further changes in configuration, or you return to the starting situation by pressing **F8** and entering the password or **ESC**.

### Note

**Determining the format for the "dimension text" is relevant only for Point Dimensioning.**  
**The text columns of Point and Element Dimensioning are configured together.**

Figure 9.5-1:  
CNC Turning;  
Schematic of the  
Peak-to-Valley Height  
menu.

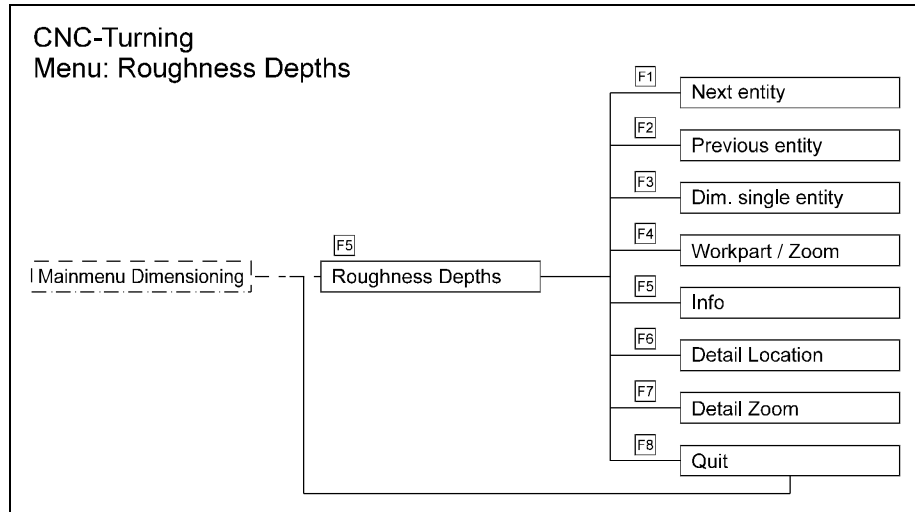


Figure 9.5-2:  
CNC Turning,  
Menu for calculating  
the Peak-to-Valley  
Height.  
Overview of the  
Contour Elements.

Part info depth of roughne

The blank contains :  
74 entities of which  
39 machined  
35 with depth of roughness >0

applied colors:  
machined selected  
not machined selected  
nominal contour selected  
rough contour selected

cursor to continue  
or function keys

(c) MTS GmbH

sngl.entity				middle					
previous entity	F3 dim. single entity	F4 Workpart Zoom	F5 info	F6 detail location	F7 detail zoom	F8 quit	F1 next entity	F2 previous entity	

## 9.5 Surface Roughness: Peak-to-Valley Height

### Surface Quality

Every workpiece surface has a certain **roughness**. It is never identical with the geometrically ideal surface, and correspond to it only within a certain tolerance range. Normally the average or maximum peak-to-valley height, which characterizes the deviation tolerance of the workpart geometry from the ideal surface quality, is indicated in the technical drawing.

### Quality Control

In production practice, the quality of the workparts is recorded among other things by measuring the **peak-to-valley height profile** indicating the surface quality. Analogously, the turning CNC Simulator enables quality control prior to production in that the **theoretical peak-to-valley heights** are recorded and displayed for each machining operation. In DIN 4768, the maximum peak-to-valley height  $R_{max}$  and the mean roughness value  $R_a$  are specified as follows:

### DIN 4768

**$R_{max}$** : the largest peak-to-valley height within a measuring distance is designated as "maximum peak-to-valley height";

**$R_a$** : the arithmetic mean of all deviations of a peak-to-valley height profile from the "center line" is designated "mean roughness value". A center line divides the roughness profile so that the area of all peaks is identical to that of the valleys.

### MTS

Deviating from this definition, the mean roughness value  **$R_a$**  is calculated in the MTS Simulator not with regard to the center line but to the "nominal contour". The following therefore applies for the CNC Simulator:


**$R_a$** : is the arithmetical mean of the deviations between the nominal and theoretical actual contour. This value is larger than that according to DIN.

In addition to the peak-to-valley height data, the coordinates of the starting and end point as well as of the zero point and for lines also the line length are displayed as well. Single elements (cf. Element Dimensioning) or several adjacent elements can be used as measuring lines. Elements with peak-to-valley height zero are not included in the calculation of  **$R_a$** .

The actual roughness of a workpart is a result from various factors. The mathematical model of the turning Simulator considers only the deviations resulting from the shape of the tool cutting edge and from the selected feedrate. Threads have no peak-to-valley height in this model and the peak-to-valley height calculation simulates an "accurate stop" at the end of each element, i.e. the tool stops for an entire spindle revolution.

### Note

In certain cases, the peak-to-valley calculation gives an incorrect result , e.g. if

- tools are no longer available for workpart machining (i.e. they have been removed from the turret),
- the calculation of the peak-to-valley heights was interrupted with ,
- a contour element has been machined with different technologies (only the most recent technology is taken into account),
- too large feedrates have been selected for machining.



The calculating method used by MTS is, of course, a compromise, however, unavoidable since a more accurate calculation results in unacceptable computing times.

Figure 9.5-2: CNC Turning, Peak-to-Valley Height; The screen window is divided to display the peak-to-valley heights: you can have the entire workpart or sections of it displayed in the left half, the right half shows - normally in a highly enlarged sectional display - the specified peak-to-valley height graphically. Both image sections can be varied as desired.

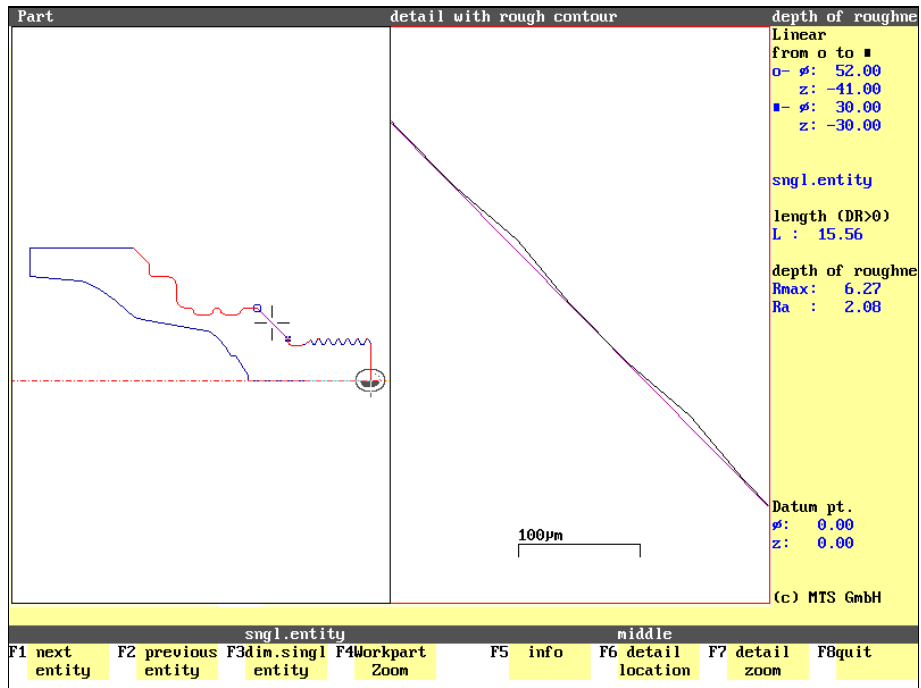
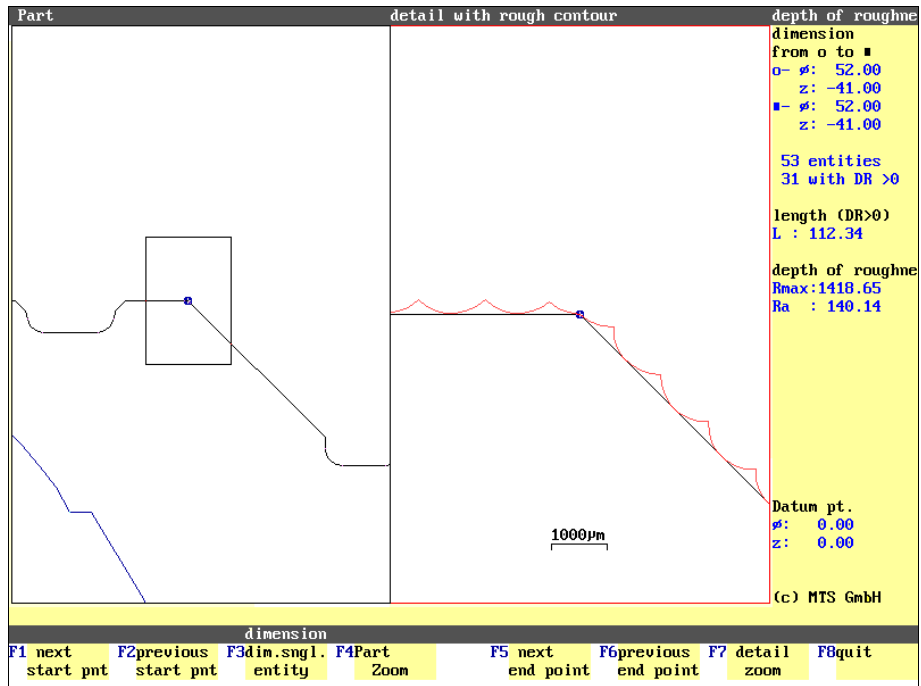


Figure 9.5-3: CNC Turning, Peak-to-Valley Height calculation of a contour section. In this case the preceding workpart machining was performed with four times the feedrate.



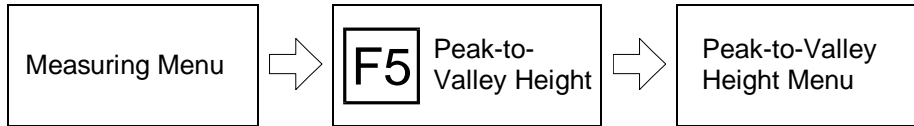


## Menu: Peak-to-Valley Height

The peak-to-valley height menu consists of two basic work steps:

- 1.) Selection of the contour elements for calculating the peak-to-valley height.
- 2.) Modifying the screen displays.

### Selection:



### Note

Proceeding from the Measuring main menu, press **F5** to reach the Peak-to-Valley Height menu:

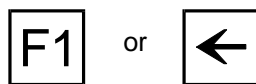
The calculation of the peak-to-valley heights may require some time. You can interrupt the calculation with **ESC**.

The Peak-to-Valley Height function can not be switched on until the blank has been machined.

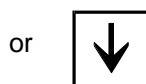
### Display

In the Peak-to-Valley Height menu the screen window is divided into two sectors: the workpart or a section of it is displayed as a rule on the left side and the currently selected contour element enlarged on the right with display of the specified peak-to-valley height (Peak-to-Valley Height Display). This section is adapted dynamically to the element selected in each case, while, however, the selected scale is retained. In addition, the section of the Peak-to-Valley Height Display is marked in the left window by a cross or a rectangle. Both sections can be enlarged or reduced as required.

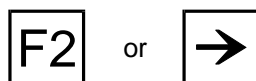
The remaining screen area is divided, as usual, into the **Information Column** and the **Function Key menu**:



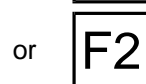
**Next element:** With **F1** you select, just like in Element Dimensioning, the next contour element whose theoretical peak-to-valley height is displayed on the right/left



side depending on the selected screen display. In addition, the geometrical values including the maximum peak-to-valley height:  $R_{max}$  and the mean roughness value:  $R_a$  are displayed in the text column.



**Previous element:** Correspondingly you select the preceding contour element with **F2**.



**Measuring range/individual element:** Instead of specifying the peak-to-valley height of a single contour element, you can also define the peak-to-valley height of a certain contour section ("measuring distance"). You switch to peak-to-valley height calculation of a measuring distance with **F3**.



**Zoom workpart:** The range of the picture in the **left window** can be redefined with **F4**.



**Info:** With **F5** you receive information on the entire workpart.



**Position detail:** With this function the peak-to-valley-height representation of the right window is positioned to the starting, end or center point of the selected element.

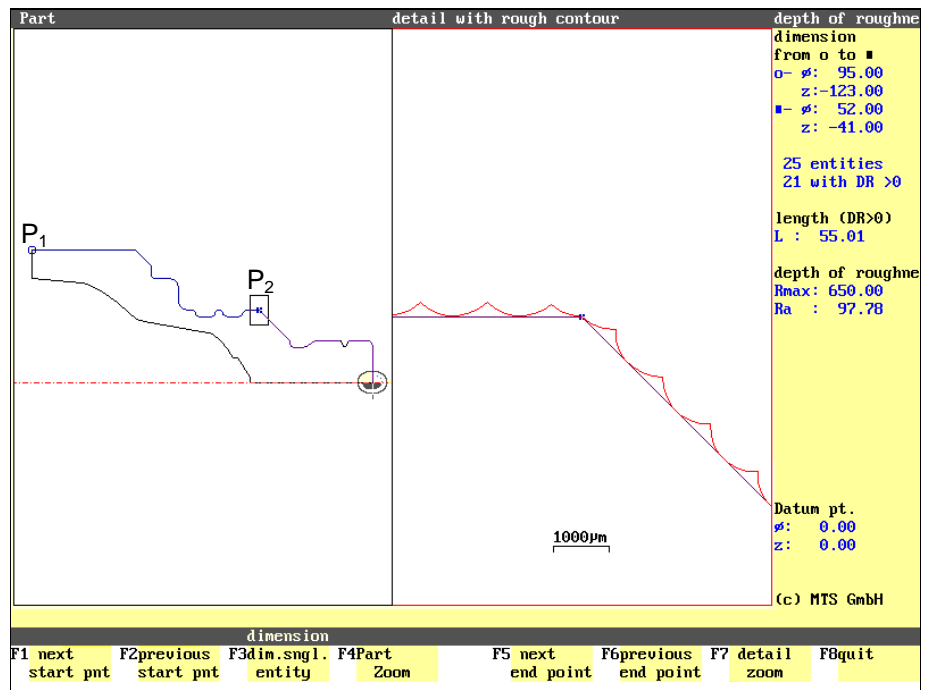


**Zoom detail:** With this function the section enlargement displayed in the right window is defined.



**Return:** With **F8** you return to the measuring main menu.

Figure 9.5-4:  
 CNC Turning,  
 Peak-to-Valley Height,  
 measuring line output;  
 With this special  
 function you can  
 output the peak-to-  
 valley heights of a  
 contour section  
 (here measuring  
 distance P<sub>1</sub> - P<sub>2</sub>).



## Peak-to-Valley Height Calculation of Measuring Distance

As an alternative to defining the peak-to-valley height of an individual contour element, you can have the peak-to-valley heights of a certain contour section ("measuring distance") calculated as well.

The elements within the measuring distance are highlighted in color and parallel to this the number of the elements, the geometrical values and the peak-to-valley heights are output in the information column. Finally, in addition to the above functions, the following operation options are available:

**F3**

**Measuring distance/single element:** Proceeding from the "Peak-to-Valley Height menu", you switch with **F3** to the **peak-to-valley height calculation of several contour elements** and then specify the measuring distance for which the peak-to-valley height is to be calculated.

To switch off the addition function, press again **F3**: the "single element" data are then displayed in the information line.

**F1** or **F2**  
**→** or **←**

**Next/previous starting point:** You set the starting point of the measuring distance on that of the next/previous element with **F1** or **F2** respectively. The peak-to-valley height is displayed correspondingly on the right side and the geometrical values of the measuring distance including the maximum peak-to-valley height:  $R_{max}$  and the mean roughness value:  $R_a$  are displayed in the text column.

**F5** or **F6**  
**↓** or **↑**

**Next/previous end point:** By analogy, you set the end point of the measuring distance on that of the next/previous element with **F5** or **F6**.

### Note

The measuring distance is the distance between the starting and end point; if these are identical, the measuring distance covers the entire workpart.

Figure 9.5.1-1: CNC Turning, Peak-to-Valley Height, zoom workpart (F4); as standard, the entire workpart is displayed in the left window. With the zoom function you change this display and can select enlarged or reduced presentations of it.

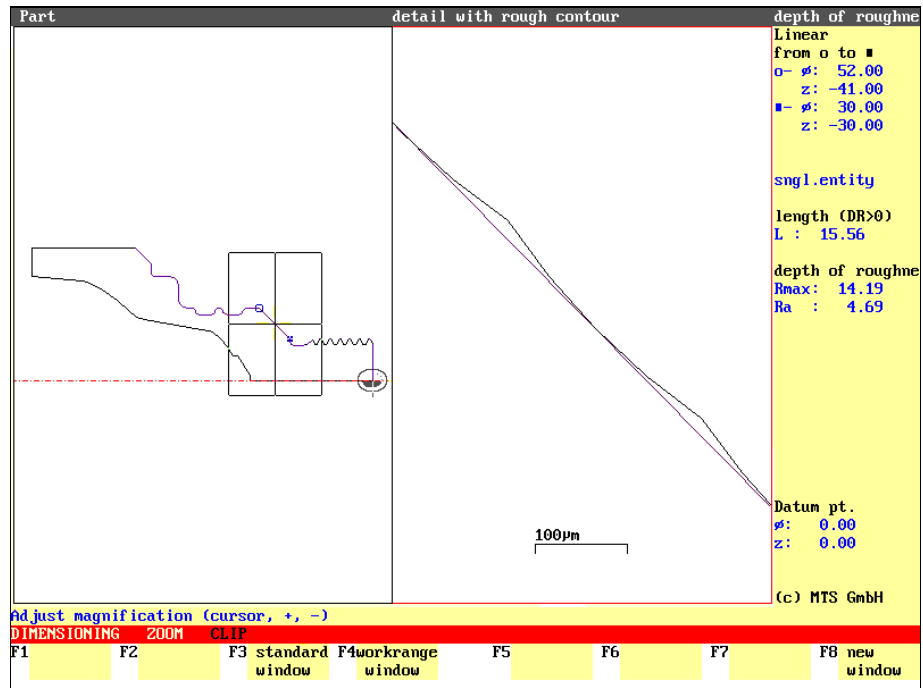
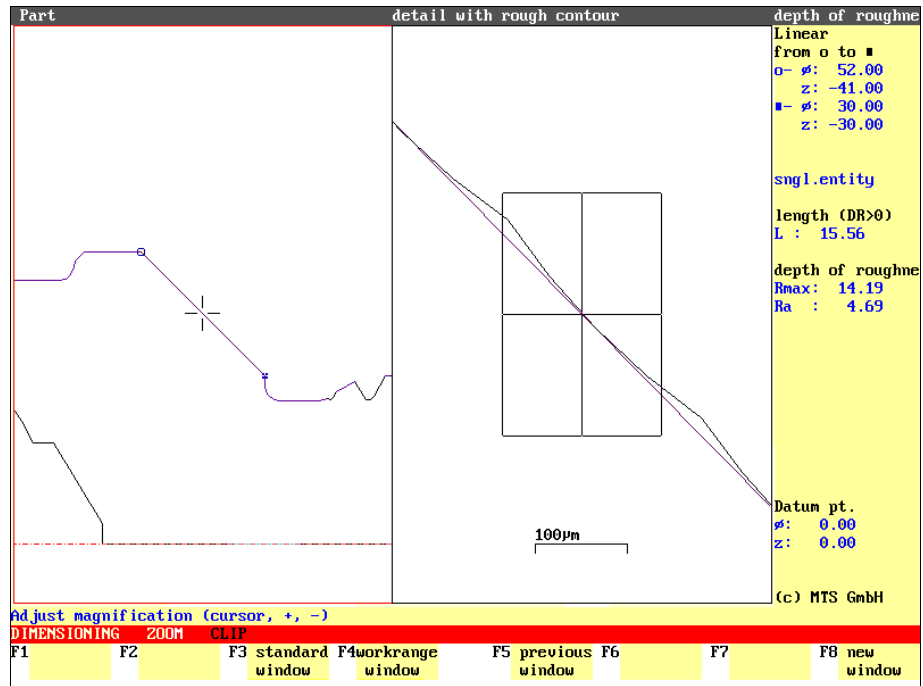


Figure 9.5.1-2: CNC Turning, Peak-to-Valley Height, zoomed detail (F7); the right window displays the specified peak-to-valley heights graphically; as in the workpart display, a section of the representation can also be selected for this. The Figure shows the choice of a new section using the "Zoom Detail" function F7.



## 9.5.1 Modifying the Screen Display

The peak-to-valley heights of a workpart are normally only a few  $\mu\text{m}$ . It is therefore necessary for a clear graphical representation to enlarge the section considerably for the display.

### Zoom

As already mentioned, the screen window in the Peak-to-Valley Height menu is divided into areas showing in the left half the finished workpart or sections of it (called "workpart display" in the following) and in the right half the specified peak-to-valley height for one or several contour elements (called "Peak-to-Valley Height Display" in the following). Both display areas can be separately adjusted with different zoom functions as desired:

### Changing the Workpart Display

The workpart display (left window) is adjusted analogous to the Zoom function in the other program sections (cf. [Chapter 10](#)):

F4

**Zoom workpart:** In the Peak-to-Valley Height menu, the Zoom function for the left window is called up with **F4**.

You then define or select a new section of the representation.

ESC

After definition/choice of the section, you return to peak-to-valley height calculation with **ESC**.

### Changing the Peak-to-Valley Height Display

There are two operating functions for modifying the Peak-to-Valley Height Display (right window).

#### 1.) Zoom Detail:

F7

**Zoom Detail:** With this function the enlargement/reduction of the representation area of the right window is specified.

You then define or select a new section of the representation.

ESC

After the definition/selection of the section, you return to peak-to-valley height calculation with **ESC**.

Figure 9.5.1-3: CNC Turning, Peak-to-Valley Height, Position Detail; With this special function you can adjust the Peak-to-Valley Height Display to the specific situation. In this example, the center of the detail window was set onto the starting point.

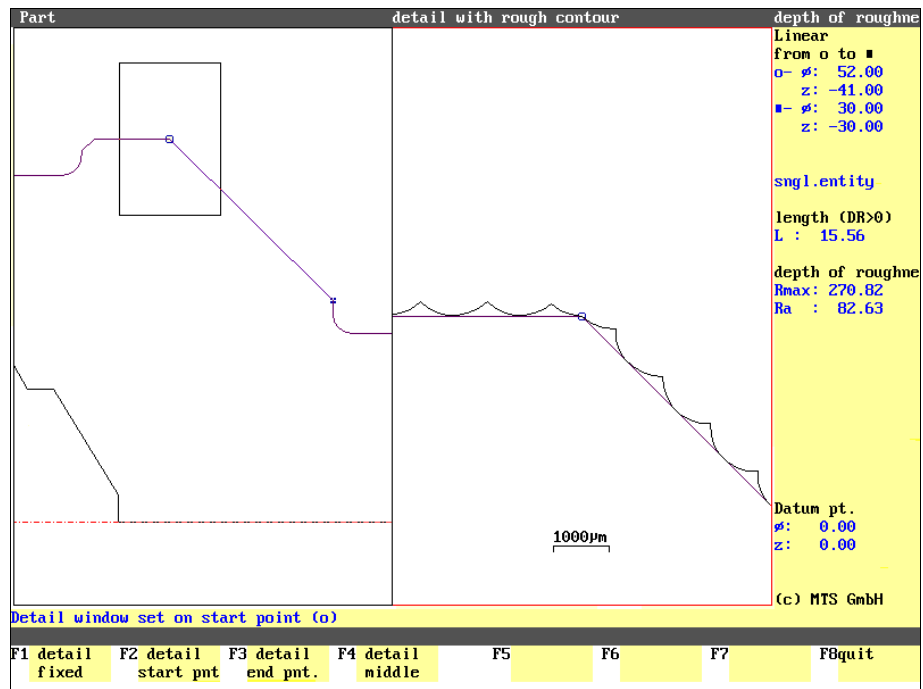
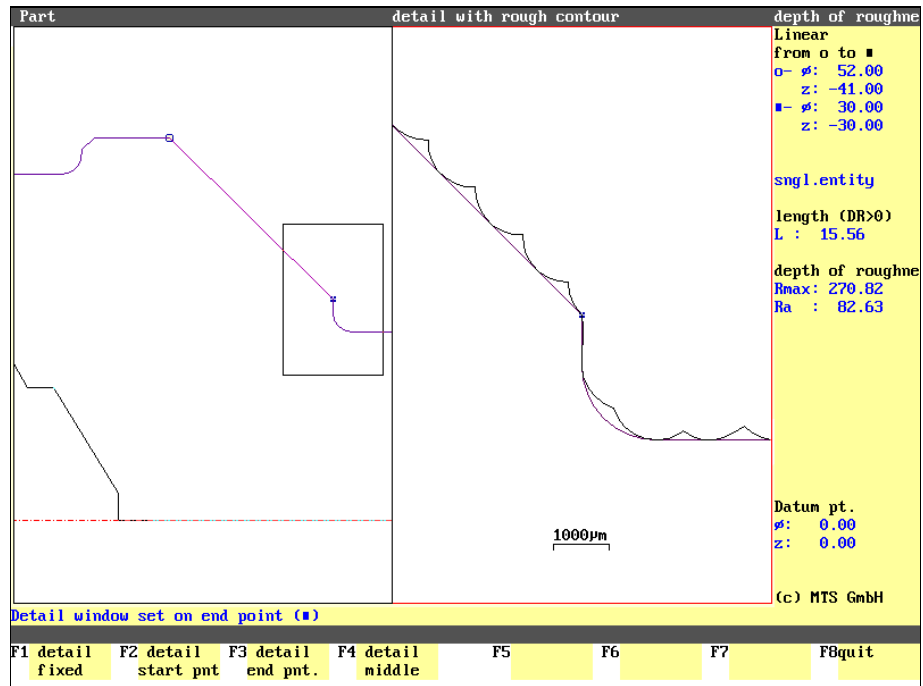


Figure 9.5.1-4: CNC Turning, Peak-to-Valley Height, Position Detail; With this special function you can adjust the Peak-to-Valley Height Display to the specific situation. In this example the center of the detail window was set onto the end point.



## 2.) Position Detail:

This function is available only in the "Single Element" mode.

**F6**

**Position detail:** The Peak-to-Valley Height Display can be re-adjusted with **F6**:

**F1**

**Fixed detail:** The selected area for Peak-to-Valley Height Display is confirmed as the current area with **F1**. Consequently, if you select further contour elements after this fixation, these section windows are not considered dynamically any more.

**F2**

**Starting point detail:** The center of the selected Peak-to-Valley Height Display is placed in each case on the starting point of the current contour element with **F2**.

**F3**

**End point detail:** The end point of the current contour element is selected as center of the selected Peak-to-Valley Height Display with **F3**.

**F4**

**Center detail:** By analogy, the center is placed on the center point of the current contour element with **F4**.

**F8**

**Return:** After you have determined the desired adjustment of the Peak-to-Valley Height Display, press **F8** to return to the Peak-to-Valley Height menu.

### Note

If the "Fixed detail" status was set, this is retained in the Measuring Distance mode. Otherwise the Peak-to-Valley Height Display in this mode is always centered on the last shifted starting or end point.



**Always select a clear representation for both display parts.**

## Color Configuration of the Peak-to-Valley Height Display

**C**

The peak-to-valley height calculation has a color configuration of its own. It is invoked with the **C** key.

The color of the marked elements should be the same as that of unmarked ones, however, of different color intensity (e.g. light-red - dark-red). The colors marked with a "\*" cannot be changed in the color configuration menu.

### Note

The color configuration is discussed in the following chapter.





## 10. Changing Graphic Display

Apart from the display modes already described, the Turning and Milling CNC Simulators offer various possibilities for modifying the screen display. Certain processes can, for instance, be highlighted.

### 3D Views

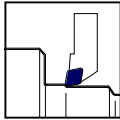
The option of three-dimensional views of the workpart displayed at any time from different viewing directions is a practical support in NC programming.

### Sectional Display

In CNC turning, the changes mainly concern different **views of the workpart**, whereas in CNC milling, apart from top view, **graphic-dynamic section displays** relating to the current working position of the tool in X/Z and Y/Z axial direction and the axes can also be displayed.

### Zoom

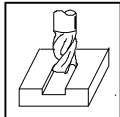
#### CNC Turning



Furthermore, the turning Simulator allows gradual enlargement or reduction of the screen section (up to the representation of the entire working space of the machine tool) even during processing.

This zoom function is also available during measuring and Workshop-Oriented Programming (WOP).

#### CNC Milling



Unlike in the Turning Simulator, **only the contours programmed with the WOP can be zoomed in milling**, and the display of the entire work space of the machine tool is possible only as a special function in the setup mode when defining clamping (cf. [Chapter 4](#)).

### Color Display

Finally, it is possible to define all **colors** of the screen elements (corresponding to the graphics card of your PC) as required.

Figure 10.-1:  
CNC Turning,  
Automatic Mode;  
As a first step the  
Measuring main menu  
is called up under the  
menu item  
"Measuring/3D View"

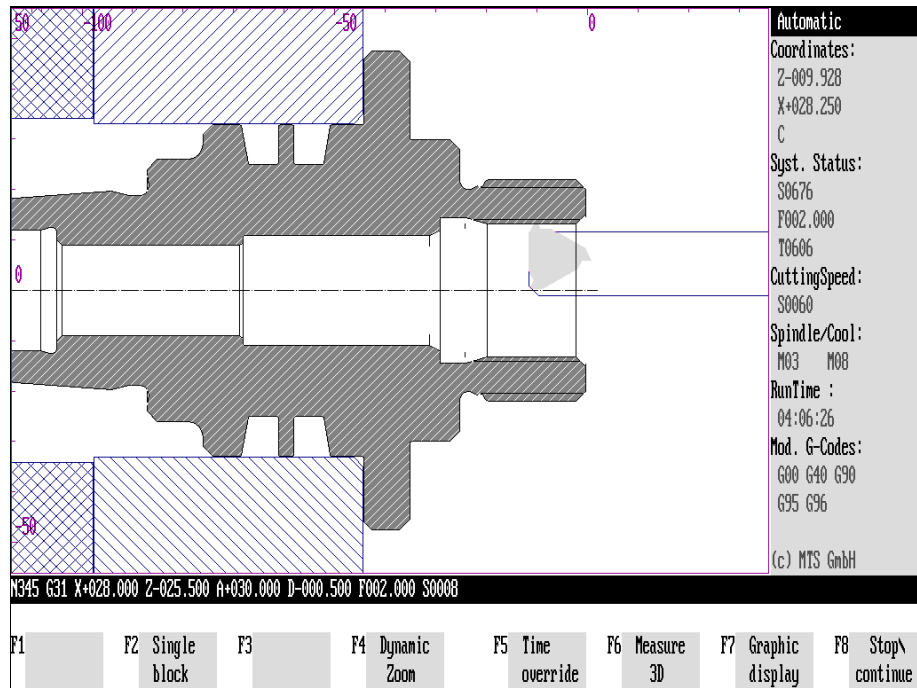
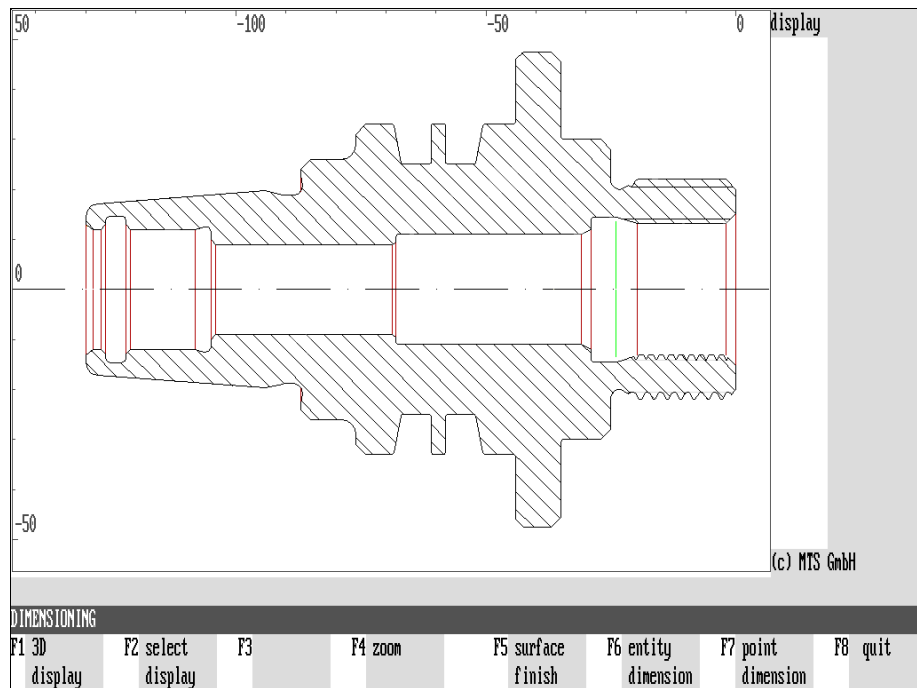


Figure 10.-2:  
CNC Turning,  
Measuring main menu;  
Press **F1** to switch to  
the 3D Display menu.

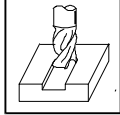


## 10.1 3D Display

With the version 5, 3D displays can be called up both in the turning and in the milling Simulators. To display machining inside the workpart, it is possible to cut out any cylinder sectors (CNC turning) or workpart quadrants (CNC milling) in the 3D display.

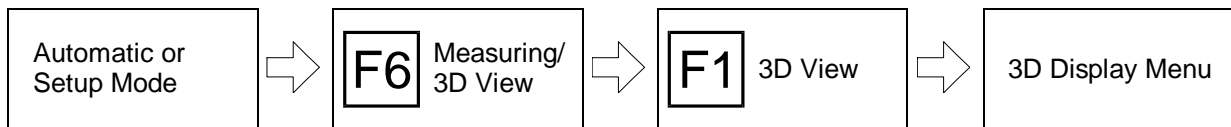
### Variable Sections

#### CNC Milling

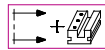
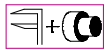
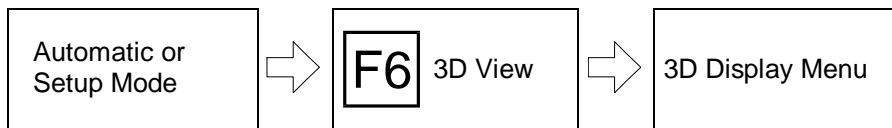


In addition to the 3D display, freely definable sectional displays can be selected in the milling Simulator. The displayed section areas are no longer tied to the milling cutter center point in this display mode but can be re-defined in X and Y parallel axes. In the sectional displays, a horizontal line is also displayed indicating the relevant Z plane. Naturally its position can also be changed.

#### CNC Turning Selection Example:



#### CNC Milling Selection Example:



You select the 3D display using the "Measuring/3D View" (turning Simulator) or "3D View" (milling Simulator) function. It is called up with **F6**

- in the **Setup Mode**,
- in the **Automatic Mode**,
- during **Interactive Programming** or
- during **Teach-In Programming**

If you are currently working with an other operating function of the CNC Simulator, first switch to the Automatic Mode or Setup menu.



In turning, the "3D Display" function is called up in the Measuring main menu with the function key **F1**. The 3D Display menu is then loaded.

#### Note

Since the 3D Display function is slightly different in turning and milling Simulator, it is discussed separately for each technology.

Figure 10.1.1-1:  
CNC Turning,  
Menu: 3D Display,  
Schematic Display.

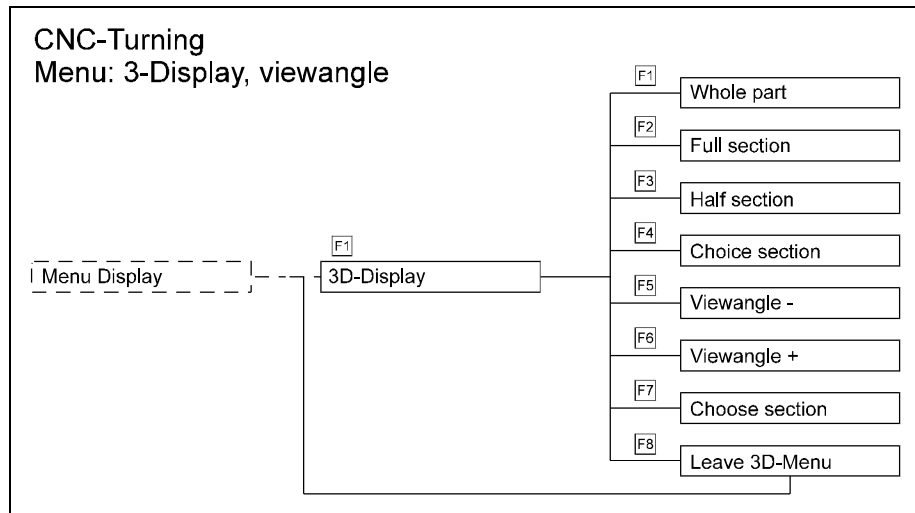


Figure 10.1.1-2:  
CNC Turning,  
3D Display,  
Select viewing angle;  
before the 3D view is  
loaded, the viewing  
angle of the workpart  
can be chosen.

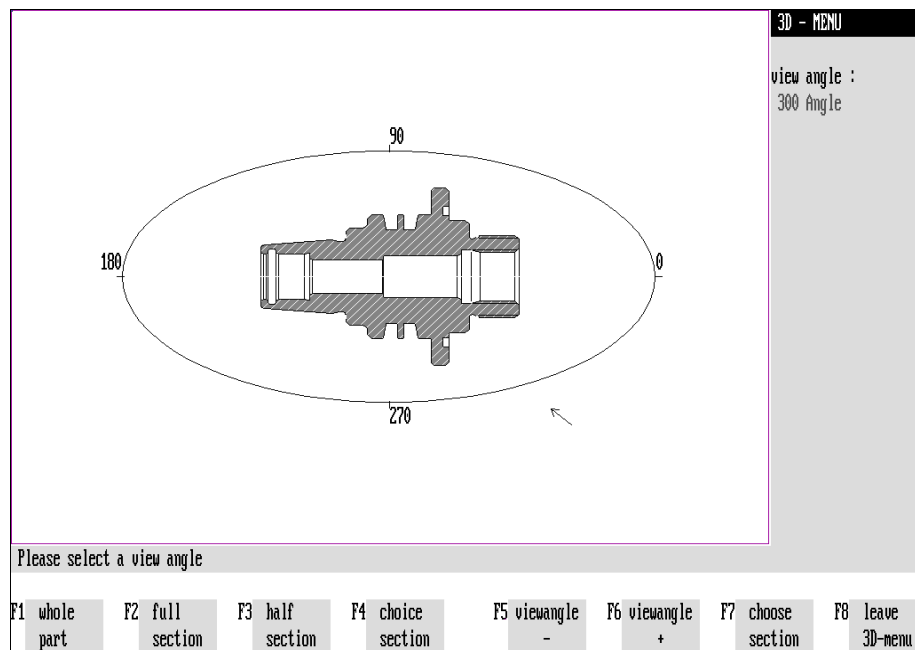
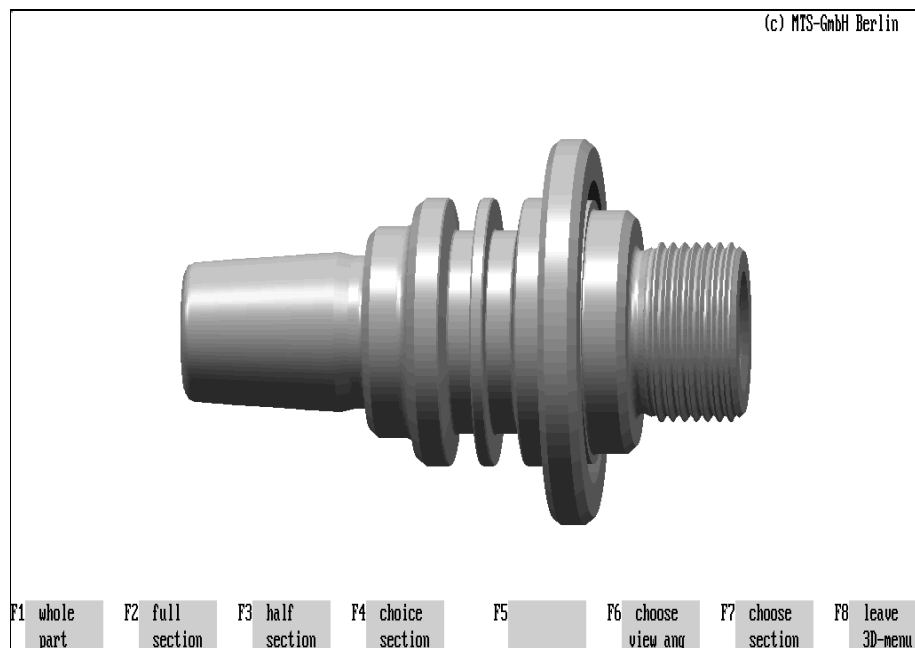
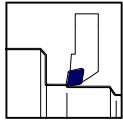


Figure 10.1.1-3:  
CNC Turning,  
3D Display, outside  
view corresponding to  
the above selected  
viewing angle.



## 10.1.1 CNC Turning: 3D Display



The CNC Turning Simulator allows three-dimensional display representation of the workparts from different viewing angles. Furthermore, the 3D representation can be displayed as a half or full section or with any cut-out sectors.

### Procedure:

Before the spatial representation is displayed, you first define the viewing angle onto the workpart and the display mode. The corresponding 3D view is then loaded and you can subsequently select a new viewing angle or display mode again, or you return to the starting situation to continue processing.

### Select Viewing Angle



The intermediate menu "Select Viewing Angle" shows in the center of the screen window a graduated ellipse enclosing the sketched workpart (quasi as "circulatory path" for the viewing point) with a small arrow next to it. This arrow indicates the currently selected viewing angle. The current viewing position is also displayed in the information column:

F5



or



**Viewing Angle -:** You change the view angle clockwise with **F5** or the cursor keys **↓** or **←**. The position arrow is shifted accordingly and the corresponding number of degrees is displayed in the information column.

F6

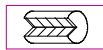
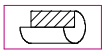
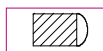


or



**Angle of View +:** You change the viewing angle counter-clockwise with **F6** or the cursor keys **↑** or **→**. The position arrow is shifted accordingly and the corresponding number of degrees is displayed in the information column. After you have determined the viewing angle, select the display mode:

### Select Display Mode



The following display modes are available: external view, half or full section or "any section" according to your section definitions.

F1

**Full workpart:** The workpart is displayed as viewed from outside with **F1**.

F2

**Full section below:** The workpart is shown in full section with **F2**.

F3

**Half section:** To display the workpart in half section, press **F3**.

F4

**Selected section:** The workpart is displayed according to your specification as a variable section with **F4**; if you have not made any section definition yet, the standard setting is displayed.



**Try the different 3D display options!**

Figure 10.1.1-4:  
CNC Turning,  
3D Display, Select  
section;  
Apart from the half and  
full section, any  
section can also be  
selected and  
displayed.

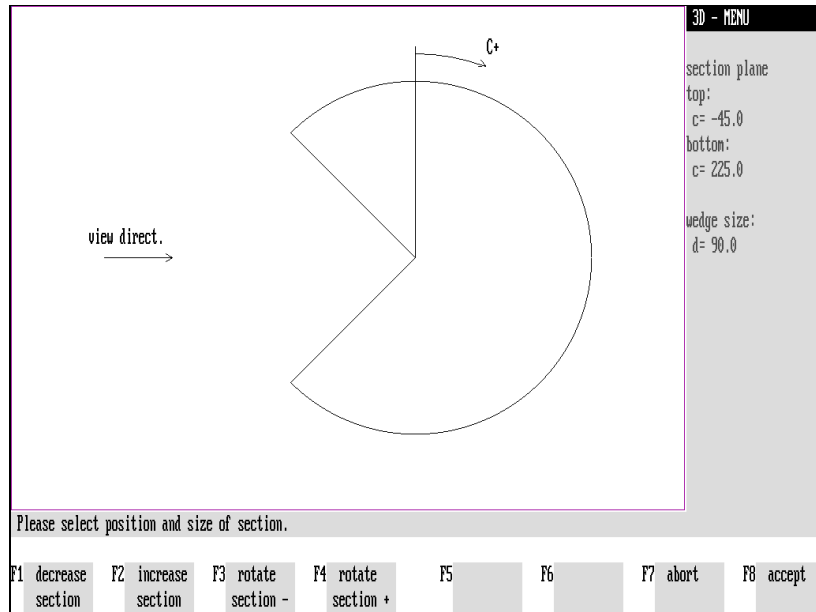


Figure 10.1.1-5:  
CNC Turning,  
3D Display, Select  
section;  
When defining the  
section, both the  
section size and the  
viewing angle onto the  
section can be  
changed.

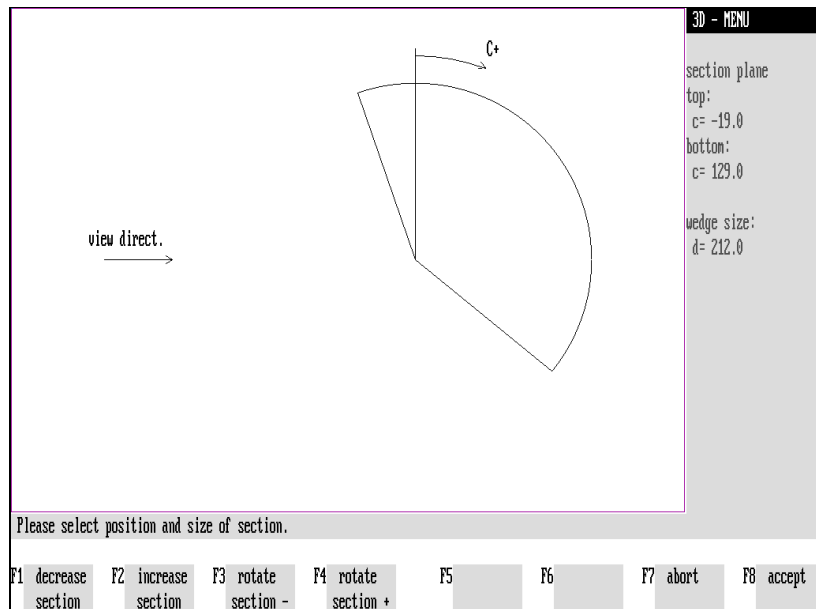
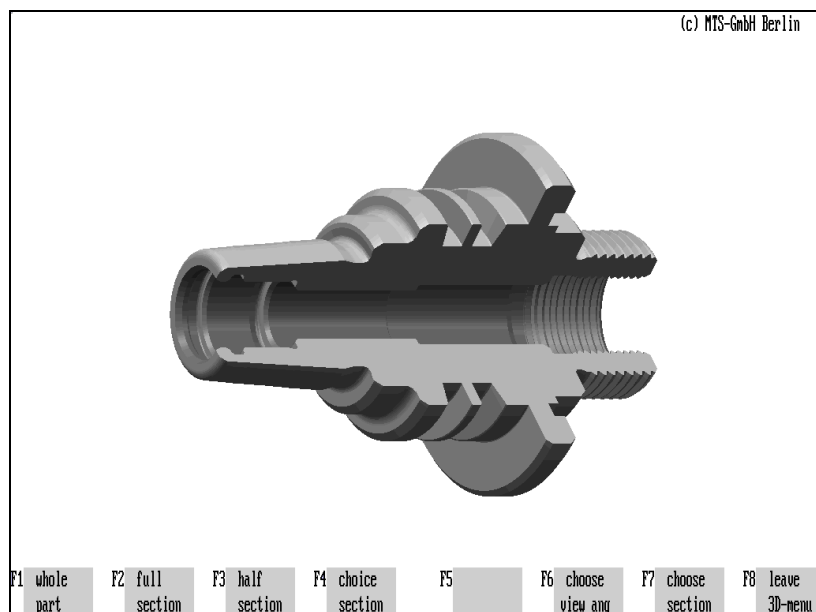
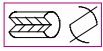


Figure 10.1.1-6:  
CNC Turning,  
3D Display, Optional  
section;  
Example of a sectional  
display.

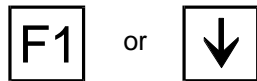


## Select Section

The standard setting of the optional section cut-out displays a 90° cut-out wedge viewed towards the center of the workpart. Both the viewing direction and the section cut-out can be redefined with the "Select Section" menu.

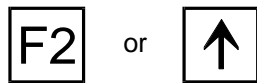


The "Select Section" menu shows in the screen window a circle with cut-out sector as a symbol for the cut-out workpart in the top view. You can enlarge or reduce the circular sector as necessary and change the viewing direction onto the section:



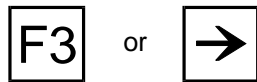
or

**Section smaller:** You reduce the cut-out circular sector with **F1** or the cursor key **↓**. Parallel to this, the angle of the circular sector is displayed under "**Wedge size**" in the information column.



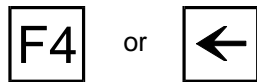
or

**Section larger:** You enlarge the cut-out circular sector with **F2** or the cursor key **↑**. Parallel to this, the angle of the circular sector is output under "**Wedge size**" in the information column.



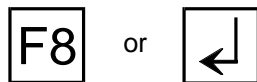
or

**Rotate section -:** You rotate the workpart clockwise with **F3** or the cursor key **→** and the viewing direction onto the section changes. At the same time, the current angle of rotation is displayed under "**Section planes**" in the information column.



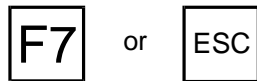
or

**Rotate section +:** You rotate the workpart counter-clockwise with **F4** or the cursor key **←** and the viewing direction onto the section changes. At the same time, the rotation angle is displayed under "**Section planes**" in the information column.



or

You confirm the selected setting with **F8** or **↵**.



or

You interrupt section definition with **F7** or **ESC** without taking over the selected setting.

Figure 10.1.1-7:  
CNC Turning,  
Menu: 3D Display,  
Schematic of the  
menu sequence.

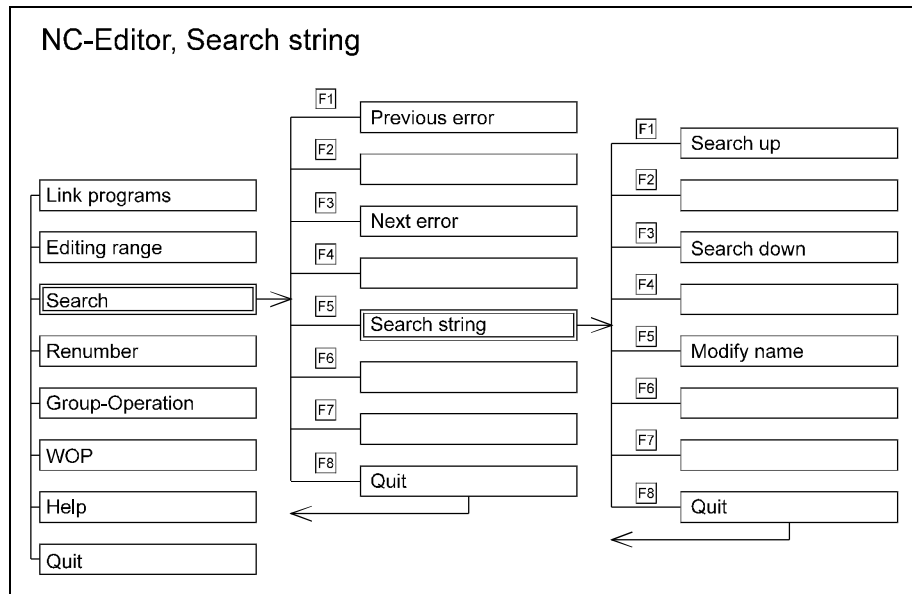
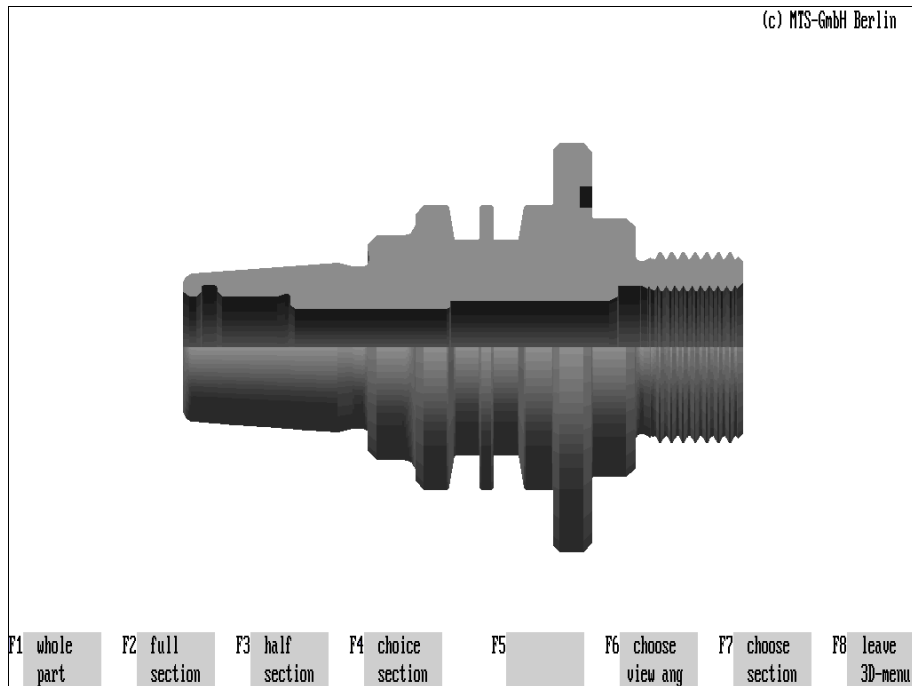


Figure 10.1.1-8:  
CNC Turning,  
3D display in half  
section.





## Menu: 3D Display

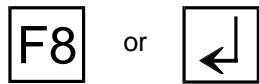
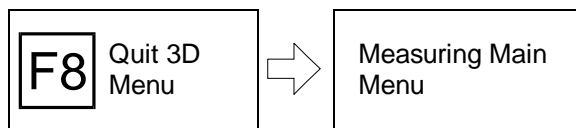
The quality of the 3D display depends on the graphic card of your PC. A reality-corresponding display representation can be received with the VGA standard (and higher) using different gray shade. According to the standard setting, 16 gray shades are reproduced with VGA video cards whereas, in most cases, these would be able to display as many as 256 colors (cf. hardware configuration).

The processing scope of the 3D Display menu is identical with that discussed earlier. Therefore, consult for reference the previous section, if necessary,



**You can load different 3 dimensional views of the workpart one after the other.**

### Return:



Press **F8** to return to the Measuring main menu.

### Note

The 3D view of complex turned parts including displays of threads requires a large working memory space. Should there not be enough memory space available, a corresponding error message is displayed.

Figure 10.1.2-1:  
CNC Milling,  
Menu: 3D Display,  
Schematic display.

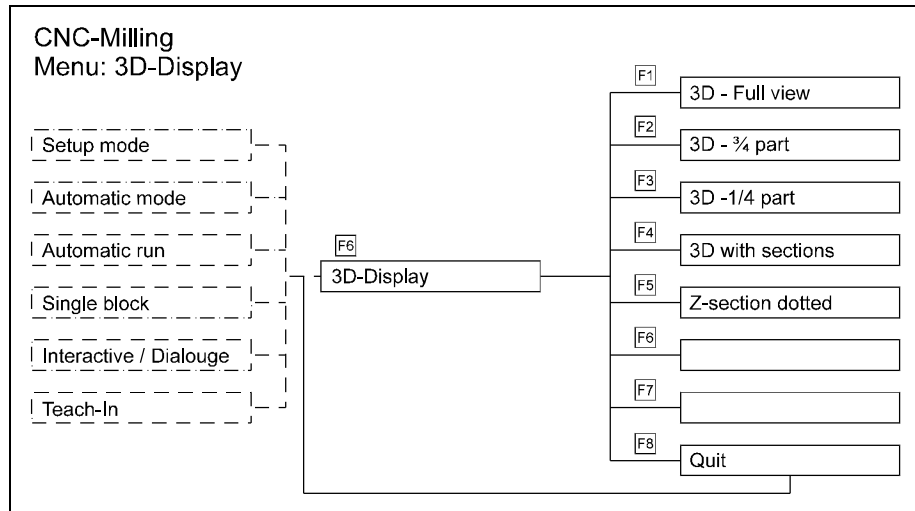


Figure 10.1.2-2:  
CNC Milling,  
3D Display,  
Select display;  
Before the 3D display is computed, the display mode and the viewing direction onto the workpart needs to be defined. The workpart quadrants are designated with A - D in the Figure.

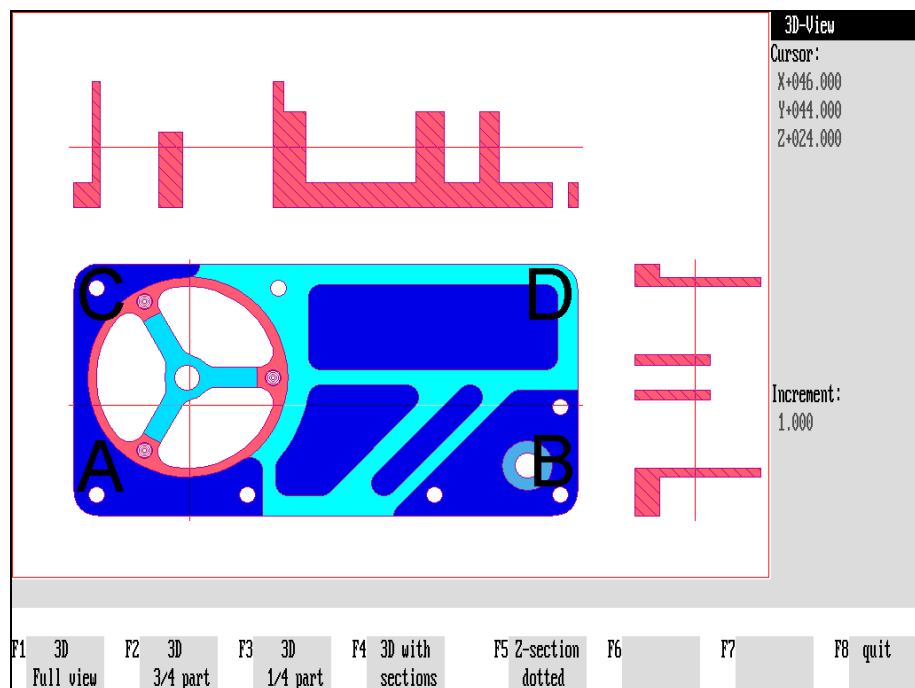
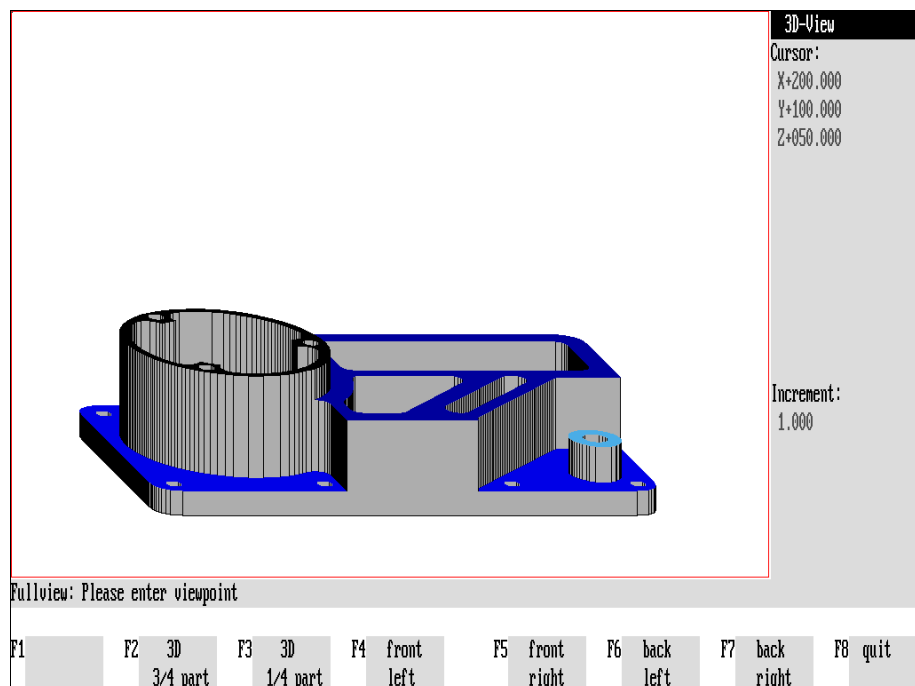
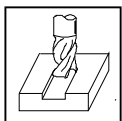


Figure 10.1.2-3:  
CNC Milling,  
3D Display, full view of the workpart viewed from bottom left of the screen ("front left").



## 10.1.2 CNC Milling: Sectional and 3D Display



To reproduce a top view and sectional views of a workpart requires spatial imagination from the viewer and from the drafter. As assistance, it is possible to have a **3 dimensional representation** of the workpart displayed for any machining situation.

Furthermore, it is possible to display **sectional representations** of any sectional axes as special functions.

As in turning, a 3D display representation of the workpart can be generated from any viewing angle. To check machining inside the workpart, the workparts are "cut open" and only specified "part pieces" are displayed.

### Procedure:

To display a 3D view, two or three settings must be selected first:

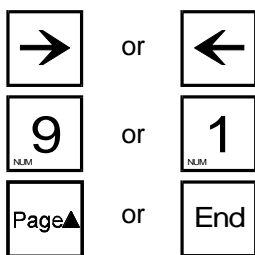
- 1.) For the sectional displays and the 3D views of a "cut open" workpart, first position the **sectional axes**. This operation is not necessary for displaying the whole part.
- 2.) Also choose the **display mode** and then select
- 3.) the **viewing angle** the workpart should be viewed from.

The corresponding 3D view is then built up and you can proceed to select new sections, viewing directions or display modes again, or you return to the starting situation to continue your processing.

### Select Sectional Position

After you have called up the 3D View function, an intermediate menu for positioning the sectional axes and selecting the display mode of the 3D view is loaded.

For a side view and a partial 3D side view, position the sectional axes in the X and Y direction (according to the plane selection "G17"). These axes divide the workpart into max. four parts (called "quadrants"). The corresponding sectional representations are "carried along" and the parallel coordinate values are displayed in the information column.



You shift the parallel line to the Y axis with the cursor key or .

You shift the parallel line to the X axis with the or or or keys on the **numerical keypad**.

The unit of movement of the sectional axes is determined by the **increment**. 1.0 mm, 0.1 mm, 0.01 mm or 0.001 mm are possible as setting for this and you determine these in the Setup Mode (cf. [Chapter 4](#)). The currently selected increment is shown in the information column.

For clarity, the workpart quadrants resulting from the division by the sectional axes are marked below with the letters "A" to "D" (cf. Figure opposite page).



**The workpart can be measured in simplified form with the aid of the sectional axes.**

Figure 10.1.2-4:  
CNC Milling, 3D  
Display,  
select viewing angle;  
you can choose  
between four different  
viewing directions of  
the 3D display in  
milling:  
(front left: 4, front right:  
5, rear left: 6, rear  
right: 7).

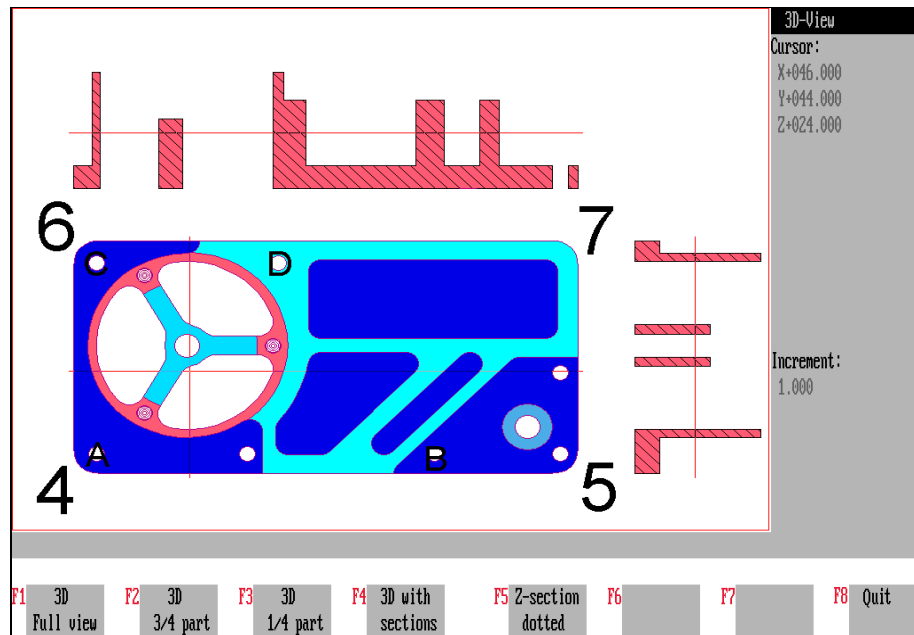


Figure 10.1.2-5:  
CNC Milling,  
3D Display, 3/4 part;  
View from the bottom  
right of the screen  
("front right").

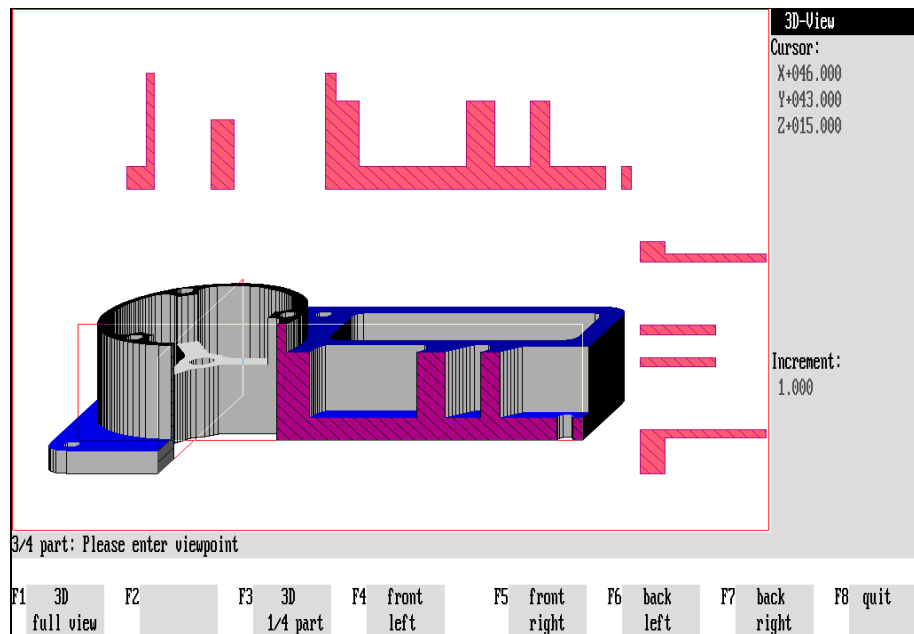
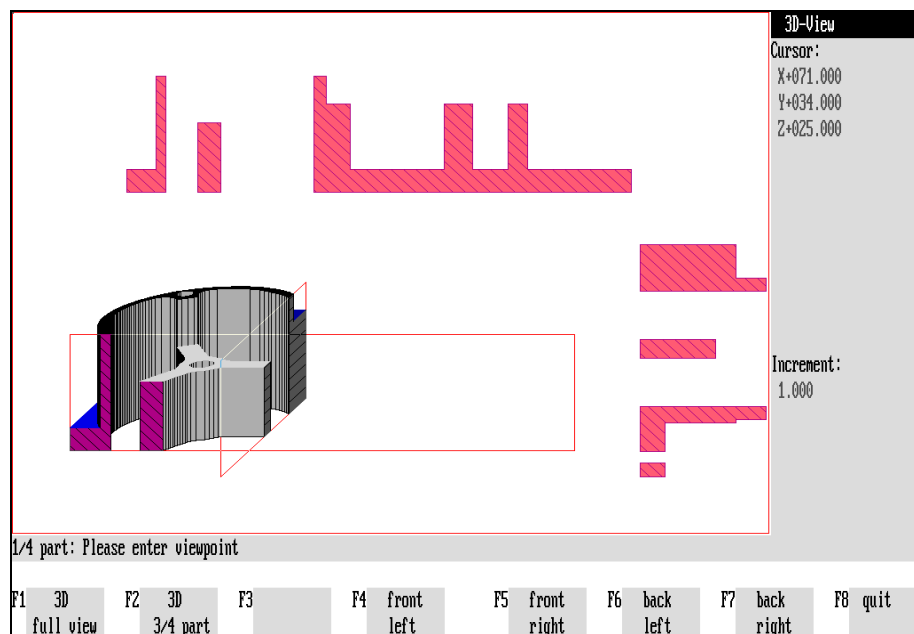


Figure 10.1.2-6:  
CNC Milling,  
3D Display, 1/4 part;  
View from the front  
right of the screen  
("front right").



## Select Display Mode

The display mode is selected using the function keys:

F1

**3D full part:** To display the complete workpart, press **F1**.

F2

**3D 3/4 part:** The front quadrant seen from the viewing direction is cut out with **F2**.

F3

**3D 1/4 part:** The three front quadrants seen from the viewing direction are cut out with **F3**.

F4

**3D with sections:** The full part and the side views, corresponding to the sectional axes, are displayed together with **F4**. In this case, the sectional areas are marked by the two rectangular frames. of the axes

F5

**Z section dotted:** The surfaces to be machined, locating above the set Z axis (according to G17), are displayed dotted in the top view with **F5**.

F8

**Return:** Press **F8** to return to the starting situation.

## Select Viewing Direction

In a 3-dimensional display, the viewing direction of the workpart is of importance. For this reason, the choice between four different viewing directions was integrated in the "3D Views" function. The perspective viewing "locations" are selected according to the workpart corners.

F4

The view onto the bottom left corner of the workpart (according to the top view) is selected as viewing direction with **F4**.

F5

The view onto the bottom right corner of the workpart (according to the top view) is selected as viewing direction with **F5**.

F6

The view onto the top left corner of the workpart (according to the top view) is selected as viewing direction with **F6**.

F7

The view onto the top right corner of the workpart (according to the top view) is selected as viewing direction with **F7**.

F8

**Return:** Press **F8** to return to the starting situation.

The following combinations are possible in the 3D View function (cf. Fig.):

Specification	1/4 part	3/4 part	Full part
to the front left	only quadrant D	A cut out	total
to the front right	only quadrant C	B cut out	total
to the rear left	only quadrant B	C cut out	total
to the rear right	only quadrant A	D cut out	total



**Now, get some training in the selection of the different 3D displays!**

Figure 10.1.2-7:  
CNC Milling,  
Menu: 3D Display,  
Schematic of the  
menu sequence.

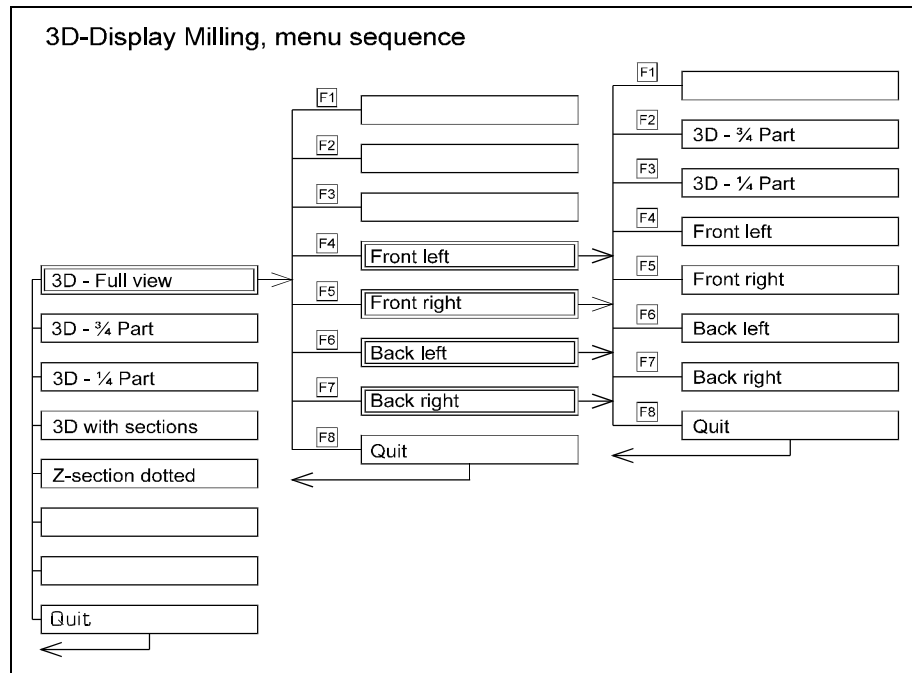


Figure 10.1.2-8:  
CNC Milling,  
3D Display  
Full part with sections

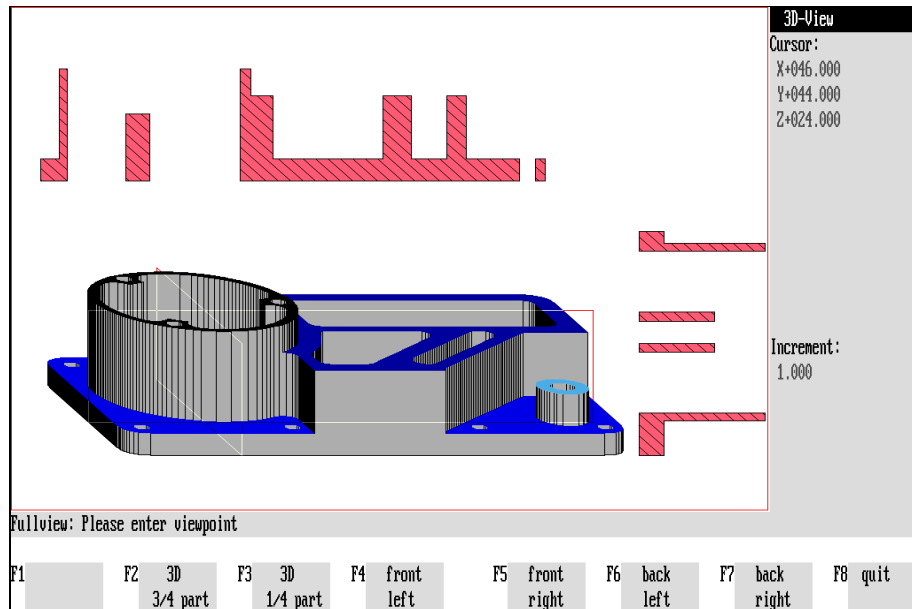
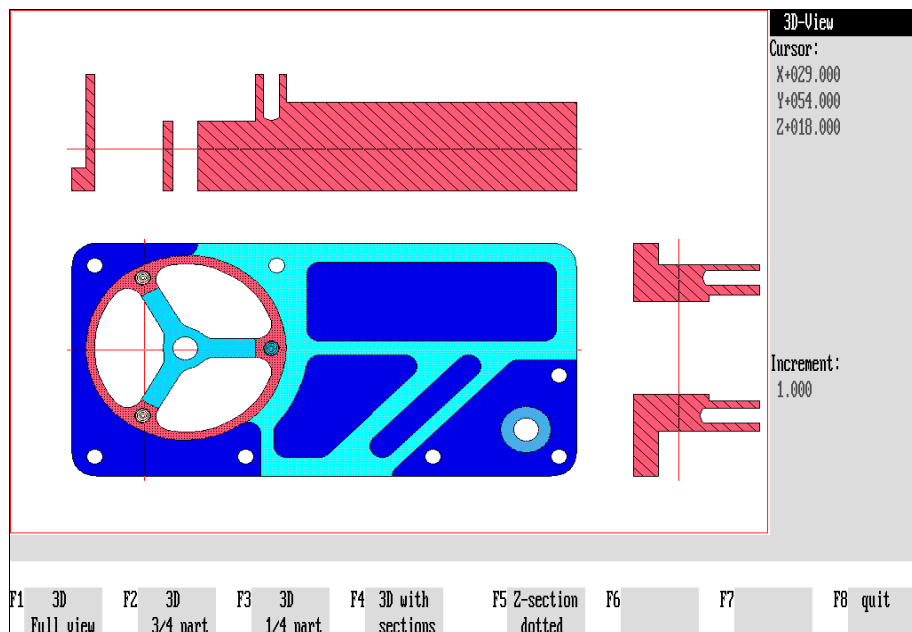


Figure 10.1.2-9:  
CNC Milling,  
variable sectional  
display,  
Display of the workpart  
depths:  
the dotted areas  
indicate the machining  
depths from the  
sectional axis to the  
workpart bottom edge.



## Menu: 3D View

After you have chosen the viewing direction, the 3D view is displayed. You can then call up further 3D views without closing the program function. Since the operation options in the 3D view menu are identical with those of turning, they are not discussed here.

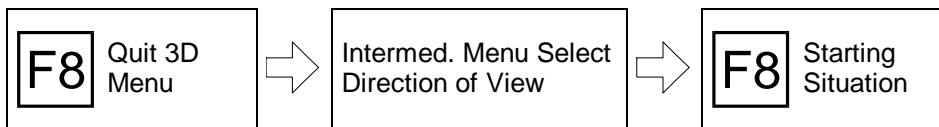
### Note

The computing performance of your personal computer as well as the complexity of the workpart geometry define how long it takes to generate a display representation.





**Different 3-dimensional views of the workpart can be computed one after another.**

### Return:



or



Press  or  to return to the starting situation.

## Calculating Machining Depths: Z Section Dotted

In addition to the sectional display representations and 3D views, the machining depths can be graphically highlighted and the corresponding Z coordinate values (according to G17) can be displayed.



### Procedure

The presettings for the calculation of the machining depths are made in the intermediate menu "Select Display":




or



The line parallel to the Z axis is positioned at the required machining depth with the cursor keys  or .

The movement unit of the sectional axes is determined by the **increment**. Settings of 1.0 mm, 0.1 mm, 0.01 mm or 0.001 mm are possible for this and you determine them in the setup mode. The currently selected increment is shown in the information column.



**Z section dotted:** To display the depth layers according to the selected sectional position, select the function .

All "layers" from the workpart top edge down to the selected sectional depth are then marked by a dotted grid.

Figure 10.2.1-1:  
CNC Turning,  
Graphical Display;  
Schematic of the menu  
sequence.

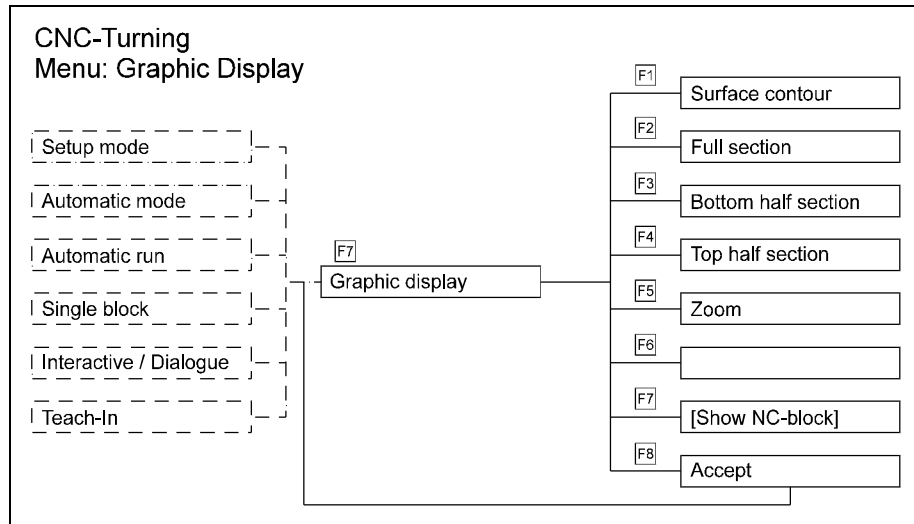


Figure 10.2.1-2:  
CNC Turning,  
Graphical Display;  
Display of the workpart  
"half section below"  
was selected.

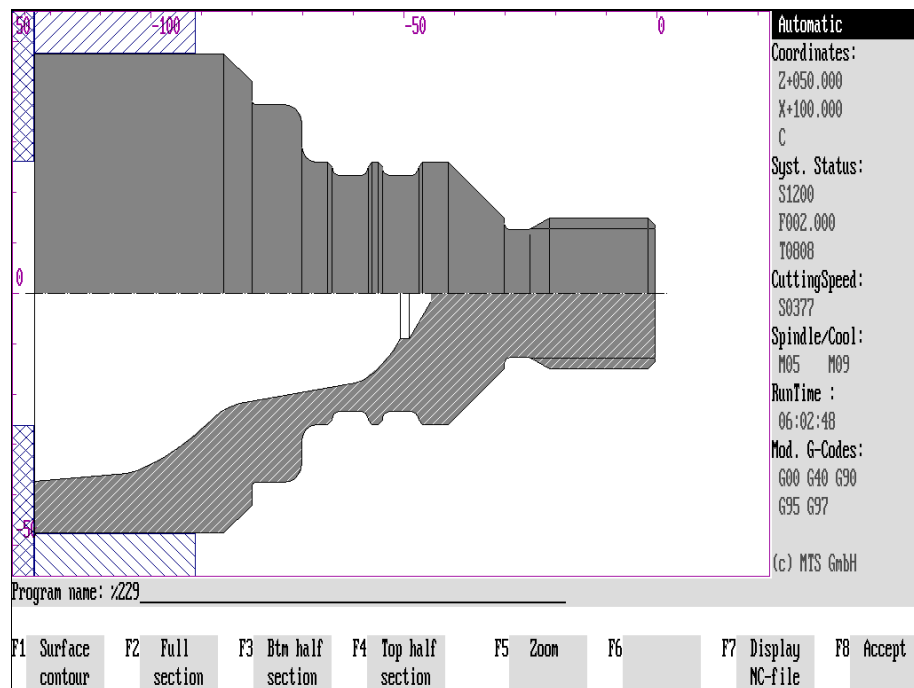
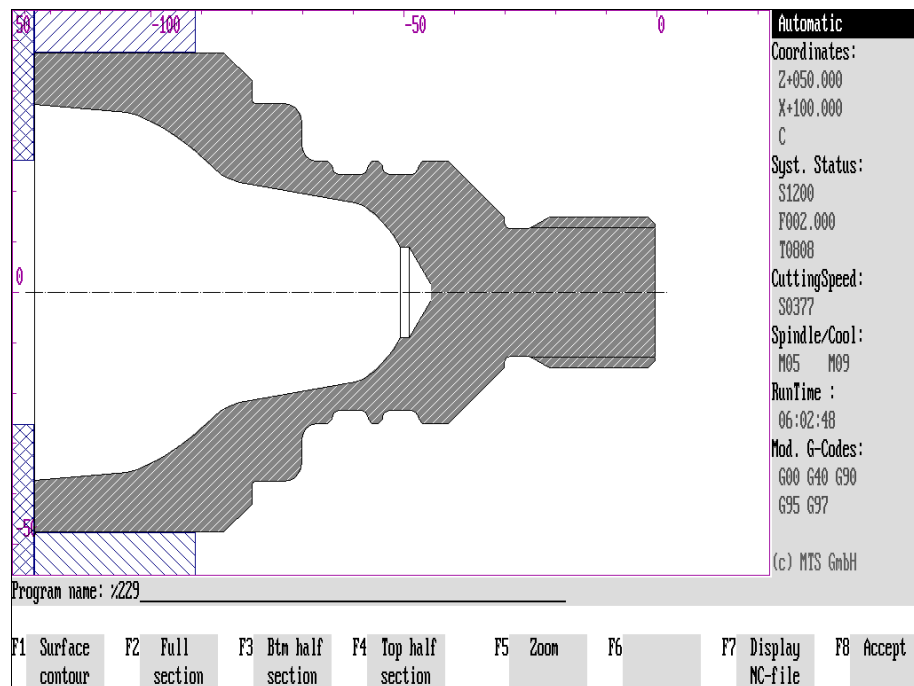


Figure 10.2.1-3:  
CNC Turning,  
Menu: Graphical  
Display;  
The display of the  
workpart in the "full  
section" was selected.



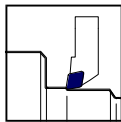


## 10.2 Changing the Graphical Display

Apart from the display modes already described, the turning and milling CNC Simulators allow different possibilities of modifying the screen display. In this way specific machining procedures can be clearly emphasized:

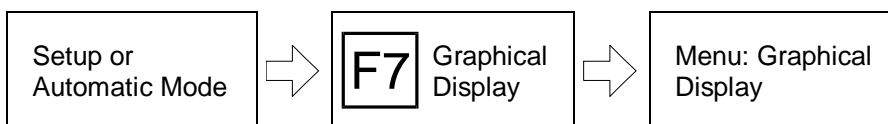
- In CNC Turning, the changes mainly concern different **views of the workpart**, whereas in CNC Milling apart from the top view, **graphic-dynamic sectional displays** related to the working position of the tool in X/Z and Y/Z axial direction and the axes can also be displayed.
- In addition, the display of turning feed operation can be enlarged gradually up to a section of 6  $\mu\text{m}$  x 8  $\mu\text{m}$ .
- Finally you can define all **colors** of the screen elements (depending on the video card of your PC).

### 10.2.1 CNC Turning: Changing the Display



The screen display is set in the "**Graphical Display**" menu which can be called up both from the setup mode and from the automatic mode:

#### Selection Example:



If you are currently using some other working function of the CNC Simulator, first switch to the Automatic or Setup mode menu.

GRAPHIC

You further switch to the "Graphical Display" menu with **F7**.  
The following options are available for changing the screen display:

**F1**

**External view:** The external view of the workpart is displayed by pressing **F1**.

**F2**

**Full section:** The full section of the workpart is displayed by pressing **F2**.

**F3**

**Half section below :** With **F3** the workpart is displayed as a section below the rotation axis and as an external view above it.

**F4**

**Half section above:** With **F4** the workpart is displayed as a section above the rotation axis and as an external view below it.

**F5**

**Zoom:** If you want to enlarge or reduce the displayed area, switch to the "Zoom" function with **F5**.

**F7**

**[Display NC line]:** The current NC block is displayed in the dialogue line during automatic mode. If you want to display several NC blocks at the same time, redefine the number with **F7**.

**F8**

**Accept:** You confirm your modifications with **F8** and return to the starting situation.

#### Note

Sectional display representations are hatched (cf. Figure) to distinguish them from external views.



The display mode changes automatically according to whether external or internal machining is being performed with the tool.

Figure 10.2.2-1:  
CNC Milling,  
Menu: Graphical  
Display;  
Schematic of the  
menu sequence.

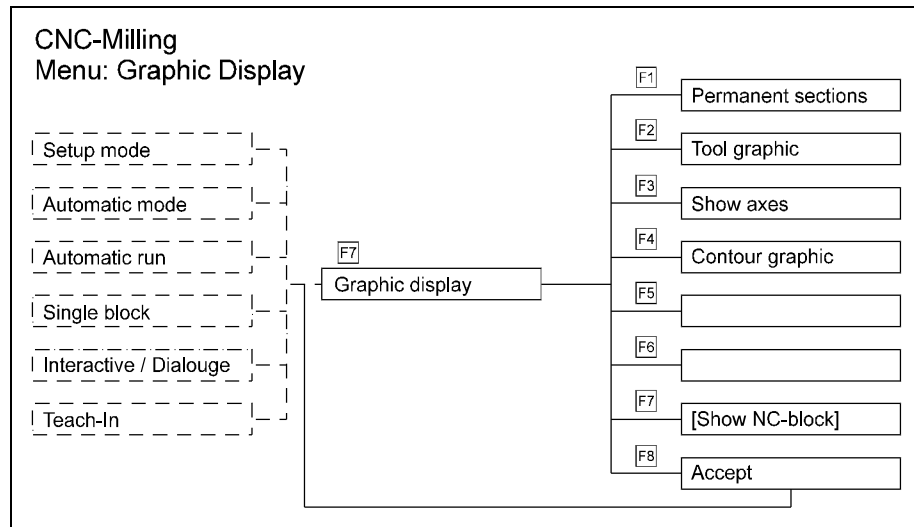
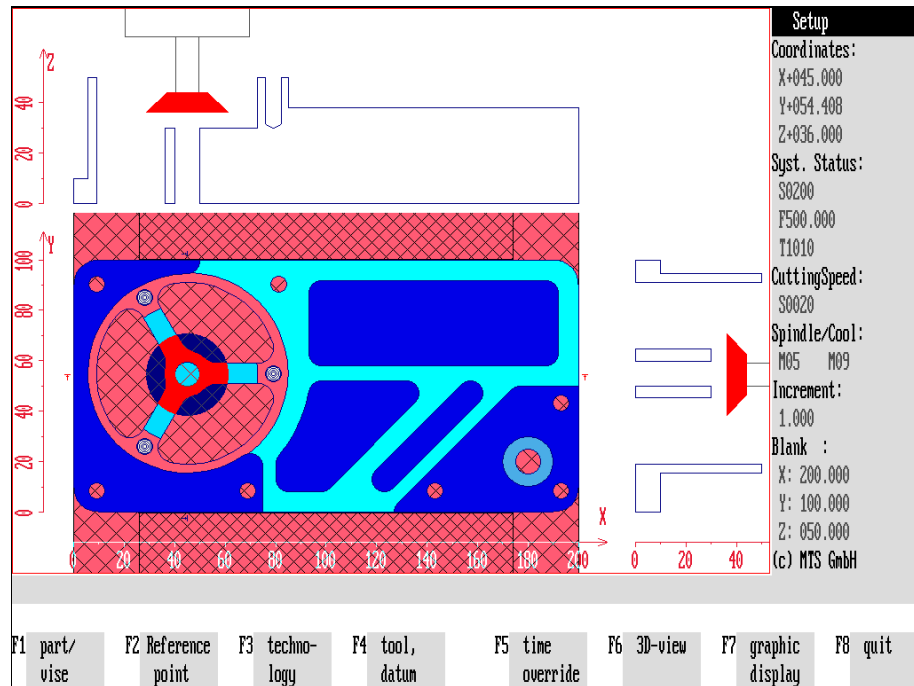
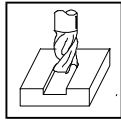


Figure 10.2.2-2  
CNC Milling,  
Setup Mode;  
Screen display: in  
addition to the top  
view, dynamic  
sectional displays of  
the workpart and of the  
tool as well as a  
system of coordinates  
can be displayed  
additionally.



## 10.2.2 CNC Milling: Changing the Display



As described, the milling CNC Simulator shows the workpart, the clamping and the tool in top view. Furthermore, these representations can be modified in the display:

- Displaying **graphic-dynamic sectional views** of the workpart,
- **Inserting sectional views** of the tool into the workpart sections,
- Displaying the **axes** of the machine tool.

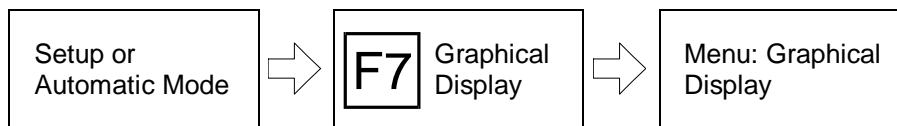
It is also possible, to display **sectional views** of any viewing location (cf. above).

### Note

The dynamic representation of sectional views should be switched off for PCs with low computing power.

The screen display is set in the "**Graphic Display**" menu, which can be called up both from the Setup Mode and from the Automatic Mode:

### Selection Example:



If you are currently applying some other working function of the CNC Simulator, first switch to the Automatic or Setup mode menu.

GRAPHIC

The menu for changing the graphical display is invoked with **F7**. The following operation options are available for changing the screen display:

**F1**

**Permanent sections:** The dynamic representation of sectional views of the workpart are displayed with this function.

**F2**

**Tool display:** If sectional representations of the tool are to be displayed, press **F2**.

**F3**

**Axes display:** The axes of the machine tool are displayed with **F3**.

**F4**

**Contour display:** The contours of the workpart are either highlighted in color or switched off with **F4**.

**F7**

**[Display NC line]:** The current NC block is displayed in the dialogue line as standard, parallel to processing. Should more than one NC block be displayed, you can redefine the number of blocks to be displayed with **F7**.

**F8**

**Confirm:** Press **F8** to confirm your modifications and return to the starting situation.

**ESC**

With **ESC** you interrupt this procedure without confirming and taking up your modifications.

Figure 10.2.2-3:  
CNC Milling,  
Setup Mode;  
Sectional displays.

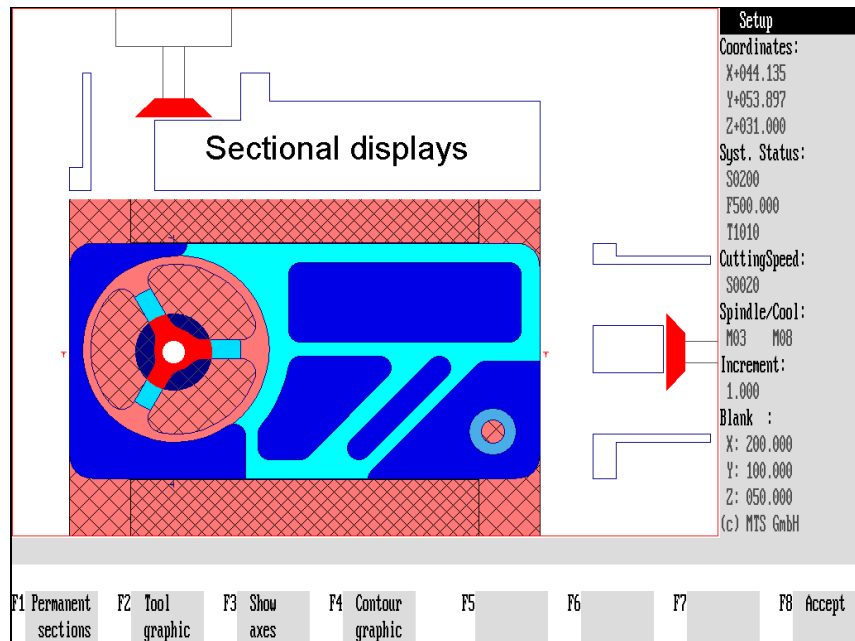


Figure 10.2.2-4:  
CNC Milling  
Setup Mode;  
Displaying the  
machine table axes

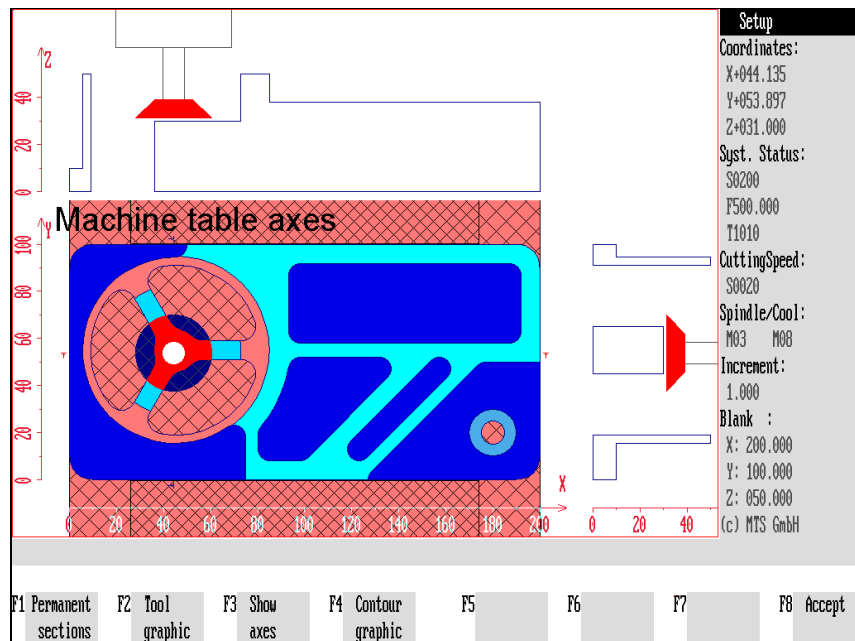
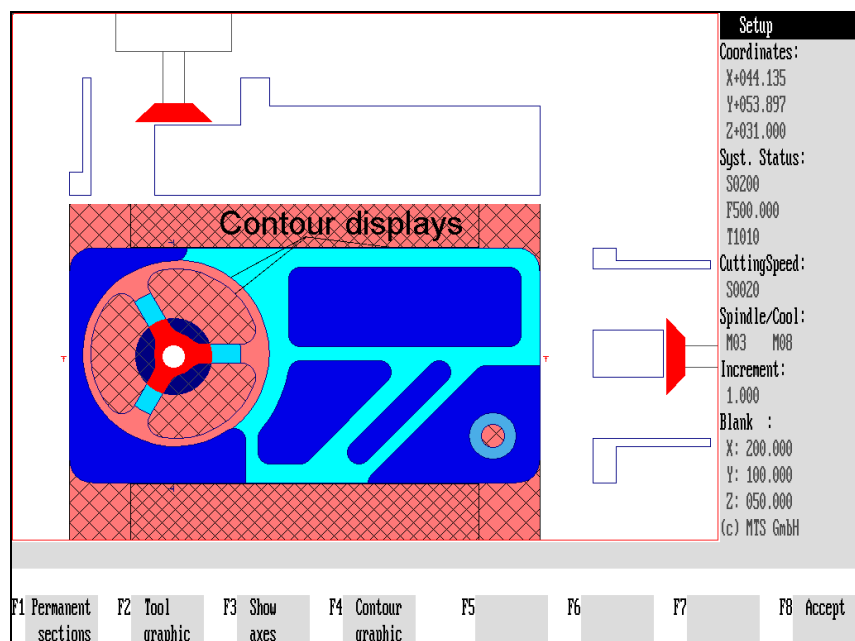
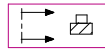
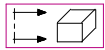


Figure 10.2.2-5:  
CNC Milling,  
Setup Mode,  
Contour display; if this  
function is activated,  
the workpart contours  
are highlighted in  
color.



## Graphic-Dynamic Sectional Displays



To obtain a 3-dimensional impression of workpart machining, **sectional displays** of the workpart and of the tool can be displayed and carried on with the machining process graphic-dynamically. The sectional axes run paraxially in the longitudinal and transverse direction ("front/side view") through the current tool position (cf. Figure).



The toggle function to display or switch off dynamic sectional displays (front and side view) is activated with the **F1** key. The following dialogue line appears:  
**"Sections carried along [Y/N] ?"** (Y = Yes, N = No)



The toggle function to display or switch off the tool in the dynamic sectional displays is activated with the **F2** key. There is then a program dialogue which offers you the following options for changing the display:  
**"Tool in front view [Y/N] ?"** (Longitudinal section)  
**"Tool in side view [Y/N] ?"** (Transverse section)  
**"Color cutting edge/shank/spindle [Y/N] ?"**



or



If you confirm a dialogue query with **Y** or **←**, the corresponding display mode is activated.



or



Should a function not be executed, "negate" the enquiry with **N** or **ESC**.

After you are through with the program enquiries in each case, you will return to the starting situation.

### Note

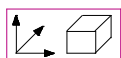


or



Apart from the above menu guidance, the "Permanent Sections" and "Tool Display" functions can also be activated directly with the **W** and **P** keys from the "Setup Mode" and "Automatic Mode" menu levels. The further dialogue guidance is as described above.

## Displaying the Machine Table Axes

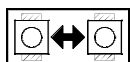


Apart from sectional displays, scaled machine table axes can be displayed or switched off as special function (cf. Figure). The scaling refers to the workpart zero point.



The **F3** key is a toggle to switch on and off the scaled axes.

## Contour Display



There is also a special function with which the workpart contours can be highlighted in color (cf. below, Color settings):



The **F4** key is a toggle to switch on and off the contour display.



You can change the screen display during workpart machining any time. Always try to select the most appropriate display mode!

Figure 10.2.3-1: CNC Turning, Graphical Display; the current NC line is displayed in the upper dialogue line during Automatic Mode. The number of NC lines can be increased in the "Graphical Display" menu.

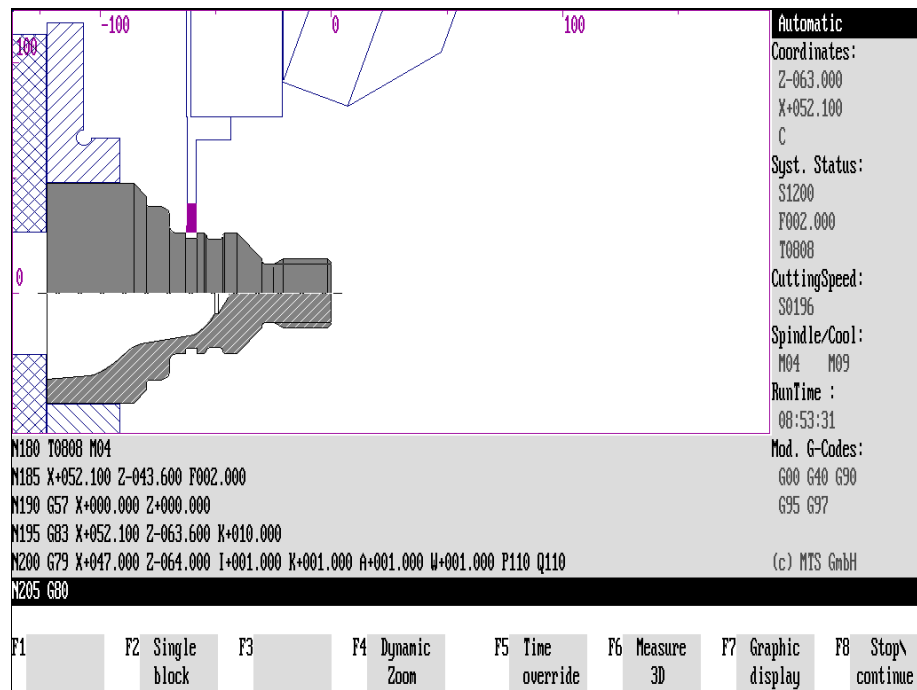
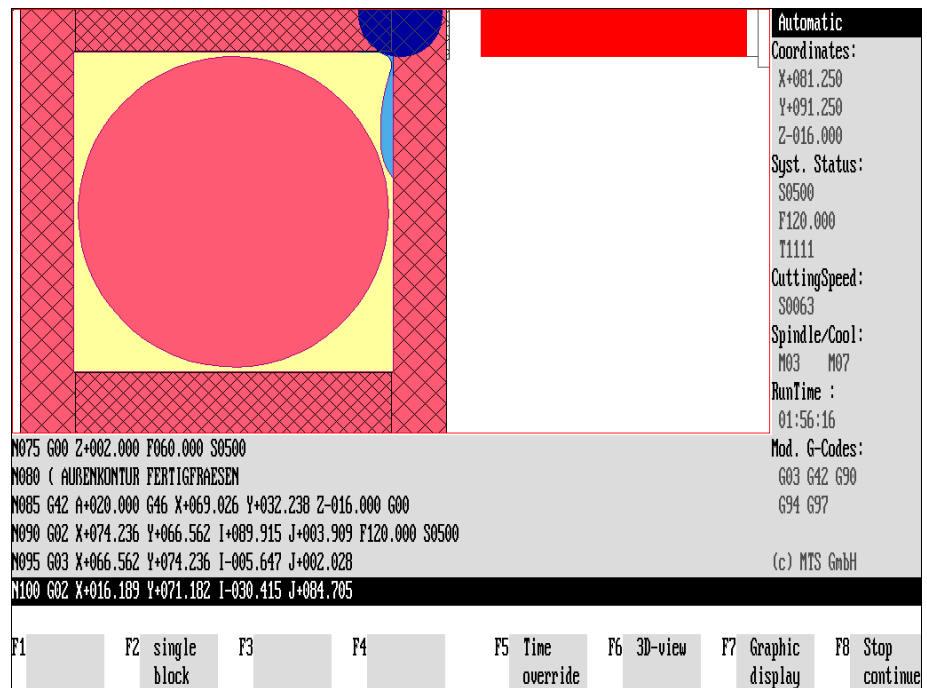


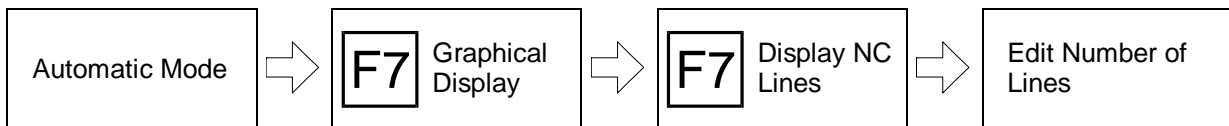
Figure 10.2.3-2: CNC Milling, Graphical Display; both in turning and in milling, the number of displayed NC lines can be varied.



### 10.2.3 Displaying the NC Lines

In the Automatic Mode, the current NC block is displayed in the dialogue line parallel to machining. If several NC blocks are to be displayed, the number of the displayed blocks can be changed in the "Graphical Display" menu, which is called up in the Automatic Mode:

#### Selection:



GRAPHIC

The menu for changing the graphical display is called up with **F7** in automatic mode as well as during interactive programming.

#### Procedure

**F7**

To edit the number of displayed NC lines, press **F7**.

A submenu with which you can set the required number of lines is then loaded:

**F1** or **F2**

**Line +/-:** To increase the number of lines, press **F1**, to reduce it **F2**; the number of lines is then increased or reduced by one in each case.

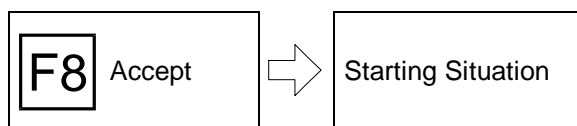
**F5**

**Default configuration:** One NC line is displayed as standard setting. You can switch (back) to this standard with **F5**.

**F8**

**Accept:** After you have selected the required number of lines, confirm this setting with **F8**. You return simultaneously to the "Graphical Display" menu.

#### Return:



**F8**

To return to the starting situation, quit the "Graphical Display" menu with **F8**.

Figure 10.3-1:  
CNC Turning,  
Zoom;  
The Zoom function can  
be called from various  
operation modes using  
the "Graphical Display"  
or "Dynamic Zoom"  
menu.

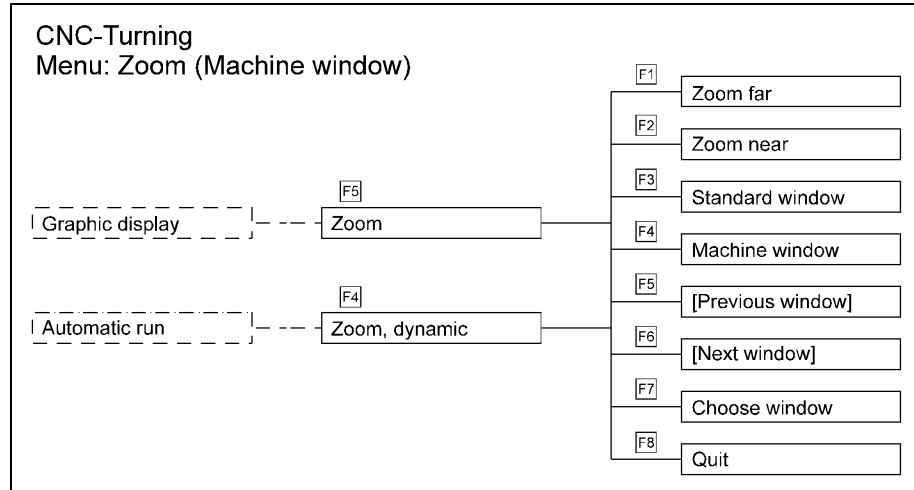


Figure 10.3-2:  
CNC Turning,  
Graphical Display;  
You call up the Zoom  
function with **F5**.

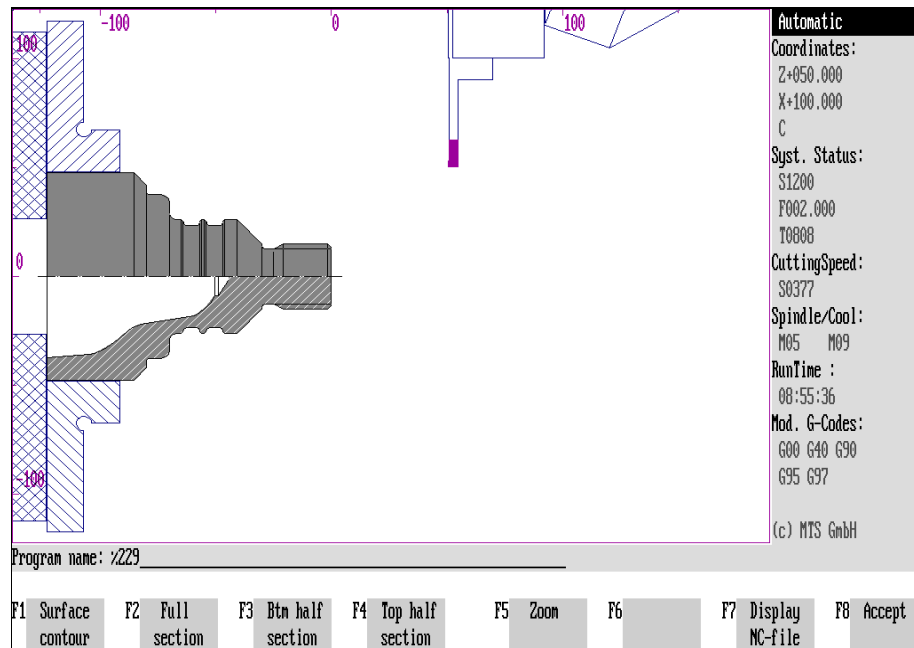
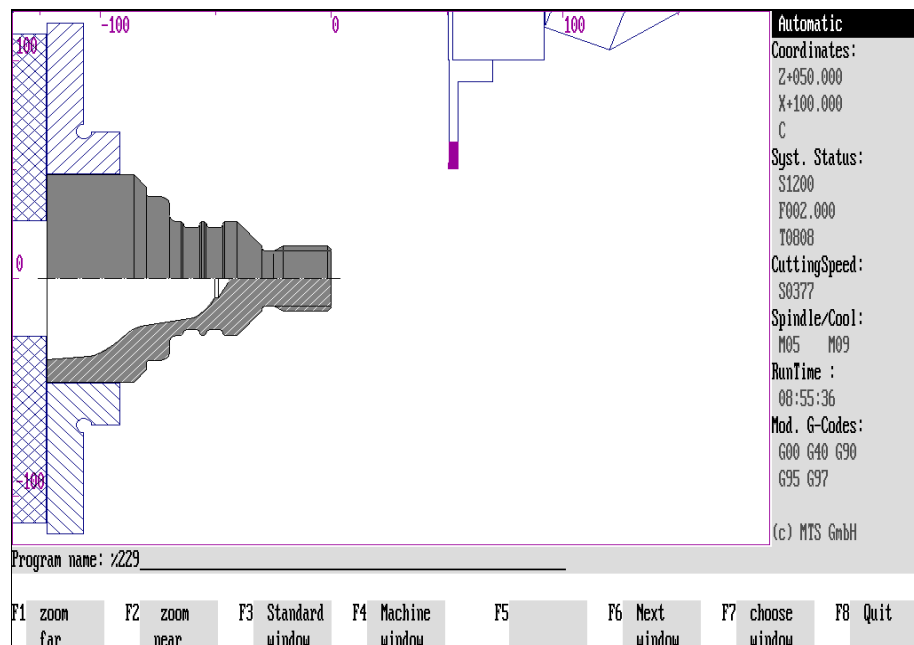


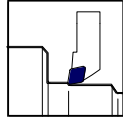
Figure 10.3-3:  
CNC Turning,  
Zoom;  
You select  
enlargement/reduction  
either via standardized  
menu functions such  
as "Zoom up" or  
"Zoom down", or you  
determine the wanted  
iarea to be displayed  
via "Select region".





## 10.3 Zooming

### Enlargement



The screen display of the turning CNC Simulator shows as standard setting a partial representation of the work space of the machine tool with the blank or workpart and clamping fixtures displayed in it as well as workpart machining. With the help of the "enlarging function" ("**Zoom**"), selected areas can be displayed and zoomed, the maximum detail resolution showing an area of  $6\ \mu\text{m} \times 8\ \mu\text{m}$ .

### Display Areas

The other way round, naturally the image can also be scaled down or the full machine work space can be displayed.

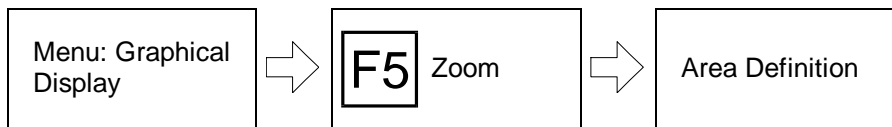
### Note

**In analogy to this zoom function, the "Measuring" function also allows the option of enlargement/reduction. Workpart contours which were programmed in the WOP mode, can also be enlarged.**



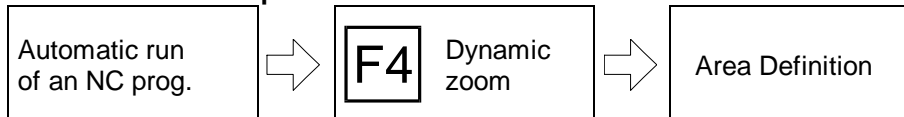
Scaling up or down the work space display (area) can be called up parallel to current processing in Setup and in Automatic Mode or during Interactive Programming, for example, through the "Graphical Display" menu:

### 1st Selection Example:



You call up the zoom function with the function key <F5> in the "Graphical Display" menu. The "Zoom" menu with which you can define new image regions is then loaded.

### 2nd Selection Example:



During the automatic run of an NC program (Automatic Mode), the displayed area can be scaled up or down during workpart machining ("Dynamic Zoom").



Of course, it is possible for you to choose different display areas and change them at any time or according to the machining situation.

Figure 10.3-4:  
CNC Turning,  
Zoom;  
with the "Zoom down"  
function, the displayed  
area is reduced in size  
by approx. 20%.

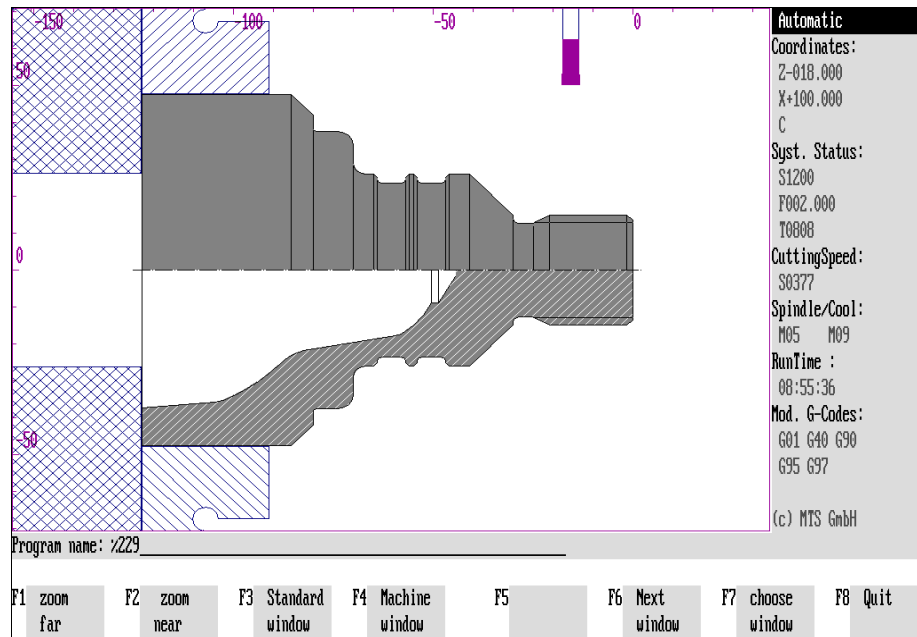


Figure 10.3-5:  
CNC Turning,  
Zoom;  
with the "Zoom up"  
function, the current  
displayed area is  
enlarged by approx.  
20%.

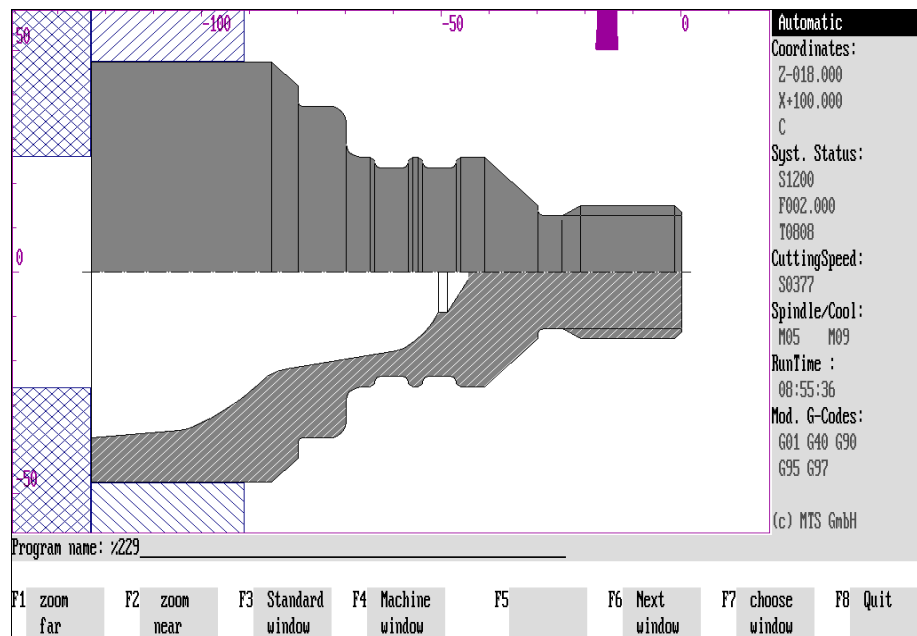
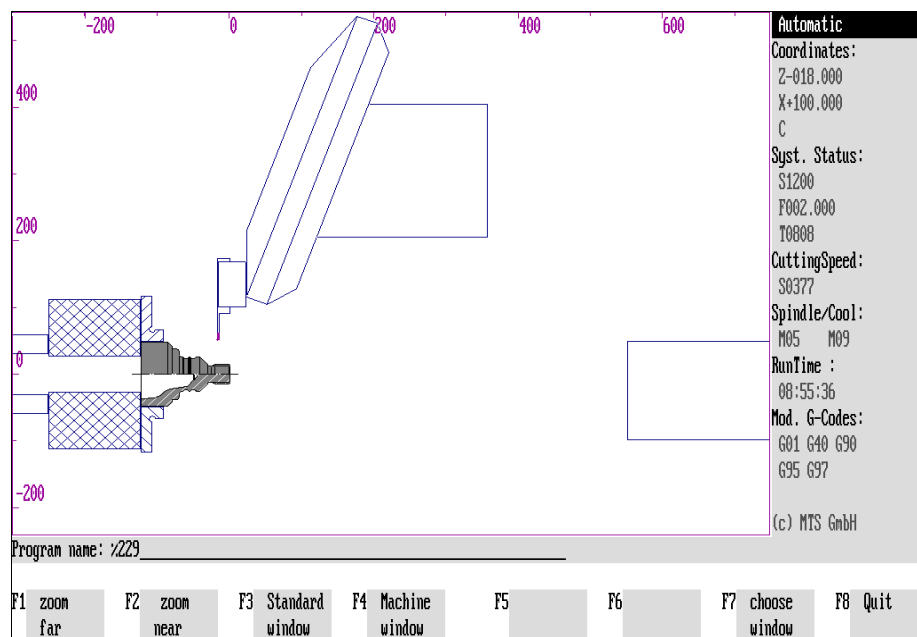
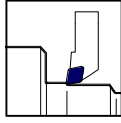


Figure 10.3-6:  
CNC Turning,  
Zoom;  
Display of the work  
space of the machine  
tool.



## Menu: Zoom (Machine Tool Work-Space)

### CNC Turning



This menu includes either standardized options for scaling up or down or you can select the display area yourself:

F1

**Zoom down:** The display is scaled down by approx. 20% in each case with **F1**. The standard window size or the display of the machine work space set the limits for scaling.

F2

**Zoom up:** With this function **F2**, the current display is enlarged by each pressing of the key by 20%. The maximum range of enlargement is at approx. 6  $\mu\text{m}$  x 8  $\mu\text{m}$ .

F3

**Standard window:** A standard window displaying the entire workpart and the clamping fixtures in an ideal way is loaded with **F3**.

F4

**Machine space window:** The entire work space of the machine tool can be displayed with **F4**.

F5

**Last region:** Several defined areas are stored as long as you work with the CNC Simulator. You call up the preceding area with **F5**.

F6

**Next region:** If several areas are defined, you call up the next one with **F6**.

F7

**Select region:** If you want to define the area to be scaled up yourself, switch with **F7** to the "Select region" menu.

F8

**Return:** Press **F8** to return to the "Graphical Display" menu.

Figure 10.3-7:  
CNC Turning,  
Zoom;  
Schematic of the  
menu sequence.

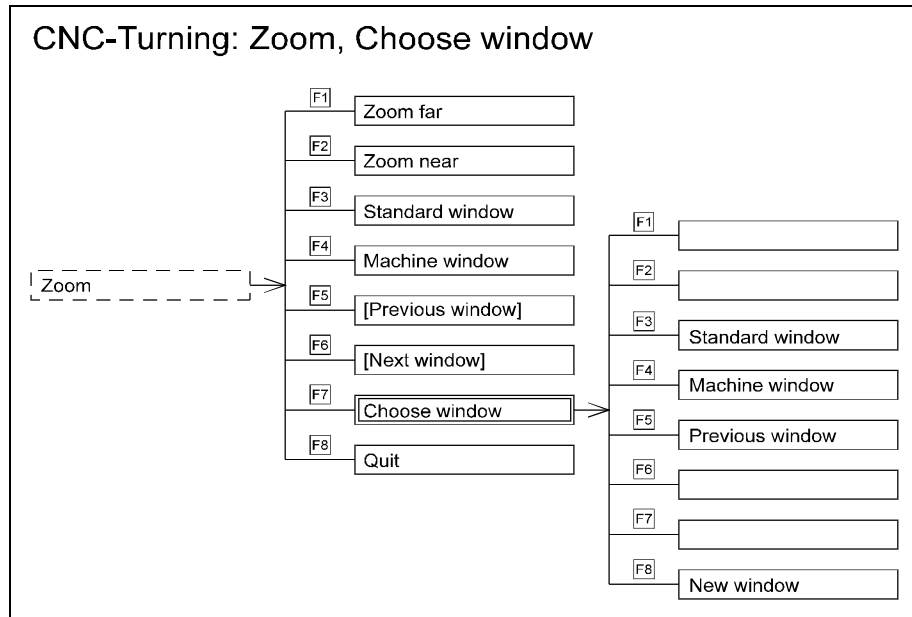


Figure 10.3-8:  
CNC Turning,  
Zoom  
Determining the zoom  
region: you define the  
wanted detail with the  
inserted rectangle.

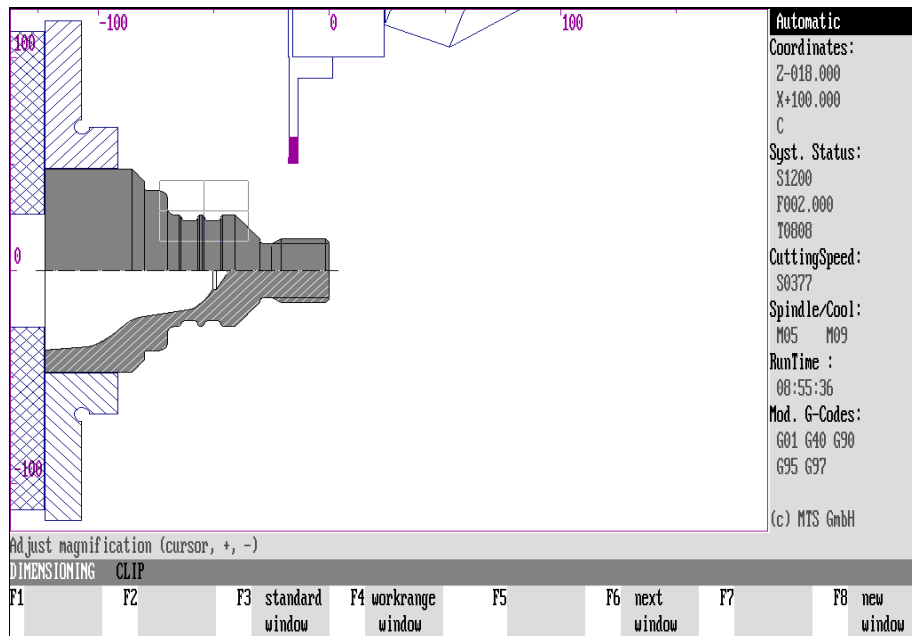
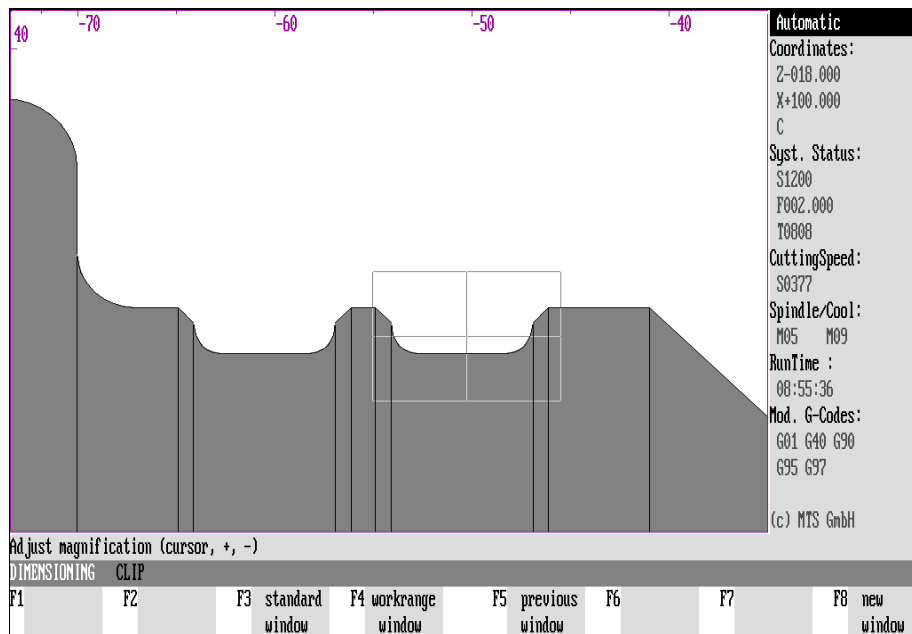


Figure 10.3-9:  
CNC Turning,  
Zoom;  
Display of the detail  
selected above; a  
further detail can then  
be defined or you  
return to the starting  
situation with **[ESC]**.

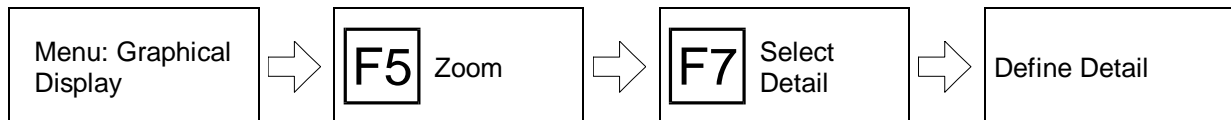


## Defining the Zoom Detail

To enlarge a certain image area, it must be defined first. For this purpose a square frame is displayed on the screen.

This square frame can be moved on the screen and enlarged or reduced in size. After confirmation of the area definition, the selected detail is displayed in full screen size.

### Selection Example:



Proceeding from the "Zoom" menu, you call up the "Select detail" function with **F7** to define a certain screen area enlargement:

**F3**

**Standard window [last detail]:** A standard window displaying the ideal setting of the entire workpart with clamping or workpart machining is loaded with **F3**. If the standard window is active, you call up the last detail with **F3**.

**F4**

**Machine space window [last detail]:** The entire work space of the machine tool can be displayed with **F4**. If the machine space window is active, you call up the last detail with **F4**.

**F5**

**Last detail:** You call up the previously defined detail again with **F5**.

**F6**

**Next detail:** If several areas are defined, you call up the next one again with **F6**.

**F8**

or



**New detail:** You confirm a detail without quitting the menu with **F8**, you confirm a detail and return to the starting situation with .

### Procedure

A detail display is defined in two steps:

- 1.) The inserted square frame is positioned and if necessary also enlarged or reduced in size.
- 2.) You then confirm your selection.



or



Use the cursor keys to position the frame in the right place.



or



You can move the frame a bit faster with the key combination and <"cursor key">, e.g. and .



or



The square is enlarged with the key and reduced in size with the key.

**F8**

or



**New Detail:** After the square has been positioned the right place, confirm with or **F8**. The enlargement of the framed area is then carried out.

**ESC**

**You return to the starting situation with .**

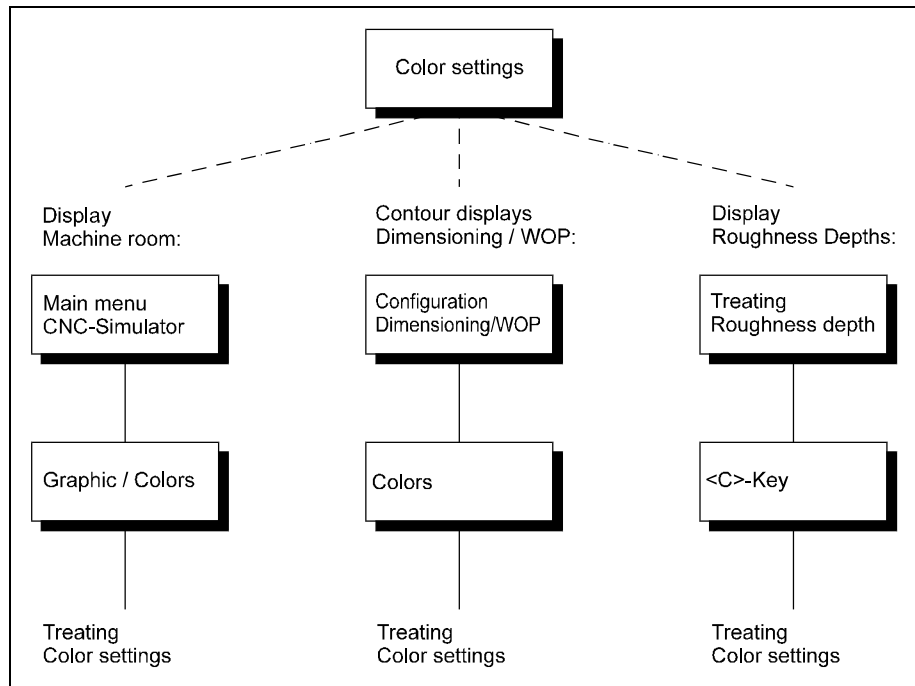
### Note

The display with a zoomed detail remains valid until you define a new way of representation or call up an existing window again.



**Get some practice in the zoom function and enlarge and reduce different screen areas.**

Figure 10.4-1:  
Schematic overview of  
the different  
possibilities of  
changing the color  
settings.



## 10.4 Color Settings

The shades and colors of the screen display are selected in the color settings. Thus it is possible to design the representation of the tool, clamping fixtures or workparts etc. similar to the representation on the display of a machine tool. Or if individual work stations are equipped with different monitors, you can match the color display to each another with corresponding settings.

The color settings can be changed and stored **during processing**. A special configuration is not necessary.

Depending on the processing situation, different "screen interfaces" are provided to support the work steps. Accordingly there are various possibilities for defining the color settings. Thus the display of the workpart machining contains other screen elements than the measuring and WOP program sections or the acquisition of the surface peak-to-valley heights.

Basically there are **two different menus for color definition** :

- Settings for the display of the work space of the machine tool;
- Settings for the contour displays in the WOP, measuring and acquisition of peak-to-valley height program parts.

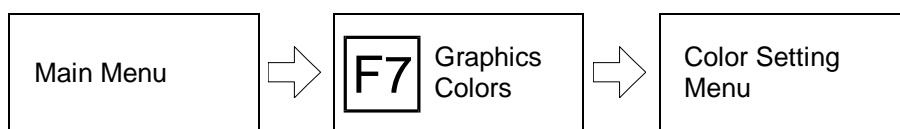


Naturally you can decide whether you want to change the color settings at all. And also it is not necessary to make use of all of the above possibilities at once. However, note that certain fields of the available color scale ("palette") define the coloring of various screen objects. A change can therefore lead to a situation where screen objects in a number of submenus become invisible or vague.

### 10.4.1 Color Setting for CNC Simulator

The menu for changing the colors in the display of the work space is called up from the main menu of the CNC Simulator:

#### Selection:



If you are currently working in one of the operation modes of the CNC Simulator, first return to the main menu with **F8**.

The menu for color setting is called up with the **F7** key.

Should you have defined a **password** for protection, you must first enter this for further processing and confirm it with **↵**.

#### Note

In milling the menu for color settings is loaded immediately after this, in turning, however, the CNC Simulator offers an intermediate menu for processing:

Figure 10.4.1-1:  
CNC Turning,  
Color Settings  
Intermediate menu;  
In this menu certain  
settings for the screen  
display are specified.

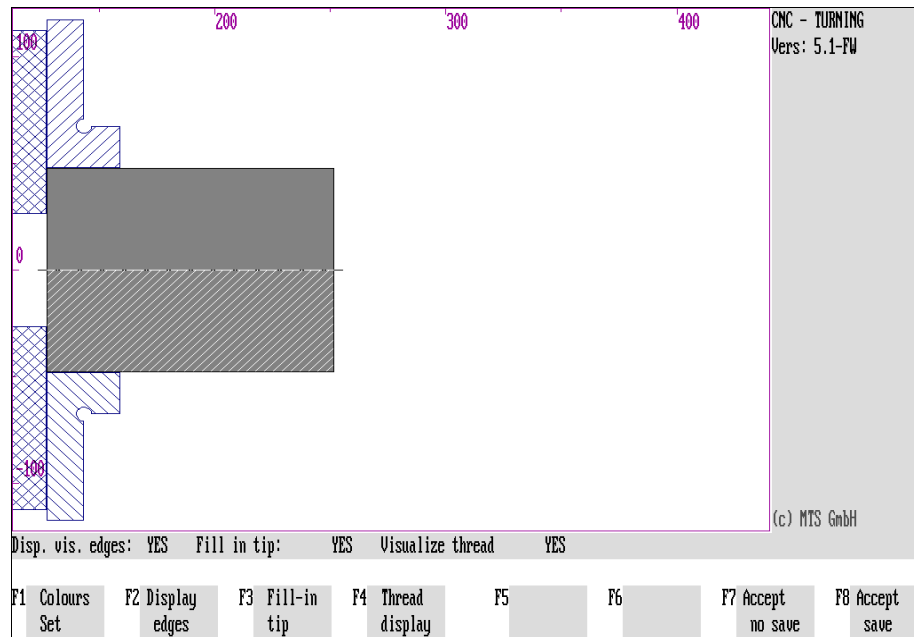


Figure 10.4.1-2:  
CNC Turning,  
Color Settings  
Both the selection of  
the available set of  
color shades (palette)  
and the assignment of  
certain colors to  
individual screen  
elements are part of  
the color setting.

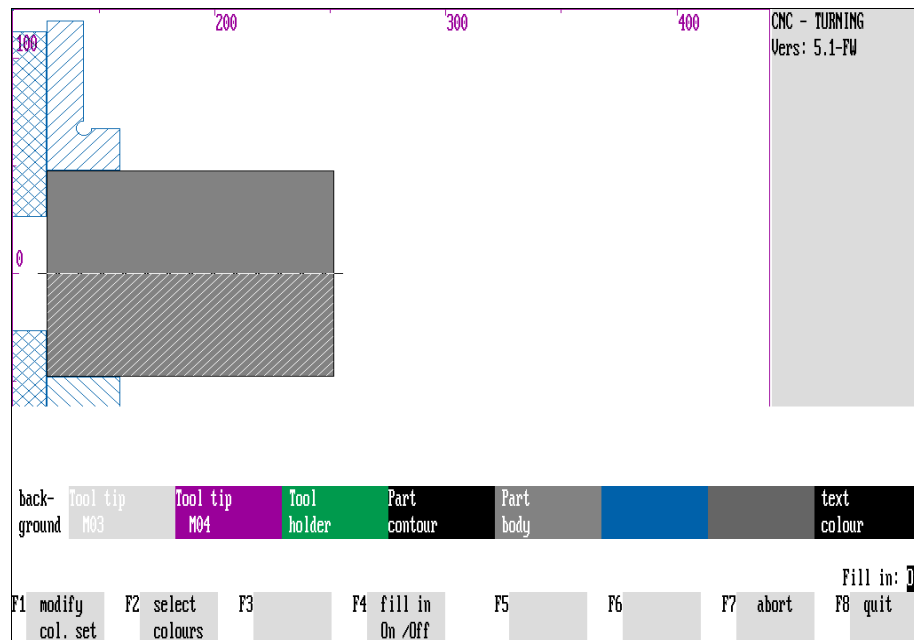
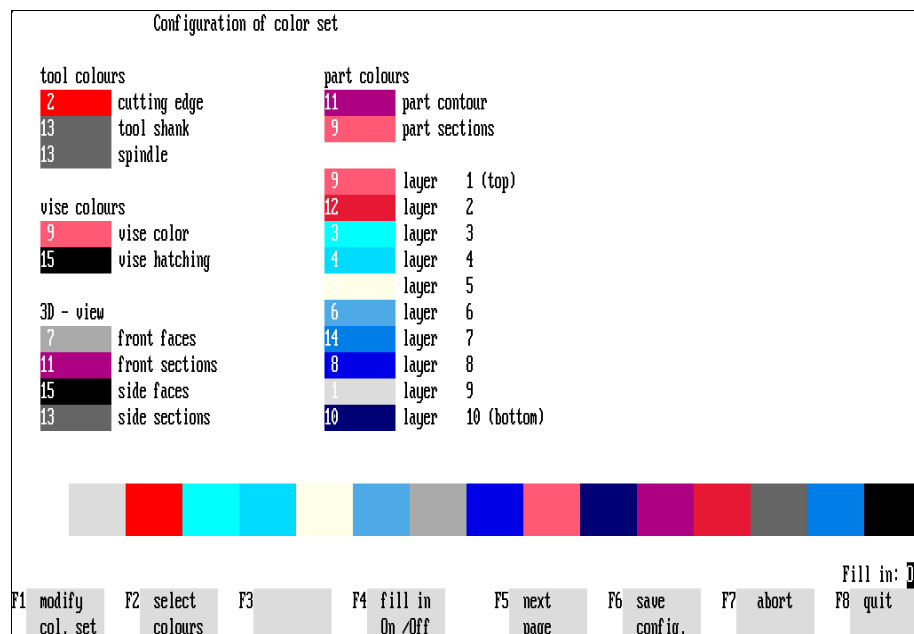
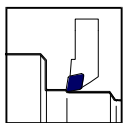


Figure 10.4.1-3:  
CNC Milling,  
Color Settings  
As in turning, the color  
display can be  
redefined in milling as  
well.





## CNC Turning: Intermediate Menu for Color Setting



The intermediate menu of the turning Simulator provides the opportunity to specify some graphics display settings. This may be an advantage especially for computers with low computing capacity. (Functions requiring long computing times are marked in the following with (\*), those requiring large memory space with (+).) The currently selected settings are stated in the information line.

F1

**Colors/palette:** You call the menu for color setting with **F1**.

F2

**Visible edges (\*)**: The display of the visible and clearance edges can be switched on and off with **F2**. Rounded edges are called "clearance edges"; they are marked with lines which end before the physical edges.

F3

**Fill out cutting edge (+)**: You define whether the tool cutting edge is to be filled out in color or only its outer contour should be displayed with **F3**.

F4

**Update thread (\*+)**: With version 5 of the turning Simulator, thread geometries are calculated and displayed exactly. With low computing and storage capacity, it is possible to switch off this function with **F4**.

F7

**Accept without saving**: If you want to confirm the changes for the current machining only, return to the main menu with **F7**.

F8

**Accept with saving**: You store the changes and return to the main menu with **F8**

ESC

You interrupt processing with **ESC** without confirming the changes made.

## CNC Turning & Milling: Menu for Color Setting

If you switch from the intermediate menu into the color and palette setting, the following processing options are available.

F1

**Change palette**: The "color palette" defines the available color shades of the screen display. It is called up with the function key **F1**.

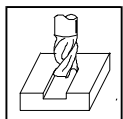
F2

**Change colors**: To allocate the colors to the screen elements, press **F2**.

F4

**Coloring on/off (\*)**: With **F4** you switch the option of coloring the workpart during machining on or off. The colors to be used are specified in the "Change Colors" menu.

### CNC Milling



You can define ten different colors as different "depth shades" in the "Change Colors" submenu. You define with this switch whether the machining depths are colored accordingly in the top view.

F5

**Next page**: Only the CNC Simulator for milling offers this function to display the other color definitions.

F6

**Save configuration**: Since the milling CNC Simulator does not have the above intermediate menu, the "permanent storage of the changes" function is integrated separately here.

F7

**Cancel**: If you want to cancel processing without confirming the changes made, press **F7**.

F8

**Return**: You confirm your changes and return to the Setup mode with **F8**.

Figure 10.4.1-4:  
CNC Turning, Color  
Palette;  
The color values are  
"mixed" according to  
the red/green/blue  
proportions determined  
in the setting of the  
"Color Palette".

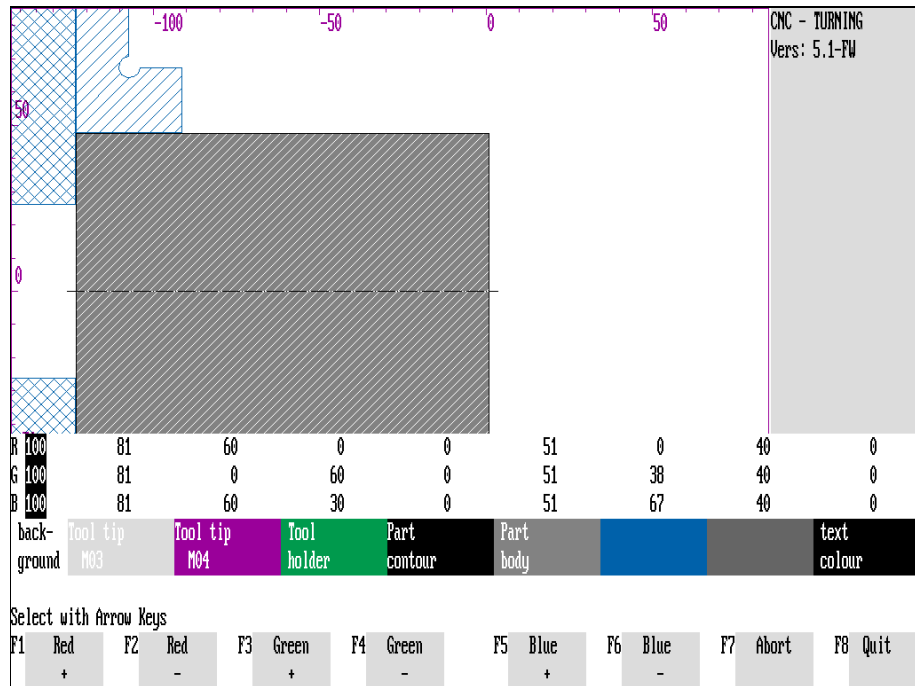
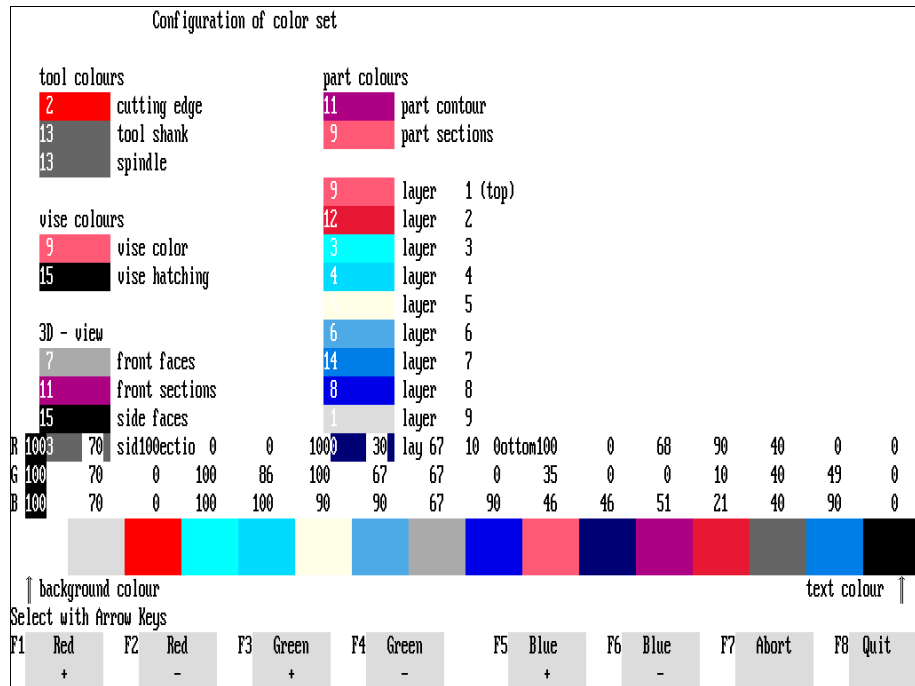


Figure 10.4.1-5:  
CNC Milling, Color  
Palette;  
The milling Simulator  
provides 16 color fields  
whose color values are  
always a mixture of  
red/green/blue in  
varying proportions



## Change Palette

In total nine palette fields are available in turning and sixteen in milling for the definition of various color shades (cf. Figure, square fields above the function keys). Each color shade is a result of a mixture of the so-called "RGB proportions" (RGB = Red/Green/Blue), which can be determined with the "Change Palette" function:

### Procedure

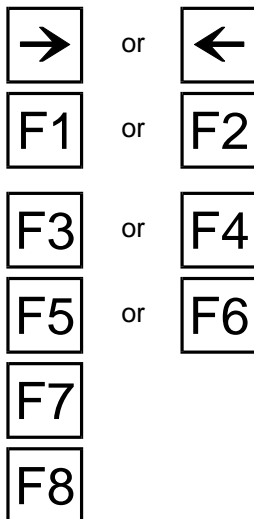
In the submenu for the "Change Palette" function, the corresponding values of the RGB scale are displayed for the individual color shades above the color fields. Furthermore, the currently selected column has a different color background. The values of the RGB scale are between 0 and 100. Depending on the graphic mode different gradations are possible:

e.g. in the EGA graphic mode, the numerical values for the relevant color proportions are:

0      33      67      100

Furthermore, the outer color fields determine the **background color** (on the left, 1st field) and the **text color** (on the right, 9th or 16th field). This allocation does not depend on the color assignments of the further display elements.

Finally the different RGB proportions can be increased (+) or reduced (-) and thus the color shades "re-mixed". These changes are made in two operational steps:



The palette area whose shade is to be changed is selected with the cursor keys or .

[Color +/-]: You can then increase or reduce the RGB proportions with the function keys to ; the color in the active palette field changes simultaneously.

As an orientation for "color mixing" use the data of the adjacent fields; the color black has the RGB values 0/0/0, white the values 100/100/100.

**Cancel:** If you want to cancel processing without confirming the changes made, press .

**Return:** With you confirm the changes and return to the "Color Settings" menu.

Figure 10.4.1-6:  
CNC Turning, Color  
allocation  
In the color allocation,  
the colors are  
allocated to further  
representations of the screen  
display.

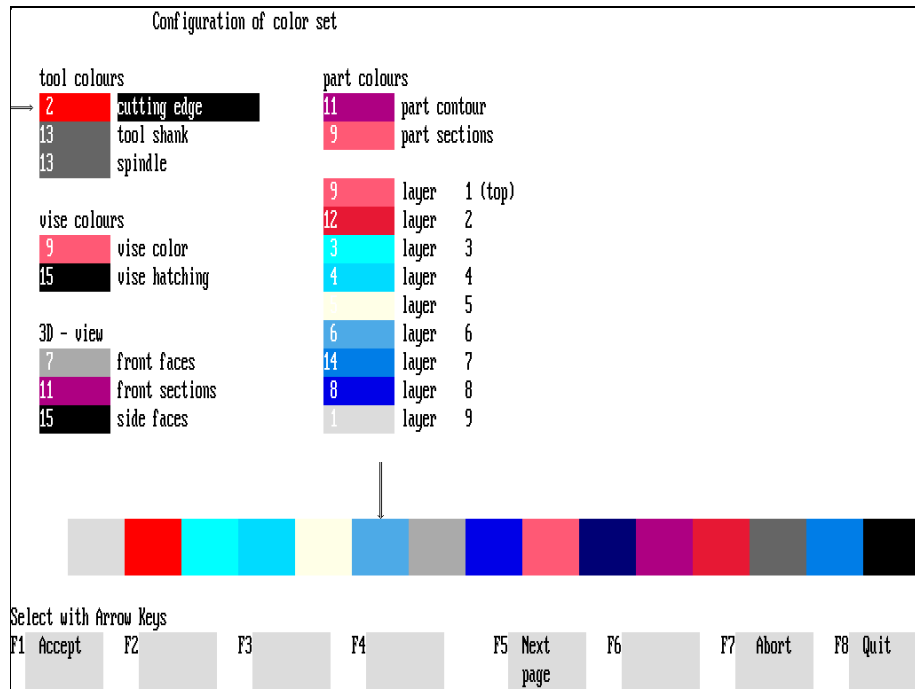
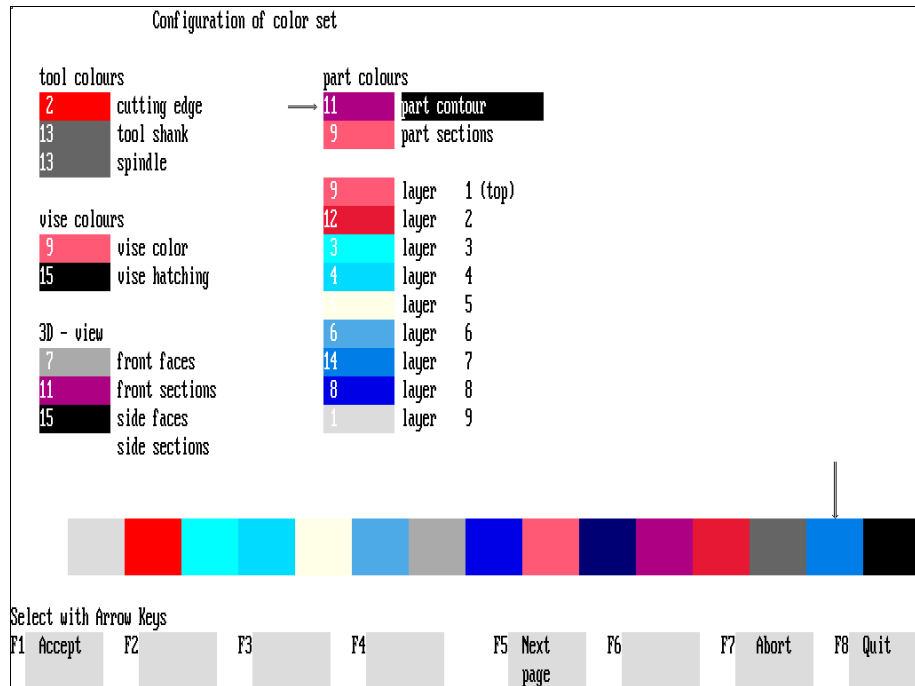


Figure 10.4.1-7:  
CNC Milling, Color  
allocation  
Selection of screen  
representations for  
color setting.



## Change Colors

With this function, you allocate a specific color from the "Color palette" to certain screen elements.

The various elements are listed in the upper screen half. As their designation is clear (cf. Figure), the list is not repeated here.

The square fields in front of the object description in the upper screen half indicate the current colors. To change these, the object area is selected first and then a color is assigned.

### Procedure

After you have called up the function "Change Colors" with **F2**, a horizontal arrow is set in front of one of the object specifications to activate this item and one of the palette fields is also marked with a vertical arrow.



or



Select the object whose color is to be changed with the cursor keys **↑** or **↓**. The designations in the list indicate the relevant object (cf. Figure; different machining depths are designated as "layers" in CNC milling).



or



Then mark with the cursor keys **→** or **←** the color shade in the color palette.



**Accept:** To confirm the color assignment set by the two arrows, press **F1**. The color field in front of the object specification is then changed accordingly and you can define a new object.



**Next page:** In the CNC Simulator for milling, the remaining objects are displayed with **F5**.



**Cancel:** You cancel processing with **F7** without confirming changes made.



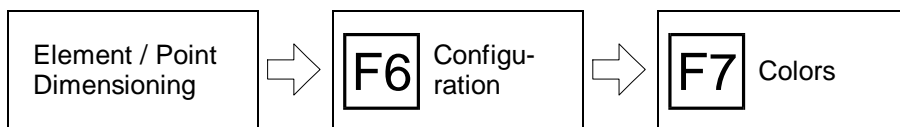
**Return:** With **F8** you confirm all color changes made and return to the starting menu.



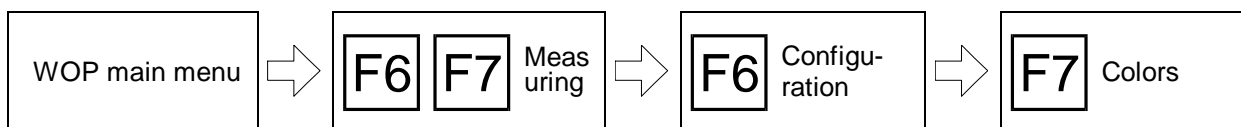
### 10.4.2 Color Settings for WOP, Measuring, Surface Peak-to-Valley Height Calculation

The second color definition option covers the screen elements of the program functions of Measuring, Workshop-Oriented Programming and those of Surface Peak-to-Valley Height calculation - this means, among other things, the screen background, the workpart contours and edges as well as the individual menu fields, the dialogue line, information column, input fields etc.

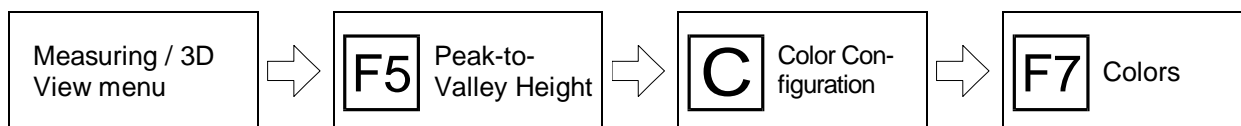
**Selection Example: Measuring, CNC Turning**



**Selection Example: WOP, CNC Turning & Milling**



**Selection Example: Surface Peak-to-Valley Heights, CNC Turning**



**F6**

**Configuration:** Proceeding from the point or element dimensioning, first call up the "Configuration" menu item with **F6**.

**F7**

**Colors:** To change the color display, select the "Colors" item in the configuration menu with the function key **F7**.

**C**

During the **peak-to-valley heights** calculation, you can call up the color settings for this processing menu directly with the **C** key.

#### Menu: Color Settings

A submenu with two processing possibilities is then loaded: as in the color configuration of the CNC Simulator (cf. above), both the color shades can be defined and the relevant colors can be assigned to the screen elements:

**F1**

**Change palette:** The "Color palette" defines the available color shades of the screen display. It is called up with the function key **F1**.

**F2**

**Change colors:** To define the assignment of the colors to the individual screen elements, press **F2**.

**F6**

**Store configuration:** You store the changes with **F6**.

**F7**

**Cancel:** If you want to cancel processing without accepting changes made, press the function key **F7**.

**F8**

**Return:** With **F8** you confirm the changes for the current processing.

## Change Palette

As in the CNC Simulator for milling, 16 palette fields are available to define different color shades (cf. Figure above, square fields above the function keys). The "Change palette" function is identical with that described above and need not to be described here.

## Change colors

In the menu for color configuration, the upper screen half contains the names of the objects which can be assigned a color separately. This function is identical to that already described and is therefore not repeated here.

The abbreviations in the measuring/WOP object list stand for the following:

<b>FuncKeys BG</b>	Background of the function key fields
<b>FuncKey Lab</b>	Labels of the function key fields
<b>FuncKey Number</b>	Numbering of the function key fields
<b>Status Text</b>	Color of the text in the status line
<b>Status BG</b>	Background color of the status line
<b>Geo. elements</b>	Color of the geometry elements (only WOP)
<b>Contour color</b>	Color of the contour elements
<b>Alternatives</b>	Display of one of several possibilities (only WOP).
<b>Help Screen BG</b>	Background color of the help screens (only WOP)
<b>Help Screen lines</b>	Color of the drawing in the help screens (only WOP).
<b>Cursor</b>	Color of the cursor and of the frame
<b>Scale</b>	Color of the graduation
<b>Visible edges</b>	Color of the visible edges
<b>BG elements</b>	Color of the background elements (only WOP)
<b>Elem. selection</b>	Color for identification of elements (only WOP).
<b>Text line print</b>	Color of the output in the text line
<b>Error Mess.</b>	Color of the error messages
<b>NC block BG</b>	Color of the NC blocks
<b>Dimension line</b>	Color of the dimension line (only point dimensioning)
<b>Dimens.aux.line</b>	Color of the help lines (only point dimensioning)
<b>Dimens.text</b>	Color of the dimension text (only point dimensioning)
<b>Elem.mark</b>	Color for marking elements (only measuring)
<b>End points</b>	Color of the end points (only element dimensioning)
<b>Program ZP.</b>	Color of the program zero point
<b>Workpc. Zp.</b>	Color of the workpart zero point
<b>Text column BG</b>	Background color of the text bar in dimensioning
<b>Heading</b>	Print color : headings in dimensioning
<b>Standard text</b>	Print color of the output dimension values
<b>Special text</b>	Print color for the "incr. zero point" text (element menu).

### Note

If positioned on each other in the screen display, some objects are displayed in the complementary mode, i.e. black <-> white, blue <-> yellow, red <-> green etc.



The abbreviations in the peak-to-valley height object list stand for the following:

<b>unmachined</b> <sup>1</sup>	Coloring of the unmachined workpart contours
<b>unmach. &amp; mark.</b> <sup>1</sup>	Coloring of the unmachined workpart contours, which are currently marked
<b>machined</b> <sup>1</sup>	Coloring of the machined workpart contours
<b>mach. &amp; mark.</b> <sup>1</sup>	Coloring of the machined workpart contours, which are currently marked
<b>SP-EP marking</b> <sup>1</sup>	Color of the starting and end point
<b>Rough contour</b> <sup>1</sup>	Color of the displayed rough contour
<b>R. contour &amp; mark.</b> <sup>1</sup>	Color of the displayed marked rough contour
<b>Inactive frame</b>	Color of the frame of the inactive screen window
<b>Active frame</b>	Color of the frame of the active screen window
<b>Det. box in overview</b>	Color of the detail display in the left screen window
<b>Scale</b>	Displayed scale for the displayed peak-to-valley height
<b>Axis color</b>	Color of the rotation axis
<b>Text column BG</b>	Color of the NC blocks
<b>Text column header</b>	Color of the NC blocks
<b>Text column text</b>	Color of the NC blocks
<b>Header BG</b>	Menu header, background
<b>Header text</b>	Menu header: print color
<b>Text line 1 BG</b>	Information line: background
<b>Text line 1 Text</b>	Information line: print color
<b>Text line 2 BG</b>	Dialogue line: background
<b>Text line 2 Text</b>	Dialogue line: print color
<b>Function keys BG</b>	Color of the function key background
<b>Function keys text</b>	Color of the function key lettering
<b>Function keys No.</b>	Color of the function key numbering
<b>Zero point color</b>	Color of the workpart zero point

<sup>1</sup> For clarity, the assigned colors should be changed neither in the color allocation nor in the palette setting.

**F8**

**ESC**

or [Password]

You conclude the color configuration with **F8**. There is then a program dialogue in which you either **enter the password** or confirm with **ESC** or **F8**.

Figure 11.1-1:  
CNC Simulator;  
Programming Aids  
menu schematic.

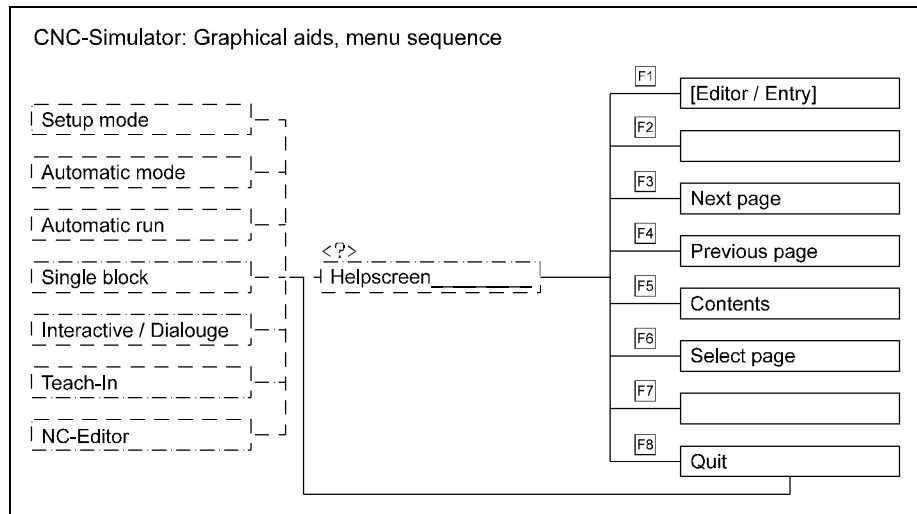
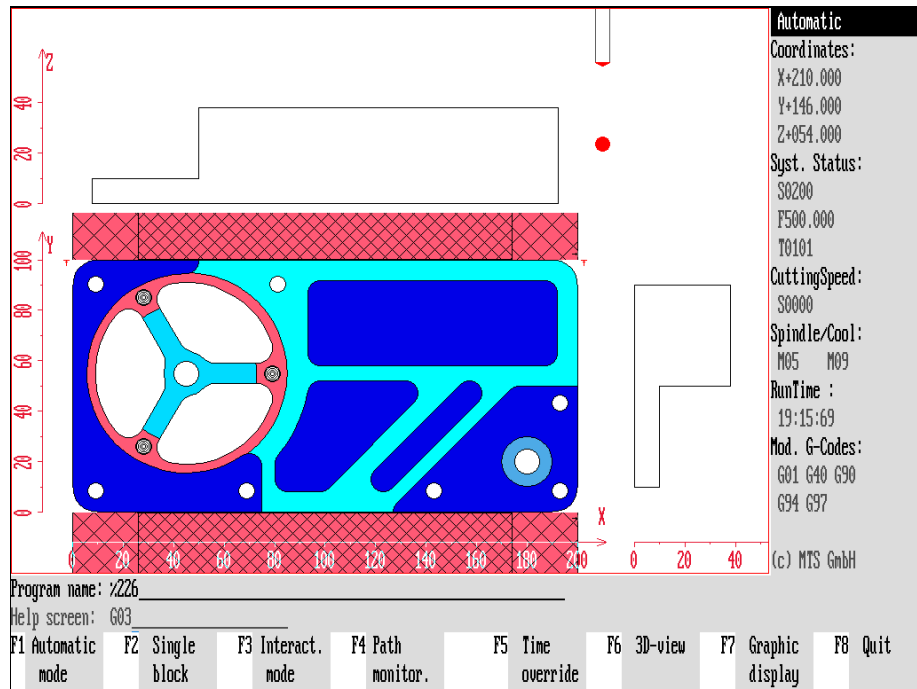


Figure 11.1-2:  
CNC Simulator,  
accessing a  
Programming Aid;  
after the help function  
has been activated, its  
name is displayed in  
the dialogue bar.



## 11. Miscellaneous Functions

To supplement the program features, miscellaneous additional functions, such as programming aids, as well as the option of generating a printout of the current screen display, are provided.

### 11.1 Programming Aids

Whenever you need assistance or have a question during the creation or testing of an NC program, you can call for help. The available help functions provide basic DIN-based NC programming information, and contain detailed descriptions of the MTS programming code.

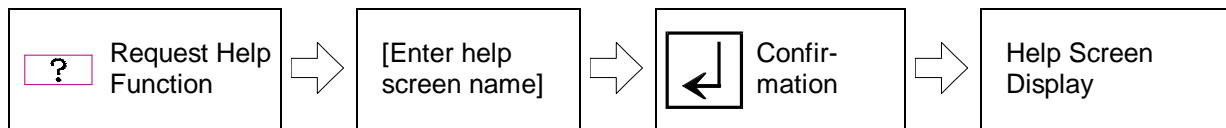
The programming aids and control information are available in the form of "help screens", and are directly displayed in the graphic window on the monitor. As a rule, they contain a short informative text, a context-based graphic display, and an application example referring to the subject.

Almost 200 help screens are available. Based on subject matter, they are divided into groups; for better orientation, each group is preceded by a table of contents or a schematic function display. In this manner, you have almost the entire set of programming instructions available, without having to interrupt programming.

#### Note

If required, the scope of available information can be individually extended, and tailored to meet the requirements of the user. The optionally available MTS program package "CAL Generator" can be used to generate new help screens, or to modify existing ones for subsequent reintegration into the CNC Simulator.

Selection:



Accessing the Programming Aids is possible from almost all working situations within the CNC Simulator; the <?> key is used for this purpose during:

- **Setup Mode,**
- **Automatic Mode,**
- **Interactive Programming,** and
- **Teach-In Programming.**

Since the "?" (question mark) character is used in NC programming comment texts, the **F6** function key is used for calling help functions while working with the **Editor**.

[Designation]

Subsequently you enter the name of the help screen in the dialogue line. The corresponding program message is displayed, "Help screen: \_\_\_\_\_"



Confirm the name entry by pressing . If a help screen with that name is available, it is then loaded. If not, the error message: "Help screen not found" is displayed.



and

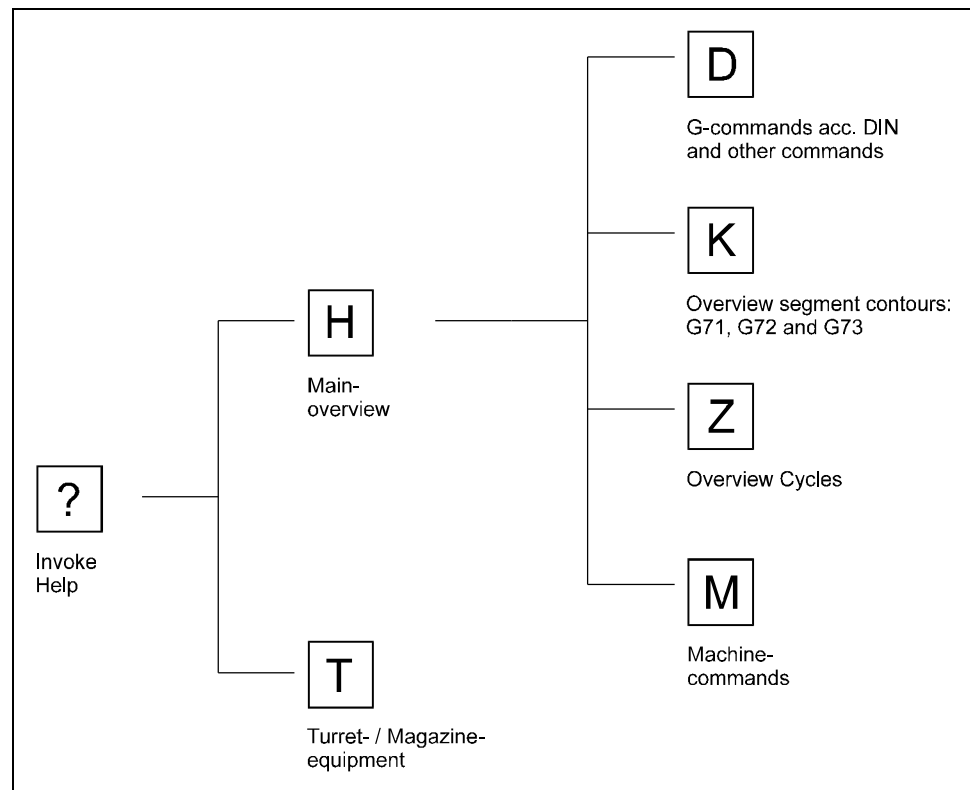


You can use the **CTRL** + keys on the PC keyboard, or the <CONT> key on the CNC keyboard to recall the previous help screen.



Accessing a help function during a program dialogue (i.e., "Accept NC block?" in interactive programming) is not possible. In this case, first finish the dialogue and then call the help function.

Figure 11.1-3:  
CNC Simulator  
Help Function;  
Overview of the main  
programming help  
groups; all primary  
Help Functions ("H",  
"D", "F", "K", "Z", and  
"M") contain their own  
table of contents, and  
have priority over the  
secondary, context-  
based groups.  
Subsequently, you can  
address a help function  
directly, or scroll  
through one of the  
groups, until you have  
located the desired  
information.  
The "T" help function is  
a special function  
allowing a direct  
display representation  
of the current turret or  
magazine  
configuration.



## Names/Designations and Group Assignments

### Designation

In order to facilitate direct access to help functions, each help screen, has a designation of its own (also called Help Name in the following). In the same fashion, the tables of content and overview schematics are identified with names. The uppermost table of content in the hierarchy is called by pressing the **[H]** key.

**The names of the individual help screens are generally based on the designations of the corresponding commands.**

For example: the name of the help function for linear interpolation is identical to the appropriate DIN command, namely "G01"; the help function for the thread tapping cycle is "G84", etc.

### CRC / Segment Contour Programming

The only exceptions are the overviews, the information on cutter radius compensation (CRC), and the subgroups for segment contour programming:

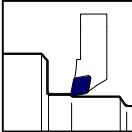
Since on the one hand, different combinations of several NC blocks are possible, very often the only difference being the address characters, but on the other hand, the names of programming aids cannot exceed a length of eight characters, additional subgroups had to be formed, and names had to be shortened:

The linking possibilities of geometrical elements, such as straight line and circular arc, were selected as subgroup identifiers. A further differentiation into left-hand and right-hand orientated arcs would have needlessly multiplied the number of help screens. For this reason, only one example is presented in each case (either "G72" or "G73"). To call a help function for a specific contour segment string, first search the respective names of the required help in the table of contents, or scroll through one of the subgroups until you find the required information.

### Turret/Magazine

In addition to the NC programming information discussed here, the help function also provides you with the option of displaying the current turret/magazine configuration. This function is accessed by pressing the **[T]** key.

The following sections present a help screen designation overview that is structured according to the group assignments: Magazine configuration, "G" commands based on DIN programming standards, segment contour programming (subgroups only), machining cycles and machine commands.

 Selection	<b>CNC Turning Programming Aids</b>
T H D K Z M	Current turret configuration (tooling) Turning, Primary overview DIN programming, G commands and other commands Contour segment string, Overview: G71, G72, G73 Cycles overview: G31, G57, G75 to G88 Machine commands
[D] G00 G01 G02 G03 G04 G09 G22 G23 G33 SRK G41G42 G40 G53 G54 G59 G90 G91 G92 G96 G97	Rapid traverse Linear interpolation Circular interpolation, clockwise Circular interpolation, counter-clockwise Dwell In-position programming Subroutine invocation Program segment repeat Thread cutting, constant lead Tool nose radius or tool path compensation (TNC) Cutter compensation, left (G41) or right (G42) Cancel cutter compensation Cancel incremental datum shift Set datum (global) Incremental datum shift Absolute dimensions Incremental dimensions RPM limit Constant surface speed Cancel G96
[Z] G31 G36 G75 G76 G78 G79 G80 G81 G82 G83 G57 G84 G85 G86 G87 G88	Thread cycle Multiple cycle with travel range limitation Rectangular longitudinal roughing cycle Rectangular transversal roughing cycle Undercut cycle Recess cycle Cycle invocation for G81, G82 and G83 (ibid.) Longitudinal roughing cycle, arbitrary connecting contour Transverse roughing cycle, arbitrary connecting contour Multiple cycle with travel range limitation G36 Finishing allowance Deep hole drilling cycle Diameter dependent undercut cycle Recess cycle Radius cycle Chamfer cycle
[K] ZA G71 G72 G71G71 G71G72 G72G71 G72G72 VPZ OK P000	Additional addresses for segment contour programming Line (subgroup) Circle (subgroup) Line/line (subgroup) Line/circle (subgroup) Circle/line (subgroup) Circle/circle (subgroup) Four-point strings comprising different contour elements (subgroup) Contour segment strings with indeterminate end point (subgroup) Tangential transition from previous element (subgroup)

**Note**

In defining the help names for the segment contour programming help functions, the NC words "G71", "G72", and "G73" were abbreviated with the numbers "1", "2", and "3", and the respective address characters were added. Two examples:

Contour segment (circle):	G72 X.. I.. K..	Help Name: 2XIK
Contour segments:	G71 A..	
(line/arc)	G72 Z.. I.. K.. B..	Help Name:1A2ZIKB

Also note that although the **sequential order of addresses** is arbitrary during program generation, it is fixed and clearly specified when naming help functions.

**The addresses follow the order of: X, Z, I, K, L, A, B**

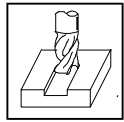
Example: If a programming block reads "G72 K.. I.. Z..", the name of the matching Programming Aid is "2ZIK".

Finally, in **four-point contour segment** definitions the circle centre coordinates I and K are grouped together with M.



**When in doubt regarding the location of Programming Aids for specific NC commands, please refer to the overviews for the corresponding names.**

While you are getting to know the program, it is helpful to scroll through the subgroups by pressing the **F3** or **F4** function keys.

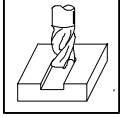


## CNC Milling Programming Aids

Selection

T	Current magazine configuration
H	Milling, Primary overview
D	DIN programming G commands and other commands
F	Overview, cutter radius compensation (CRC)
K	Contour segment string, Overview: G71, G72, G73
Z	Cycles overview: G31, G57, G75 to G88
M	Machine commands
[D]	G00 Rapid traverse
	G01 Linear interpolation
	G02 Circular interpolation, clockwise
	G03 Circular interpolation, counter-clockwise
	G04 Dwell
	G09 In-position programming
	G10 Polar coordinates, rapid traverse
	G11 Polar coordinates, linear interpolation
	G12 Circular interpolation polar coord., clockwise
	G13 Circular interpolation polar coord., counter-clockwise
	G22 Subroutine invocation
	G23 Program segment repeat
	G53 Cancel incremental datum shift
	G54 Set datum (global)
	G59 Incremental datum shift
	G90 Absolute dimensions
	G91 Incremental dimensions
[F]	FRK General notes on cutter radius compensation (CRC)
	G40 Overview: Canceling CRC
	G40-1 Cancel CRC without retract command
	G40G45 Cancel CRC, contour-parallel retract
	G40G46 Cancel CRC, semi-circular retract
	G40G47 Cancel CRC, quarter-circular retract
	G41 CRC overview, left of contour
	G41G01 CRC left of contour, no approach command
	G41G45 CRC left of contour, contour-parallel approach
	G41G46 CRC left of contour, semi-circular approach
	G41G47 CRC left of contour, quarter-circular approach
	G42 Overview, CRC left of contour
	G42G01 CRC right of contour, no approach condition
	G42G45 CRC right of contour, contour-parallel approach
	G42G46 CRC right of contour, semi-circular approach



 <b>CNC Milling Programming Aids</b>		
Selection		
	G42G47	CRC right of contour, quarter-circular approach
[Z]	S	1st and 2nd clearance level
	G77	Multiple cycle invocation on circular arc
	G78	Multiple cycle invocation on linear path
	G79	Single execution of last cycle
	G81	Drilling cycle
	G82	Drilling cycle with chip breaking
	G83	Deep drilling cycle
	G84	Thread cutting cycle
	G85	Reaming cycle
	G86	Hollowing cycle (boring out a drilled hole)
	G87	Rectangular pocket cycle
	G88	Circular pocket cycle
	G89	Pin cycle
[K]	ZA	Additional addresses for segment contour programming
	G71	Line (subgroup)
	G72	Circle (subgroup)
	G71G71	Line/line (subgroup)
	G71G72	Line/circle (subgroup)
	G72G71	Circle/line (subgroup)
	G72G72	Circle/circle (subgroup)
	VPZ	Four-point contour segment string consisting of different contour elements (subgroup)
	OK	Contour definitions with indeterminate end point (subgroup)
	P000	Tangential transition from previous element (subgroup)

**Note**

As in the CNC Turning Simulator, in defining the help names for the contour programming help functions, the NC words "G71", "G72", and "G73" were abbreviated with the numbers "1", "2", and "3", and the respective address characters were added. Two examples:

Contour definition (circular arc): G72 X.. I.. J..                      Help Name: 2XIJ

Contour definition:                      G71 A..  
(line/circle)                      G72 Y.. I.. J.. B..                      Help Name: 1A2YIJB

Please, note that the name assignments of help functions is fixed and the address sequence has to follow the rules.

**The addresses are presented in the order of: X, Y, I, J, L, A, B .**

Example: If a programming block reads "G72 I.. J.. Z..", the name of the corresponding Programming Aid is "2ZIJ".

Finally, in **four-point contour definitions** the circle centre coordinates I and J are grouped together with M.



**If you are uncertain regarding the location of Programming Aids for specific NC commands, please refer to the overviews.**

During the learning process of the program, it is helpful to scroll through the subgroups by pressing the **[F3]** or **[F4]** function keys.

Figure 11.1-4:  
CNC Turning,  
Programming Aid;  
Segment contour  
programming, circular  
arc, clockwise: the  
function keys at the  
bottom of the screen  
are labeled with the  
available help  
functions

**MTS** MTS CNC 5.1
Two-point string: arc
ZXIK
page 1/1

An arc can be programmed by its center coordinates I and K and the end point coordinate X. If the Z ax-parallel intersects the arc defined by X at points P<sub>1-1</sub> and P<sub>1-2</sub>, two solutions for Z arise. The desired contour is determined as to the arc criterion in the NC block by addresses P001 P002 (shorter/longer arc).

**G72 X.. I.. K.. (P070) P001 P002**

**Example:**  
 N030 G01 X+116 Z+95  
 N040 G72 X+60 I+160 K+40 P070 P001

1 Editor input
2
3 Next page
4 Prev. page
5 Directory
6 Select help
7
8 Return

Figure 11.1-5:  
CNC Milling,  
Programming Aid;  
cycle call on graduated  
arc "G77";  
as a rule each help  
screen is sectioned  
into explanatory text,  
graphic display and  
program example.

**MTS** MTS CNC 5.1
Cycle invocation on a divided circle
G77
page 1/1

The command G77 invokes the cycle which had been defined last for multiple execution. The cycles are executed in regular distances on a circular arc.

**G77 X... Y... A... B... D... S...**

- X,Y Arc center coordinates
- A Angle of the 1st cycle invocation position to the positive X axis
- B Arc radius
- D Angle between the single cycle invocations
- S Number of executions

Addresses A and S are optional.

1 Editor input
2
3 Next page
4 Prev. page
5 Directory
6 Select help
7
8 Return

## Using Help Functions

Once you have activated the help function and entered the name of the help screen, it appears displayed on the screen. You can call the Programming Aids one after the other; you can either scroll within one group, i.e., search for and compare closely related NC commands, or you can explore wider contextual scopes by 'jumping' from group to group.

During Interactive Programming mode or NC program creation with the Editor, it is also possible to display only a part of the programming level with the help display. This allows you to correct or enter the current NC block directly.

### Screen Display

The **screen display** is divided into three areas:

The screen header contains the description and the name of the help screen. Also, if a help function comprises several 'pages', this is indicated by a corresponding display in the header.

Below the header is the main screen area, containing the help information text, and the NC command structure and parameter description that are normally supplemented by a graphic display and a programming example.

The rectangular fields at the bottom edge of the screen represent the available **function keys**, and are labeled with their respective control functions for the current help function:

**F1**

**[Editor/Input]:** In addition to the current help function, you can also display and edit the current NC block by pressing **F1**. (This function is available during programming only.)

**F3**

or

**F4**

**Next or Previous Page:** You can use the **F3** and **F4** function keys to select the previous or next information within a group.

**F5**

**Table of Contents:** You can display the respective table of contents of the currently selected group (see also overview above) by pressing **F5**.

**F6**

**Second Help Screen:** Parallel to the display of one help function during a programming function, you can call a further help screen by pressing **F6**. At the prompt "**Help Screen**", enter **the name of the desired help screen** in the dialogue line, and confirm it by pressing **↵**.

### Note

If this function has been used already, the description used last time appears in the dialogue line as a proposal. You can now either confirm this function or type over its name.

**F8**

or

**ESC**

**Return:** Use the **F8** or **ESC** key to close the help function. You will then automatically return to the working situation from where you initially called the help function.

Figure 11.1-6:  
CNC Turning,  
Programming Aid;  
Correcting an NC  
program simultaneous  
with the help display:  
the current NC block is  
always displayed at the  
bottom of the screen,  
and can be edited as  
usual.

**MTS CNC 5.1**
**G81** page 1/1

Straight roughing cycle

The command G81 defines a cycle for cutting in longitudinal direction with an optional adjoining contour for internal as well as external machining. The cycle is invoked with G80 at the end of the contour description. Before cycle invocation, a finishing allowance can be programmed with <G57>.

G81 X... Z... I... (H... u. W... )

X,Z Coordinates of the contour starting point  
I Depth of cut in X direction, incremental  
H,W Reset to chip breaking

Example:  
 N030 G00 X100 Z5  
 N040 G81 X30 Z5 I5  
 N050 (Description of the contour)  
 ....  
 N080  
 N090 G80

1 Editor input
2
3 Next page
4 Prev. page
5 Directory
6 Select help
7
8 Return

Figure 11.1-7:  
CNC Milling,  
Correcting an NC  
program while  
displaying a  
Programming Aid.

**MTS CNC 5.1**
**G83** page 1/1

Deep drilling cycle with chip breaking

The command G83 defines a deep drilling cycle with chip breaking on the first plane.

G83 Z... W... A... B... D... K...

Z- Drilling, incremental to the 1st plane  
 W 2nd plane  
 A Dwell time for chip removal  
 B Dwell time on the drill bottom in seconds  
 D Degression (decrease of the drilling depth)  
 K 1st drilling depth


If W had been programmed, the tool must be positioned on the 2nd plane with cycle invocation by G77, G78 or G79.

1 Editor input
2
3 Next page
4 Prev. page
5 Directory
6 Select help
7
8 Return

## NC Programming with Help Functions

If you access a help function during the creation/editing of an NC program, you have the option to correct or re-generate the current NC block, i.e. to enter data while you are viewing the help information.




**Editor/Input:** Provided a help function was called during programming with the Editor, the Interactive Programming or the Teach-In mode, you can display the current NC block at the bottom screen edge by pressing the  function key (see figure). In addition, this NC block can now be modified or created anew by using the usual editing functions.



Use  to cancel editing of an NC block without saving any of your changes.



To insert the (edited) NC block into your NC program, confirm it by pressing .

### Note

Especially when entering or checking complex NC commands, this simultaneous access to both Programming Aids and the NC program is very convenient and saves time as time-consuming searches of programming instructions are not necessary.

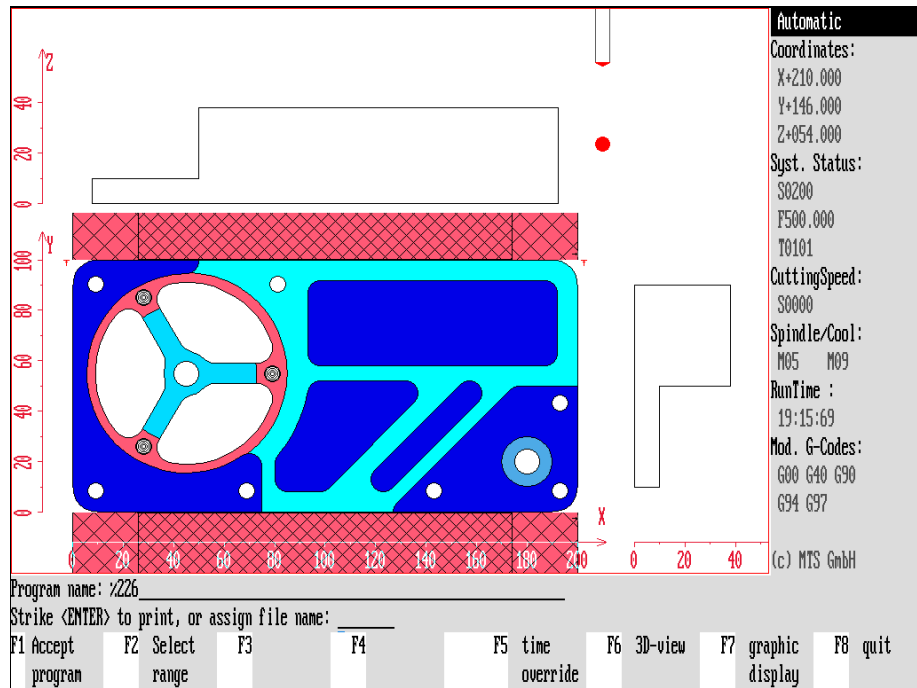
However, if it is preferred to **block the help function**, e.g., for the purpose of test preparation of the CNC trainees, the simplest method is to rename the subdirectory in which the help functions are located. This prevents the help functions from being accessible.

### Error Message

Should no corresponding help screen be available with the name entered, the message "**Help screen not found**" is displayed. It is therefore essential that you pay particular attention to the proper spelling of help screen names.

Furthermore, it is the possibility that an incorrect 'search path' was entered for the corresponding files during **configuration**. In the CNC Simulator, the default subdirectories for the programming help files are `.\DHILF` (turning), and `.\FHILF` (milling), respectively. If these files are located in a different directory, this directory, together with the proper filename extensions, must be specified during the configuration of the CNC control (see also, Configuration Instructions, Control Configuration).

Figure 11.2-1:  
CNC Milling,  
Main Menu,  
Print Function;  
Screen contents  
(‘screen dump’) can be  
printed out or saved as  
a file.



## 11.2 Printing Graphics

The program management of the Editor has an integrated print function, enabling you to output NC programs directly with a printer (see also Chapter ?? in this manual). In addition, the CNC Simulator is capable of printing a hardcopy of almost all editing situations displayed on the screen (screen dump). You need to have a compatible printer, of course, and the printer port needs to be set up correctly during the configuration of the CNC Simulator (see also Configuration Instructions).

### Selection:



A screen dump can be printed in both Setup and Automatic Mode. However, in the latter case the machining simulation must first be interrupted by pressing **F8**; moreover, immediate access to the print function is not available during a program dialogue. Conclude the dialogue first, and then access the print function.

On the PC keyboard, the **shift** + <PRINT SCREEN> key combination is used to access the print function. In response, the following message appears in the dialogue line:

**"For printout, press **←**, or type filename: \_\_\_\_\_"**

### [Enter Filename]

If you wish to save the current screen display in the form of a file (i.e., for printout at a later date), enter a filename according to the conventions of the MS-DOS operating system. This filename automatically receives the ".PRN" extension, and is saved in the main directory in which the CNC Simulator is installed.



Use **ESC** to cancel this program function.



Press **←** to confirm the entered filename; should you not have entered any filename, the screen dump is automatically output on the printer.

Before the printout is started, you will be requested through a dialogue to specify the size of the screen area and also the colour or gray scale output parameters for the printout.

### Note

To print a screen dump, the connected printer must be capable of operating in graphics mode. The MTS systems support CANON and EPSON printers, and all printers that are EPSON compatible, as well as all HP printers for which an integrated printer driver is supplied with the program. If some other printer is used, a corresponding peripheral printer driver must be supplied (see also Configuration Instructions, Hardware).

Figure 11.2-2: CNC Milling, Main Menu, Print Function; prior to printing the screen dump, you use the cutout frame to specify the desired screen area.

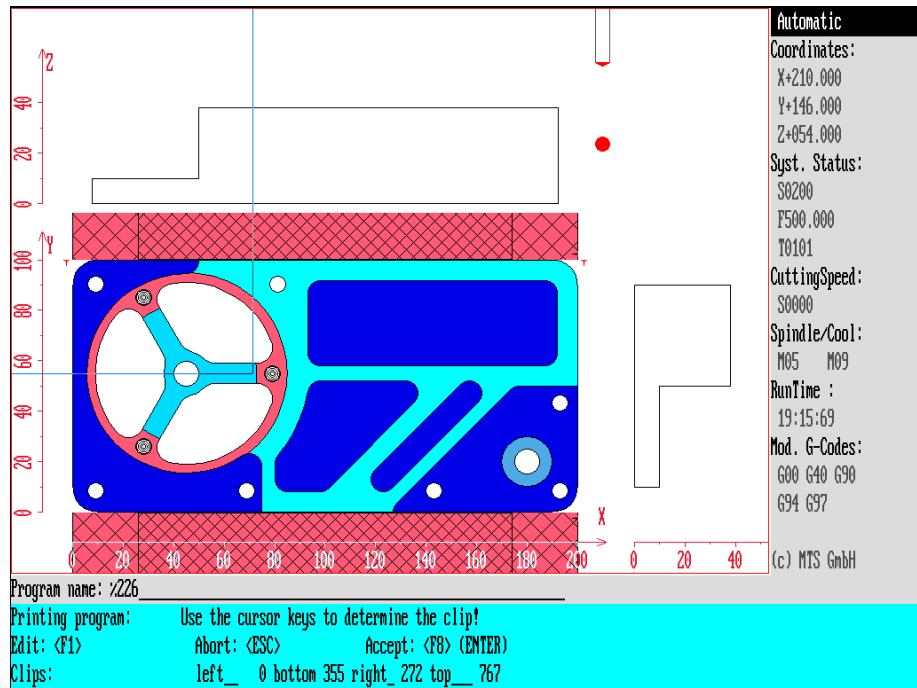
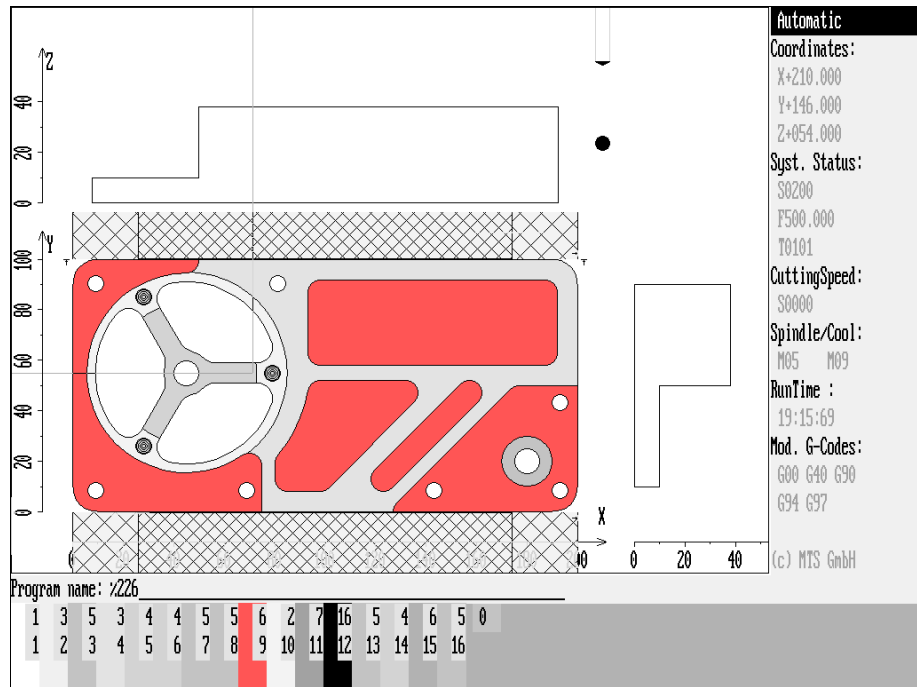


Figure 11.2-2: CNC Milling, Main Menu, Print Function; after specifying the screen cutout you can use menu parameters to determine the colour or gray scale reproduction for the printout. In terms of available gray scales and their assignment to the displayed objects, the printout is identical with the screen display.





## Selecting the Screen Area for Printouts

The dialogue field and other information for defining printouts are displayed at the bottom of the screen.

As a first step, you specify the screen area to be printed. A selection frame is inserted in the screen display indicating the size and screen area of the desired printout. In the field "Cutout borders: top left ... bottom right" are also the pixel coordinate values of the selected screen segment displayed.

**F1**



or



**F2**



or



Press **F1** to change the size and position of the screen cutout selection frame:

Begin by using the cursor keys to place the top left corner of the cutout frame at the desired position.

Continue by pressing **F2** to define the position of the bottom right corner of the cutout frame.

Use the cursor keys to move the corner to the desired position.

Confirm the selected screen section by pressing the key.

This is followed by a further dialogue for defining the colour/gray scale parameters for the printout:

## Defining Colour/Gray Scale Parameters

The definition of colour/grayscale print parameters is handled by a further dialogue function that provides information on the operational steps and default parameters for the printout:

**F1**

Press **F1** to open the printout parameter menu.

The menu consists of 16 dual input fields at the bottom of the screen, with each field containing two numbers: the upper field indicates the current colour selection, while the lower field refers to the various screen objects. Beginning from the left, for example, the first field is used to specify the colour of the work space, the second that of the information column, the third that of the displayed coordinate axes, with the ninth field setting the fill colour for the workpiece, etc:

**F2**

Use the **F2** key to toggle back and forth between the colour display and grayscale representation.



or



Press the or cursor keys to activate the fields representing the various screen objects.



or



Press the or cursor keys to select the desired grayscale for the selected object.



Press the key to confirm a selected grayscale.

This function initiates either the generation of the graphics file, or the printout of the selected screen area, with subsequent automatic return to the starting situation.

### Note

You can print a graphics file later on with the MS-DOS command:

**COPY [FILENAME.PRN] >LPTx,**

where LPTx is the port the printer is connected with.



## 12. Workshop-Oriented Programming Introduction

### Contour Definition Commands

According to the DIN Standard 66025, a workpart contour is programmed with the commands "G01" (linear interpolation), "G02" (circular interpolation, clockwise) or "G03" (circular interpolation, counter-clockwise), and by defining the coordinates of the target point or center of a circle, respectively. Considering this, the definition of contours which cannot be measured according to NC standards might require extensive calculation. Modern CNC control systems as well as the CNC Simulator feature additional commands for segment contour programming allowing the integration of data with reference to length, angles and radii, as well as tangential transition conditions. Consequently it is possible to program many contours without additional calculations.

The need to program or modify NC programs directly in the workshop constitutes another problem. On one hand, there are normally no tools for geometric calculations available at the shop level. On the other hand, entering the contour definition commands for complex geometric shapes requires good abstract thinking from the operator. According to experience even Programming Aids and related information are often insufficient when it comes to mastering this difficult subject matter at the first try.

### WOP Interface

To solve these problems, a programming method was developed to meet especially the requirements in production level: the "Workshop Oriented Programming", abbreviated "WOP". WOP stands for a programming interface for CNC technologies worked out (even if not standardized) by the German Federal Ministry of Research and Technology within the framework of one of its projects. With this method, data entry takes place with function keys only, and with the help of a program dialogue. Parallel to the data entry, the workpart geometry is graphically displayed in a fashion similar to a CAD program and, if several solutions are possible, all solution options are presented for selection.

### WOP Features

The WOP interface developed by MTS GmbH is based on these conditions. It is easy to use and convenient in everyday NC programming:

- It can be easily incorporated in the CNC Simulator, facilitating easy switching **from one programming mode to another** or enabling a program test. In this manner, the individual system components can be used to their full advantage.
- During NC programming the **workpart contour and the corresponding NC block** are simultaneously displayed on the screen. This means that it is possible to compare the original drawing and the assignment of the graphical contour display with the generated NC block parallel to each other. Possible errors in contour definition are recognized immediately and may be corrected at once. During the geometrical definition, the direct correlation between the workpart geometry and the NC blocks remains apparent.
- On the other hand, all already programmed<sup>1</sup> contours can be copied to the WOP Interface for control and modification.
- The MTS programming code does not require a complete definition of all geometrical elements; it does permit the programming of open contours and multiple-point definitions.

<sup>1</sup> providing they are available in the form of NC programs in the MTS format; it is irrelevant whether they were created by means of the CNC Simulator, or the INCAD or another NC programming system.

Figure 12-1:  
Schematic diagram:  
Combining the  
Workshop-Oriented  
Programming with the  
other CNC Simulator  
programming options.

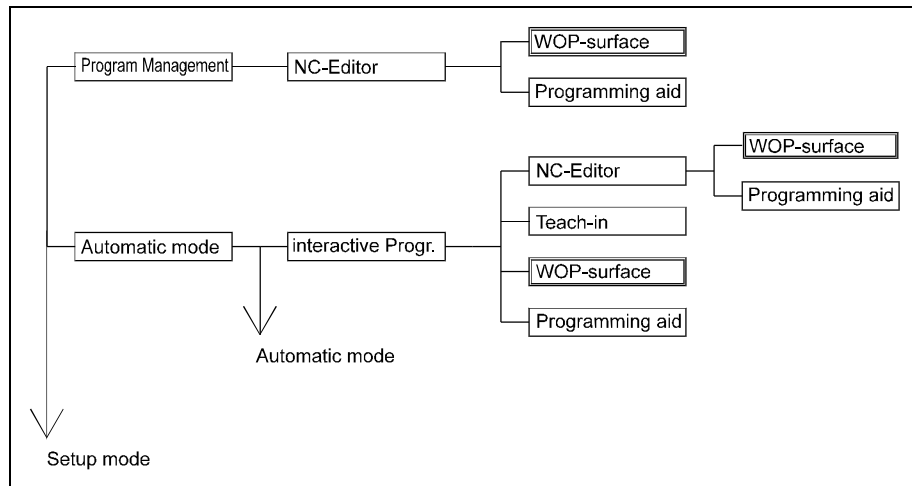
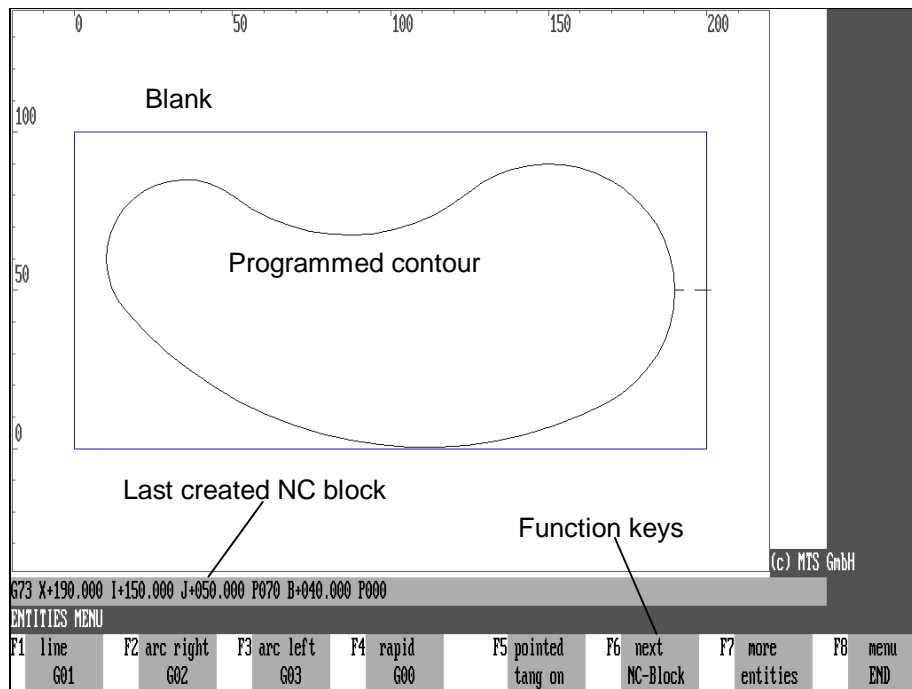


Figure 12-2:  
WOP Interface, CNC  
Milling,  
Main Menu:  
Center: Display of  
programmed workpart  
contour.  
Bottom: Function keys  
and last current NC  
block;



- The WOP Interface **dialogue guidance** automatically provides only the acceptable parameters for selection, including the data relating to radiusing or chamfers. Intersections and transitions are calculated automatically to facilitate the application by the user.
- Like in other CNC Simulator program modules, the **Measuring** and **Zoom** functions are available. This provides the option of using the screen segment zooming function while generating or modifying NC blocks.

Consequently, the programmer has immediately the confirmation that the workpart geometry is specified correctly and that the workpart can be machined as specified on a CNC machine tool. Finally, schematic help graphics are available clarifying the respective input parameters.

#### Note

**Since the WOP Interface is not a part of the CNC Simulator standard program package, it is suggested that you verify that this extension option is actually included in your system.** If this is not the case, you can skip over the rest of this chapter.

## Starting the WOP Interface

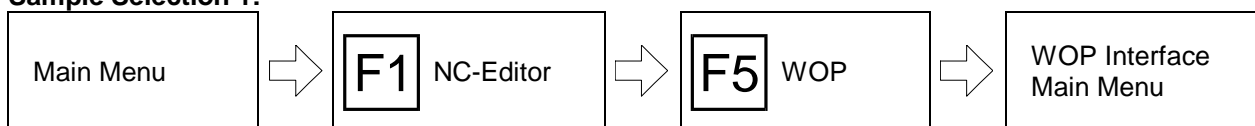
### Installation

If the WOP Interface was supplied as a part of your CNC Simulator version, it was automatically converted with the other program modules. Accordingly, no separate installation procedure is required.

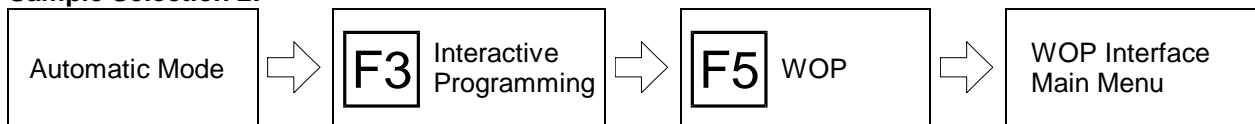
For both technologies: turning and milling, the WOP Interface is structured analogously with similar operation steps. A special introduction is needed only about the differences in NC programming (see also the Programming Instructions ["Programmer's Guide"]; the differences refer normally to the coordinate systems). If you know how to work with one of these technologies you are able to work with the other as well.

For this reason, the Workshop-Oriented Programming methods are documented here as being identical. Like the other chapters of this manual, minor differences are highlighted and described in detail.

#### Sample Selection 1:



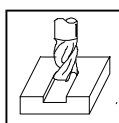
#### Sample Selection 2:



WOP

The WOP can be started while using the programming interface of the Editor, or while creating an NC block during Interactive Programming, by pressing the **F5** function key. This loads the WOP Main Menu, allowing you to modify the current NC block, insert new NC blocks into the program, or to copy existing NC blocks to have them checked and corrected if necessary.

#### CNC Milling



The Workshop-Oriented Programming is designed only for the compatibility with the MTS programming code with the "G17" plane selection. Before you start working with the WOP Interface, verify that the appropriate defaults have been set (see also Configuration Instructions).

Figure 12-3:  
WOP Interface, CNC  
Turning;  
*Bottom:* Function keys,  
information and  
dialogue bar,  
displaying the current  
NC block.  
*Right:* Input fields for  
address values;  
*Center:* Graphic  
display of  
blank/workpart,  
programmed contour  
plus selectable  
supplementary help  
screen.

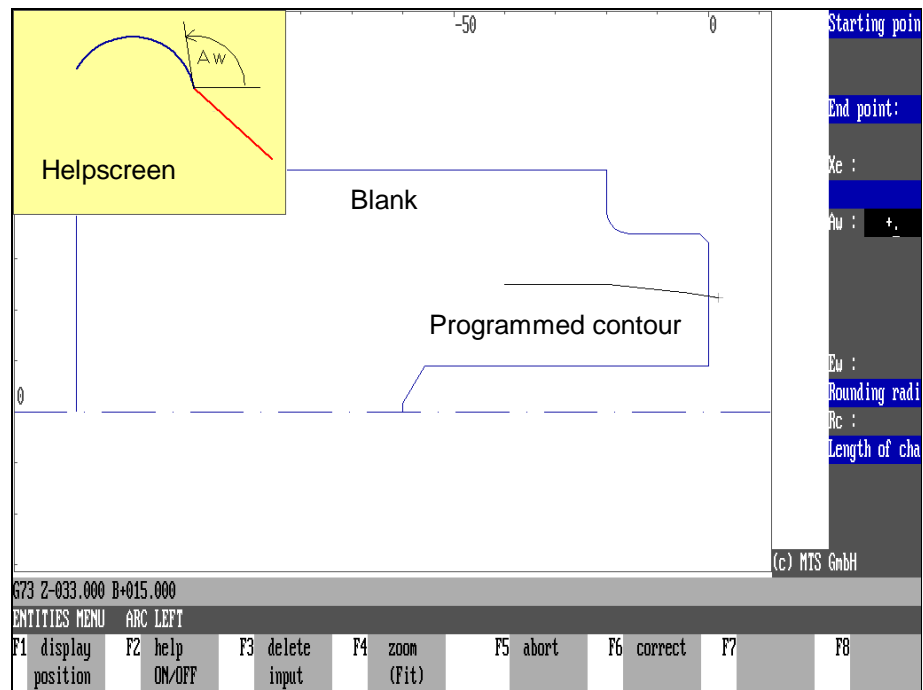
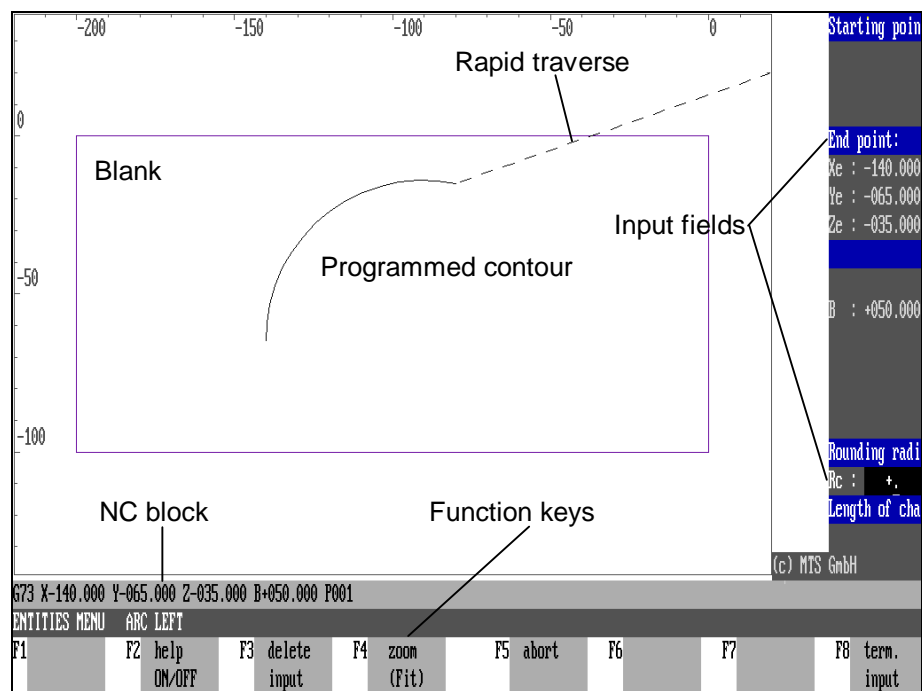


Figure 12-4:  
WOP Interface, CNC  
Milling;  
use of "G73"  
command for  
programming a circular  
arc with different Z  
coordinate points for  
end and start. The  
WOP display  
illustrates the  
projection onto  
machine table plane  
X/Y, using G17.



## WOP Interface Screen Layout

As already described, the Workshop-Oriented Programming has an interface of its own with several menus and its own dialogue guidance. Its screen display is divided into three main areas:

### Function keys

1. The numbered boxes at the bottom of the screen represent the program functions available during operation. They can be called with the function keys. Depending on the current processing, a choice of various functions is available.

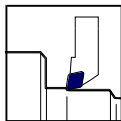
The two text lines located above the function keys are reserved as information and dialogue lines. The upper line displays the current NC block or specific error messages; for better clarity, the selected processing function is displayed in the lower line.

### Input fields

2. The **input fields** for entering the address values are located on the right side of the screen. Initially, the input fields display all the **valid parameters** relating to the selected geometrical element, and you can then enter the values according to the information in the drawing. In addition, the entry of certain data generates an **automatic evaluation** regarding the relevance of the **remaining addresses** to the programming of the contour. To eliminate the generation of incorrect NC blocks, all addresses resulting in "overdefinitions" are automatically removed.

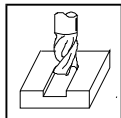
3. Based on the address data, the workpart **blank** and the corresponding **contour** are displayed in the central area of the screen:

### CNC Turning



The contour of the blank or workpart to be turned is displayed in the form of a sectioned area on the X/Z plane; the rotation axis is represented by the dot-and-dash line.

### CNC Milling



The Milling WOP Interface displays the workpart contours as top views (projection on the machine table plane). This projection mode is identical with the conventional projection used in drawings, and depends on the machine type being simulated by the CNC Simulator.

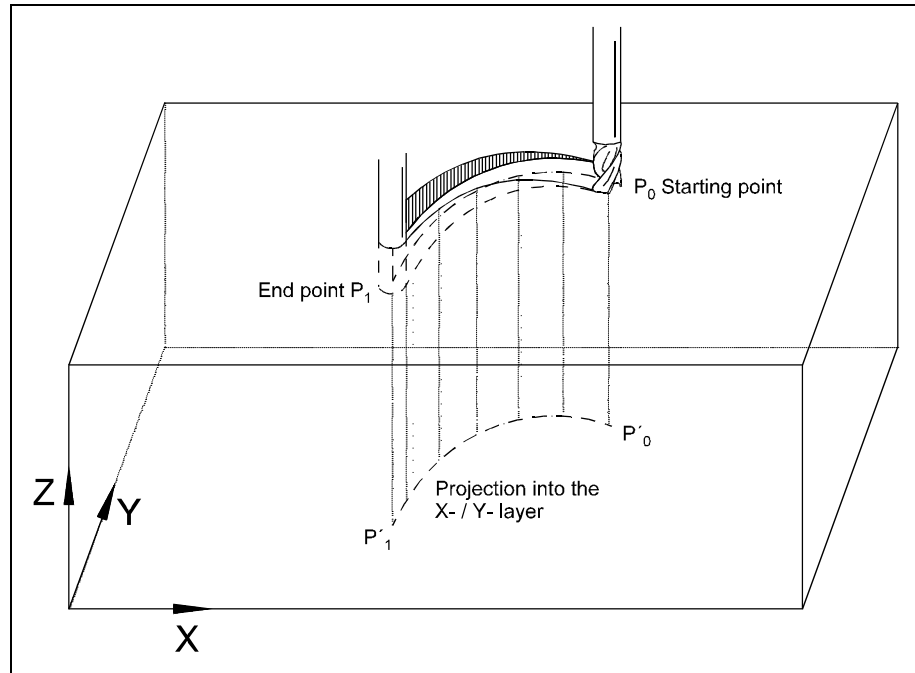
Since the configuration is based on a 3-axis milling machine with a 2 1/2 D control, it is possible to move spatially along the three coordinate axes X, Y and Z; however a circular arc motion in the X/Y plane (based upon G17) can only be effected by superimposing a linear movement in Z direction (helical interpolation). This means that the contour projection onto the machine table plane always consists of linear or circular interpolations. These are precisely rendered by the WOP Interface, and can be programmed without problems. (This sounds more complicated than it really is; so just try to use the WOP?)

When programming an NC block, the corresponding contour is graphically displayed. Successive contour elements are connected to form a contour segment string. This means that at any time during programming, you are able to see at the same time the address data, the NC block and the graphic contour display. If you have specified a **zoom** enlargement, only the contour of the enlarged segment is displayed. Moreover, during the definition of geometrical values, you can access additional **help graphics** to illustrate the selected parameter.

### Note

You also have the option to change the color used in the representation of the various screen elements. The configuration of the color setting is an integral part of the Measuring program module (see [Chapters 9 and 10](#)).

Figure 12-5:  
CNC Milling,  
3-D machining  
schematic (helical  
interpolation).





## Available User Options

You can use the Workshop-Oriented Programming Interface to define traverse paths and contour trains in accordance with the preparatory functions "G00", "G01", "G02" and "G03" of the DIN 66025 standard, or by using the contour definition commands "G71", "G72" and "G73" as specified in the MTS programming code.

### NC Programming

Starting from the WOP Main Menu, the NC programming function is located under the **Create Geometry** menu option.

In the subsequent **Entity menu** each contour element is initially selected as a "linear" or "circular" path.

After this the corresponding **Input Menu** is loaded allowing you to enter the relevant address values: not the NC command, but the geometrical data of the drawing are prompted; this may be data referring to the length, angles, coordinates or tangential transition paths as well as chamfer lengths or radiusing. In addition, during address specification a context-based function checks which additional addresses are acceptable.

Example: For programming a circular arc with "G72" or "G73" the end coordinate and radius data are sufficient; if these values are available, the addresses "I" and "J" (center point coordinates of the circular arc) are no longer displayed.

When a contour element has been correctly defined, it is displayed as a graphical representation. If several (mathematical) solutions are possible, they are all presented for selection.

A specific feature of the WOP Interface is the unequivocal (and reversible) assignment of the contour definition steps to the corresponding DIN commands. This allows easy switching from one programming mode to another irrespective of eventual differences of NC syntax.

Due to the user dialogues and their graphic support functions, NC programming can be quickly learned with the WOP Interface. The program functions are described in detail in the subsequent sections:

- Menu levels:           Main Menu  
                          Entity menu  
                          Input Menu
- Programming a contour element
- Modifying a contour
- Segment contour programming using non-NC compatible data from drawings:
  - Multiple-point definitions and open contours



The **illustrations** in this manual are monochrome. Accordingly, certain reproductions of screen representations with color differences cannot be rendered in their full effect.

Figure 12.1-1:  
WOP Interface,  
schematic diagram of  
Main Menu and its  
integration with the  
CNC Simulator.

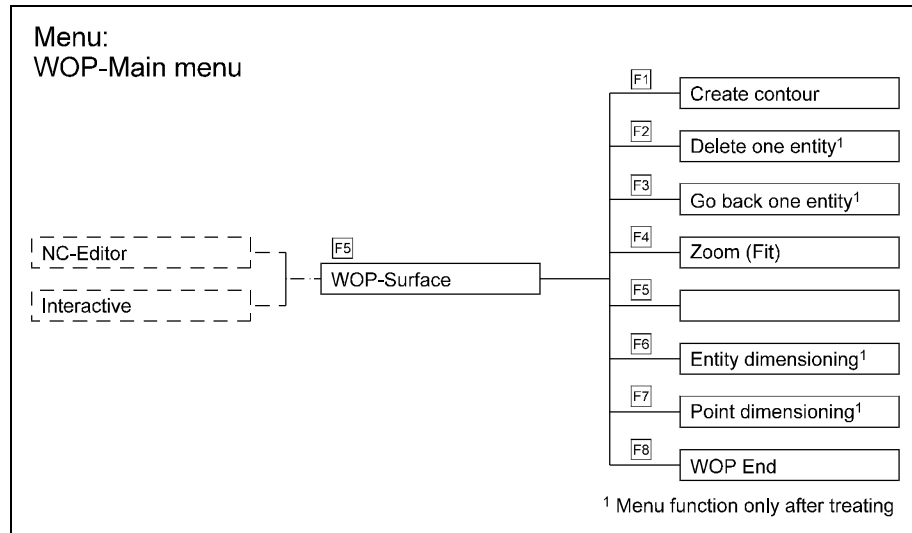


Figure 12.1-2:  
WOP Interface, CNC  
Milling, Main Menu,  
Variant 1:  
When starting the  
WOP, the Main Menu  
is opened, containing  
these options.

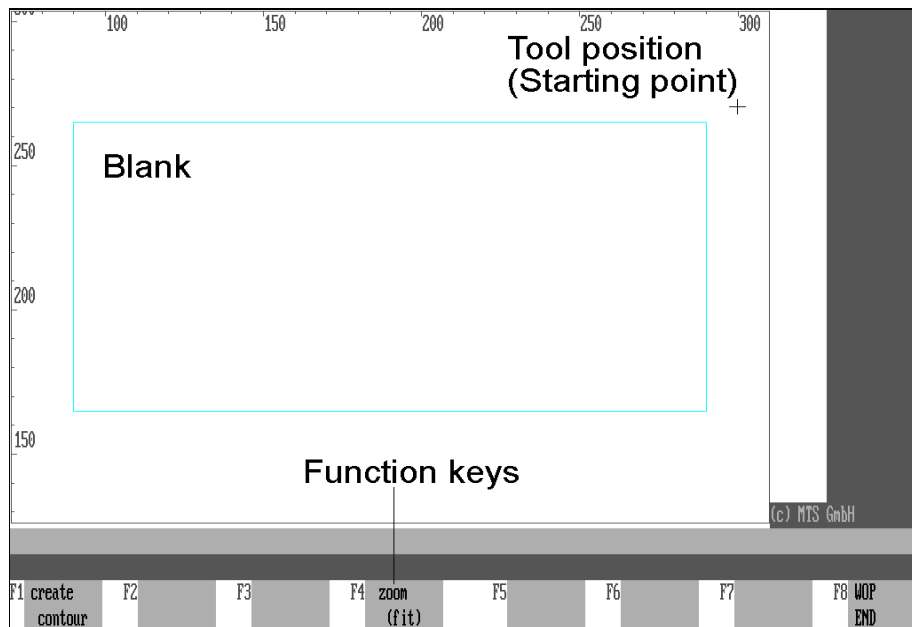
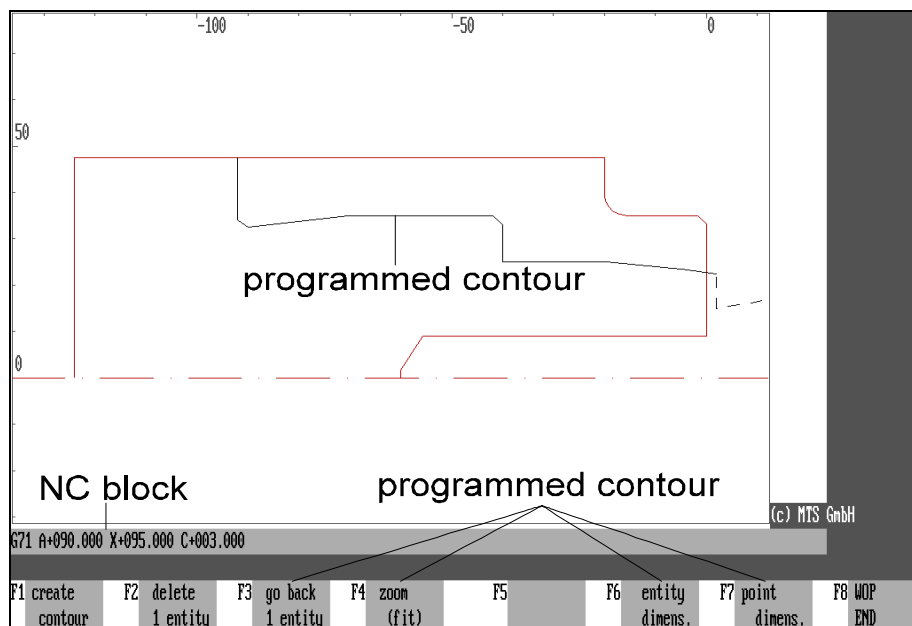


Figure 12.1-3:  
WOP Interface, CNC  
Turning, Main Menu, Variant 2:  
After completing  
processing, on  
returning to the Main  
Menu further menu  
options are available.



## 12.1 Menu Levels

### WOP Sequence Control

As a rule, NC program editing in the Workshop-Oriented Programming mode is done with a series of user dialogues. Hereby, a difference is made between the WOP sequence control for selecting the desired machining techniques, and the contour definition (entering address values). The WOP sequence control refers to such versatile steps as the selection of a geometrical element for the next programming step, toggling the help graphic display on and off, as well as enlarging a contour area, etc.

For each individual processing situation, the possible control sequences are listed in the "menu line", and they are selected with the function keys. Since they partly depend on the previous steps, different processing situations may result in a varying number of available menu options on the same menu level: For example, after a G01 command all geometrical elements required for segment contour programming are available. However, if the last NC block contains the programming of a chamfer as a contour transition, the programming code requires a linear string as a connecting element. This means that in such a situation the choice is in fact limited to that of a linear path.

### Main Menu

WOP

After the Workshop-Oriented Programming Interface has been loaded by pressing the **F5** function key - either from the programming interface of the Editor or from the Interactive Programming Mode - the Main Menu is loaded (see figure on facing page).

In all processing situations the Main Menu presents three default menu options:

**F1**

**Create Geometry:** You select this function to generate or modify an NC block. Subsequently an Entity menu is loaded (see below).

**F4**

**Zoom (Fit):** Use the **F4** function key to enlarge a selected detail of the contour display, or to toggle back to the full view.

**F8**

**End WOP:** Press **F8** to conclude working with the Workshop-Oriented Programming, and to return to the position from which you initially started the WOP Interface.

The Main Menu contains additional program functions which are available if editing or programming functions have been selected:

**F2**

**Delete One Entity:** You can use this function to delete the last programmed or copied NC block.

**F4**

**Reset:** If you wish to change again the NC programming based on one of the available geometrical elements, you can press **F4** to reset the editing progress.

**F6**

**Entity Dimensioning:** Entity dimensioning is accessed by pressing the **F6** function key (see also the chapter on Measuring).

**F7**

**Point Dimensioning:** Point dimensioning is accessed by pressing the **F7** function key (see also the chapter on Measuring).

Figure 12.1-4:  
WOP Interface, Entity  
Menus 1 and 2, menu  
structure schematic.

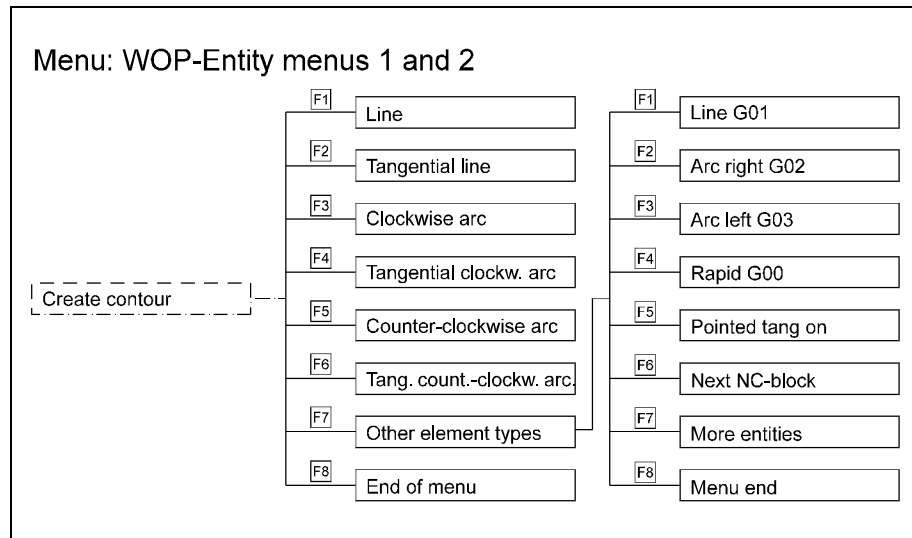
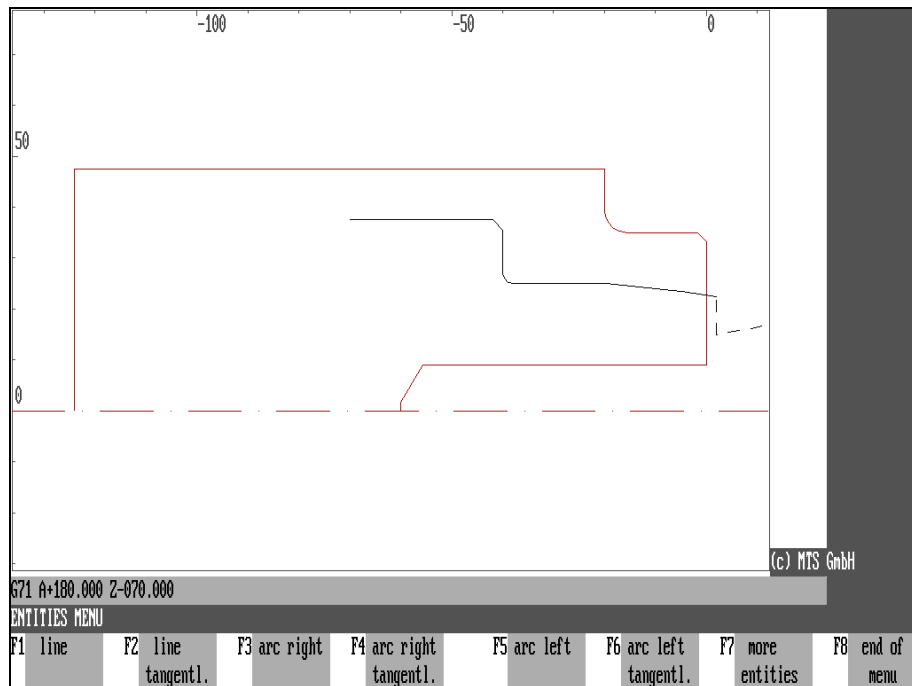


Figure 12.1-5:  
WOP Interface, CNC  
Turning,  
Entity Menu 1;  
illustration shows the  
menu with all functions  
for selecting a linear or  
circular path,  
according to segment  
contour programming.  
Use **F7** to switch to  
Entity Menu 2.



## Entity Menu 1

Immediately following the Main Menu, this menu allows the selection of the basic geometric entities (contour elements) to be programmed by contour definition commands (see also Programming Instructions). It should be remembered, however, that the selectable elements depend on the preceding string of contour entities. For example, if a rounding was defined as for the transition in the last NC block, this makes sense only if the segment transition does not take place tangentially; correspondingly a "tangential transitions" option is not available in this case.

F1

**Line:** The next element is a linear path that is programmed with the contour definition command "G71".

F2

**Tangential Line:** The next element is a tangentially connected linear path that is programmed with the contour definition command "G71 ... P000".

F3

**Clockwise Arc:** The next element is a clockwise circular arc (right-hand turning) that is programmed with the contour definition command "G72".

F4

**Tangential Clockwise Arc:** The next element is a tangentially connecting clockwise arc (right-hand turning) that is programmed with the contour definition command "G72 ... P000".

F5

**Counter-clockwise Arc:** The next element is a counter-clockwise arc (left-hand turning) that is programmed with the contour definition command "G73".

F6

**Tangential Counter-clockwise Arc:** The next element is a counter-clockwise arc (left-hand turning) that is programmed with the contour definition command "G73 ... P000".

F7

**Other Element Types:** The Entity menu 1 contains only those contour elements that are programmed by means of contour definition commands. If instead you wish to use the DIN command set "G00" to "G03" for defining a geometry, you can press the F7 function key to switch to the Entity menu 2.

F8

**End of Menu:** Press the F8 function key to return to the Main Menu.

Figure 12.1-6:  
WOP Interface, CNC  
Milling, Entity menu 2:  
The illustration shows  
the menu including all  
DIN 66025 standard  
programming functions  
of a linear path or  
circular arc. [F7] is  
used to switch over to  
Entity menu 1.

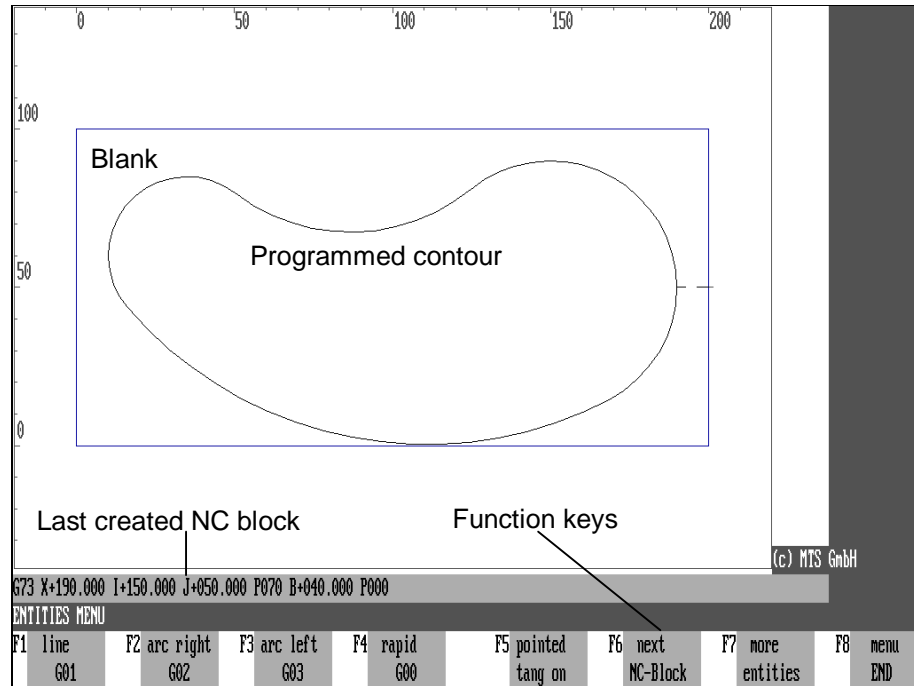
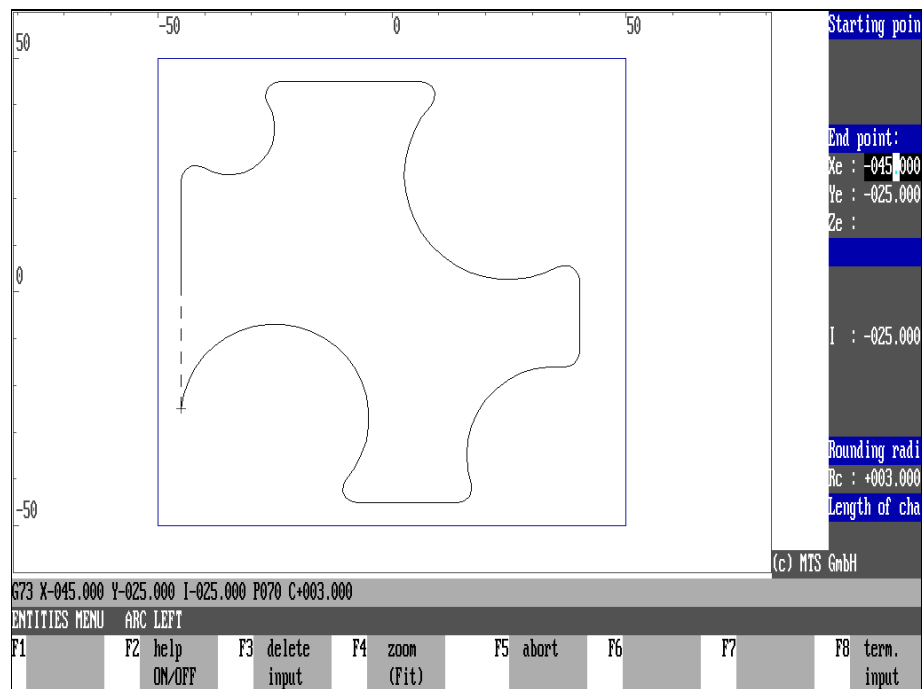


Figure 12.1-7:  
WOP Interface, CNC  
Milling:  
Programming a string  
of contours containing  
reverse tangential  
transitions ("acute  
tangents").



## Entity menu 2

Since the Entity Menu 2 is located on the same level as the Entity Menu 1, it is possible to change from one to the other at any time. It contains the geometrical entities corresponding to the NC commands "G00", "G01", "G02" and "G03" (see also, Programming Instructions). As in Entity Menu 1, the entities available depend on the context;; their availability is based on the preceding string of contour elements (see above).

F1

**Linear Path G01:** The next element is a linear path that is programmed with the NC command "G01".

F2

**Clockwise Arc G02:** The next element is a clockwise arc (right-hand turning) that is programmed with the NC command "G02".

F3

**Counter-clockwise Arc G03:** The next element is a counter-clockwise arc (left-hand turning) that is programmed with the NC command "G03".

F4

**Rapid Traverse G00:** The DIN command "G00" is used to program rapid traverse infeed adjustments.

F5

**Acute Tangential Angles ON/OFF:** This toggle switches on and off the option of programming a string of contour elements with a tangential transition, in which the successive element forms a "reverse" connection (with a seemingly sharp angle between the elements) with the previous element (see figure and Programming Instructions). The function key label changes from ON to OFF.



In this case the labeling **Acute Tangents ON** means that this programming option is currently not activated, while the display **Acute Tangents OFF** indicates that this option is currently activated.

F6

**Next NC Block:** If you start the WOP Interface from the Interactive Programming Mode, and intend to edit an existing NC program, you can use the **F6** function key to copy the subsequent NC blocks from the program into the WOP Interface for further editing.

F7

**Other Types:** Press the **F7** function key to toggle to Entity Menu 1.

F8

**End Menu:** Press the **F8** function key to return to the Main Menu.

Figure 12.1-8: WOP Interface, CNC Milling, Input Menu for defining geometry: Input fields for address value data are located at right; the help graphic always corresponds to the current input field at right.

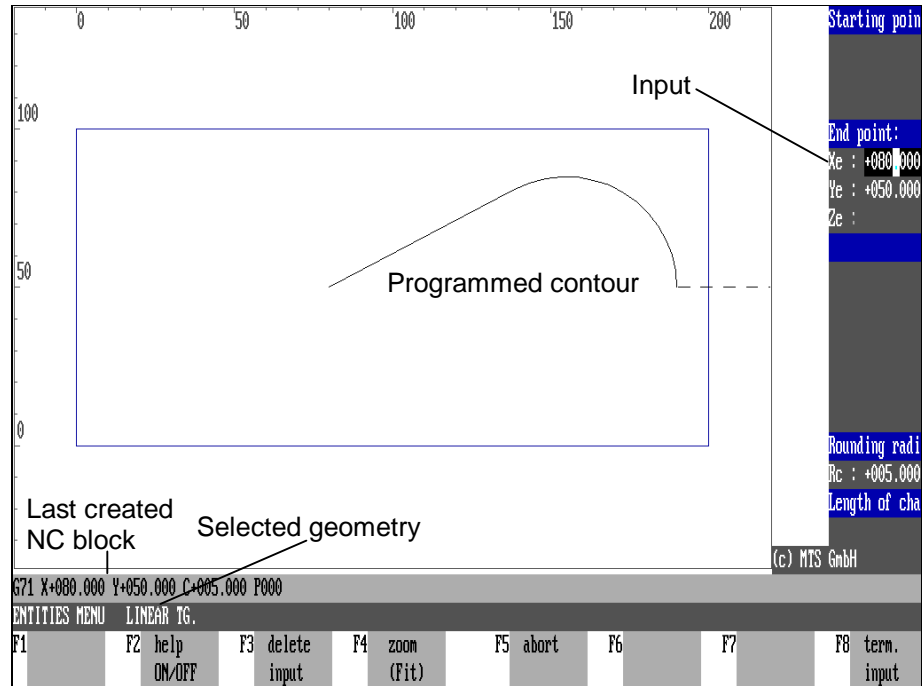
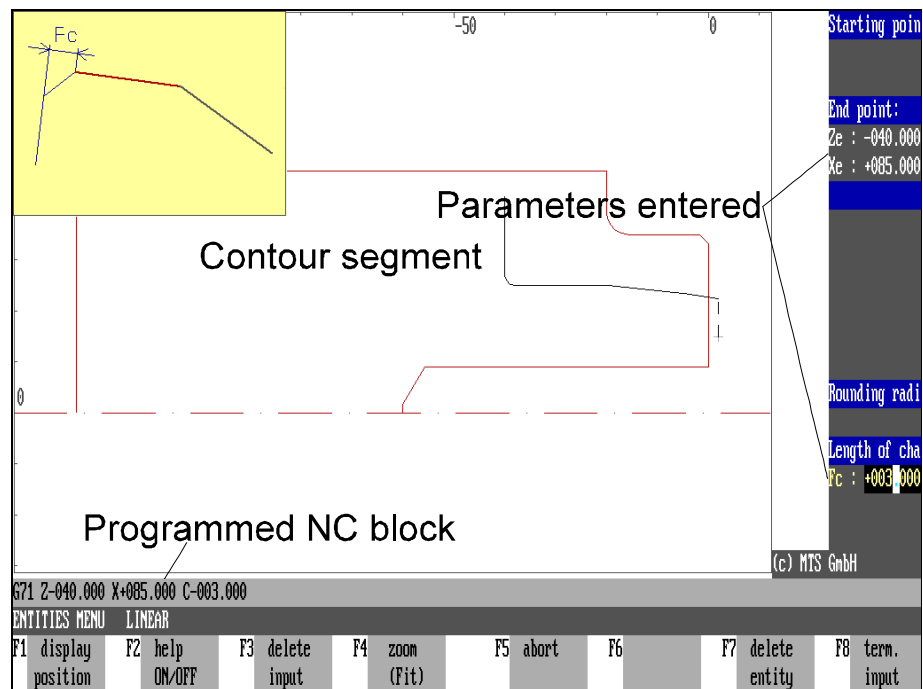


Figure 12.1-9: WOP Interface, CNC Turning, Input Menu; as soon as the string of contour elements can be calculated, it is graphically displayed. The NC block can be confirmed by pressing the Accept Entry function key, or modified with the Correction function key. You can enter a "chamfer" as "fine (detail) contour".





## Input Menu for Geometry Definition

The Input Menu comes after the Entity Menus. The input fields for the address values are located at the right side of the screen. Even if all values have not been defined, a contour is graphically displayed as soon as its calculation is possible. For example, when programming a linear path, if the starting angle is defined but the end point is so far unknown, a "help line" in the form of a half line can be displayed. As in the Entity Menus, **various context-dependent program functions** are available:

**F1**

**Position Help Screen:** To avoid covering the contour display with the help screen, it can be positioned with the **F1** function key. This menu option is available only if the help screen has been activated.

**F2**

**Help Screen ON/OFF:** The help screen can be toggled on and off with the **F2** function key. No help screen is provided for the preparatory function "00".

**F3**

**Delete Entry:** To delete the value displayed in the current entry field, press the **F3** function key.

**F4**

**Zoom (Fit):** It is possible to enlarge a screen cutout during the input of NC blocks or, to return to viewing the full display again (see also the chapter on Zooming) after viewing the enlargement.

**F5**

**Cancel:** If you cancel a contour definition, all entries in this menu are deleted, and you will automatically return to the previous Entity menu.

**F6**

**Correction:** During normal data entry, all input fields containing data are skipped. If an already entered value needs to be changed, the correction mode needs to be activated by pressing the **F6** function key. The "Correction" label is correspondingly deleted from the respective function key.

**F7**

**Delete Element:** Similar to the Correction function, this function key is activated only under special conditions. For example, if you are copying NC blocks from an existing NC program, they can also be modified by means of this menu. However, if the complete NC block is to be deleted, press the **F7** key.

**F8**

**End Input:** If the entered data stand for a contour, the entry of the last data item causes automatic acceptance of the generated NC block. If this is not the case (i.e. in Correction mode, or while the geometric definition of the last contour element remains incomplete, as in open-ended contour definition programming), the End Input option is made available, and you conclude editing by pressing the **F8** function key.

Figure 12.2-1:  
WOP Interface, CNC  
Turning, geometry  
definition Input Menu:  
Programming a  
counter-clockwise  
circular arc (left-hand  
turning), G73

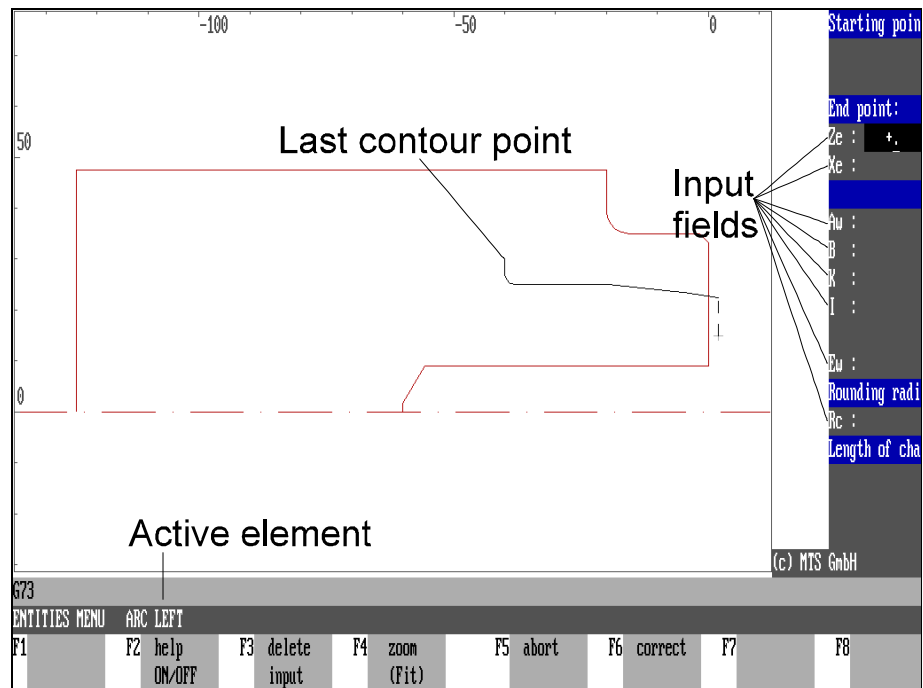
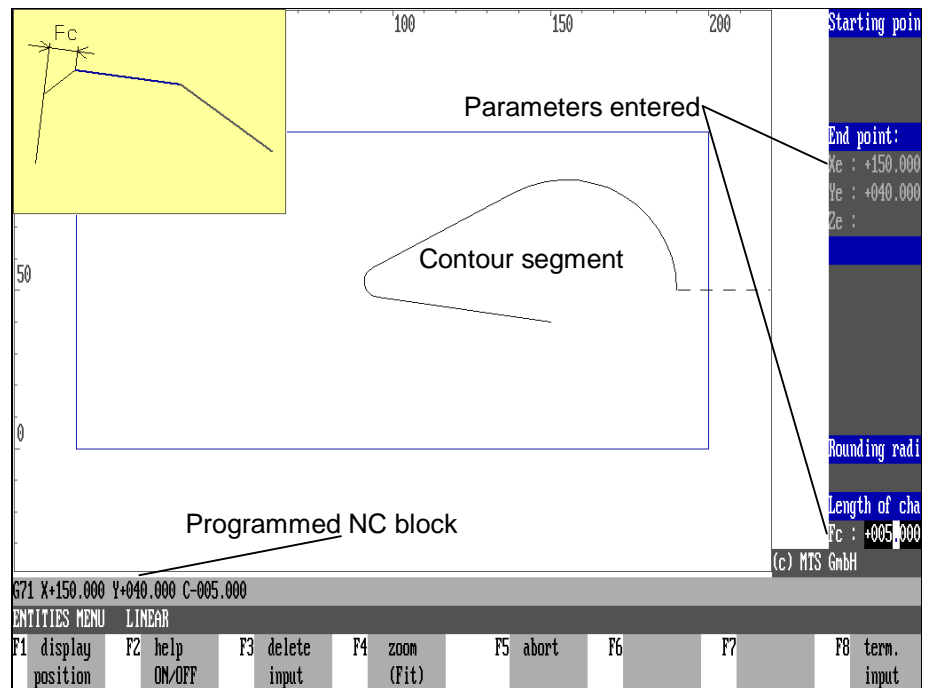


Figure 12.2-2:  
WOP Interface, CNC  
Milling, geometry  
definition Input Menu:  
Input Menu for contour  
definition of a linear  
path; as soon as  
sufficient data for  
calculating the contour  
is available, it is  
graphically displayed.  
The linear path ends in  
a 5 mm chamfer.

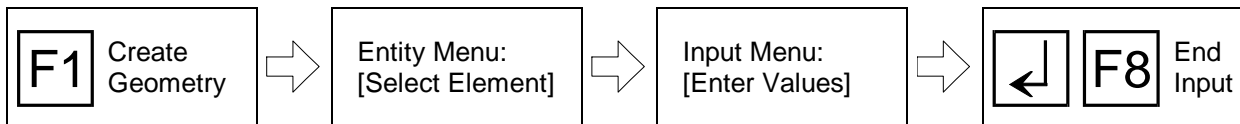


## 2. Programming a Contour Element

### Geometry Entities

The term "**workpart contours**" stands for one or several related axis departure commands that are executed as a result of an NC program. They consist of the geometrical entities of "line" and "circular arc", which can also be subject to certain additional transition conditions, such as a tangential connection, chamfer or fillet.

### Selection:



### Operation Sequence

In the WOP Interface, the NC programming function is loaded by selecting the Create Geometry option with the **F1** function key. Subsequently the desired contour element is selected in one of the Entity Menus, and the available values for the respective addresses are entered in the following Input Menu. This process is accompanied by a context-dependent user dialogue: only the menu options and input fields permissible under the current programming code are available. The following sections begin with the introduction of the editing functions, and continue with a discussion of the addresses possible with each geometrical element, as well as of alternative selections.

## Editing Functions in Input Menus

### Contour Definition

Every geometrical entity has an Input Menu of its own. After the known parameter values have been entered in the respective input fields, and sufficient data is available for calculating the contour segment, it is graphically displayed. At the same time, "open contours" (contour entities with undefined end points, however, whose location and path can be derived from the available parameters) are displayed in the form of "auxiliary contours" (see below). The graphic display of programmed chamfers and fillets is however only possible with the successive connecting element.

Parallel to this the corresponding NC block is displayed and taken into the program.

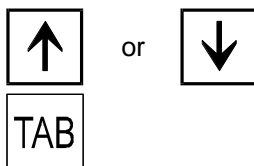
The input fields identified by the individual address characters are located at the right side of the screen. The currently active input field is highlighted in color. Finally, the current entry position is marked by the cursor.

### Procedure

To activate a specific input field, you can select from three options:

[Enter Character]

Pressing the first character of the address definition directly activates the corresponding input field. At the same time, the help screen changes accordingly.



Alternatively, you can use the **↑** or **↓** cursor keys, or the **TAB** key, to move from input field to input field, or to select a specific field.

**[Entering Values]**

You enter a number by typing the respective digits, and the places behind the comma after typing <.> (period/decimal point). Otherwise, the normal editing functions apply (see above).



or



Once you have transferred the available values into the input fields, conclude your entries either by pressing the key, or select End Input with the function key. The latter option is required in all cases in which you have made certain modifications to the geometric definition (such as Correction, Alternative Selection, etc.).

Subsequently another Entity menu is loaded for selection of the next contour entity.

**Note**

As long as you have not used the WOP Interface to specify a geometrical entity, the definition of the coordinates of the current tool position or - after a cycle start - the contour starting point is based on the system status or on the last NC programming data, however, the path of the contour line is not specified. Therefore, in this case, it is not possible to select a tangentially connecting element either.

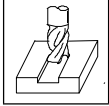
## Input Parameters for "Linear Path"

Under the DIN 66025 standard a linear path can be programmed with the preparatory function "G00" (rapid traverse motion) or "G01". The MTS Programming Code provides a further contour definition command "G71". (see Programming Instructions.)

These NC commands provide the parameters listed below. (In this context, any addresses that were stored by previous entries, and that remain valid until they are overwritten by a new value, are called "**self-retaining**" or **modal** addresses. In contrast, all addresses that **must** be entered are called "**mandatory**".)

Of course, the respective address values depend on the technology used:

	<b>G00:</b>		
	End coordinates:	$X_e, Z_e$	(modal)
	<b>G01:</b>		
	End coordinates:	$X_e, Z_e$	(modal)
	Fillet:	$R_c$	
	Chamfer length:	$F_c$	
	<b>G71:</b>		
	End coordinates:	$X_e, Z_e$	
	Start angle:	$A_w$	
Path length:	L		
Fillet:	$R_c$		
Chamfer length:	$F_c$		

	<b>G00:</b> End coordinates: $X_e, Y_e, Z_e$ (modal)
	<b>G01:</b> End coordinates: $X_e, Y_e, Z_e$ (modal) Fillet: $R_c$ (ref. X/Y plane) Chamfer length: $F_c$ (ref. X/Y plane)
	<b>G71:</b> End coordinates: $X_e, Y_e, Z_e$ Start angle: $A_w$ (ref. X/Y plane) Path length: $L$ (ref. X/Y plane) Fillet: $R_c$ (ref. X/Y plane) Chamfer length: $F_c$ (ref. X/Y plane)

### Input Parameters for "Circular Arc"

Under the DIN 66025 standard a circular arc can be programmed with the preparatory functions "G02" (clockwise) or "G03" (counter-clockwise). The MTS Programming Code provides the additional contour definition commands "G72" and "G73" (see Programming Instructions). The following parameters are used when programming with these NC commands:

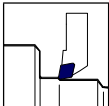
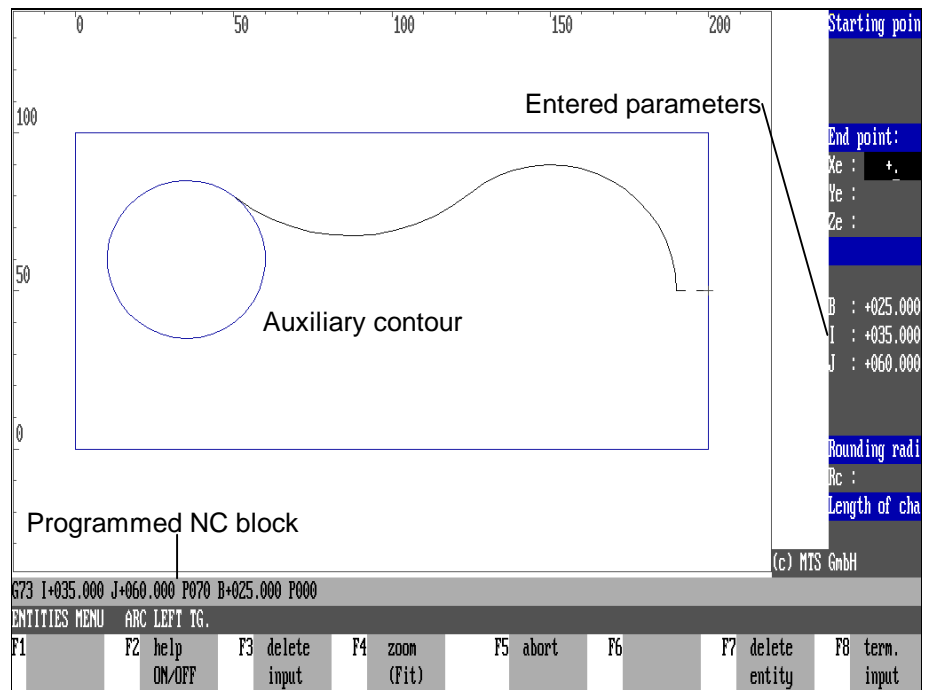
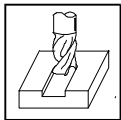
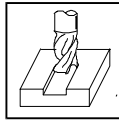
	<b>G02 (clockwise):</b> End coordinates: $X_e, Z_e$ (modal) Center point coordinates: $I, K$ (optional; either I or K should be entered, since in absence of entry the value of the center point coordinate is set equal to the corresponding start coordinate.) Fillet: $R_c$
	<b>G03 (counter-clockwise):</b> End coordinates: $X_e, Z_e$ (modal) Center point coordinates: $I, K$ (optional, see G02 above) Fillet: $R_c$
	<b>G72 (clockwise):</b> End coordinates: $X_e, Z_e$ Start angle: $A_w$ Arc radius: $B$ Center point coordinates: $I, K$ End angle: $E_w$ Fillet: $R_c$
	<b>G73 (counter-clockwise):</b> End coordinates: $X_e, Z_e$ Start angle: $A_w$ Arc radius: $B$ Center point coordinates: $I, K$ End angle: $E_w$ Fillet: $R_c$

Figure 12.2-3:  
WOP Interface, CNC  
Milling,  
Input Menu for contour  
definition,  
open contour element:  
since the end point of  
the circular arc  
(counter-clockwise,  
G73) has not yet been  
defined, an "auxiliary  
contour" in the form of  
a full circle is displayed.



	<b>G02 (clockwise):</b> End coordinates: $X_e, Y_e, Z_e$ (modal) Center point coordinates: $I, J$ (optional, s.a.: G02, Turning) Fillet: $R_c$ (ref. X/Y plane)
	<b>G03 (counter-clockwise):</b> End coordinates: $X_e, Y_e, Z_e$ (modal) Center point coordinates: $I, J$ (optional, s.a.: G02, Turning) Fillet: $R_c$ (ref. X/Y plane)
	<b>G72 (clockwise):</b> End coordinates: $X_e, Y_e, Z_e$ Start angle: $A_w$ (ref. X/Y plane) Arc radius: $B$ (ref. X/Y plane) Center point coordinates: $I, J$ (ref. X/Y plane) End angle: $E_w$ (ref. X/Y plane) Fillet: $R_c$ (ref. X/Y plane)
	<b>G73 (counter-clockwise):</b> End coordinates: $X_e, Y_e, Z_e$ Start angle: $A_w$ (ref. X/Y plane) Arc radius: $B$ (ref. X/Y plane) Center point coordinates: $I, J$ (ref. X/Y plane) End angle: $E_w$ (ref. X/Y plane) Fillet: $R_c$ (ref. X/Y plane)

**CNC Milling**

Corresponding to the provisions of the WOP Interface, any changes in the value of Z, as well as the values of angles, length and radii are displayed after their projection on the X/Y plane (helical interpolation). However, in the case of fine (detailed) contours (fillets and chamfers), any change in Z precedes the modification of the detailed contour.

**Cross Reference**

In the programming of **multiple-point contour definitions**, **open contour definitions** and **tangentially connecting contour elements**, the address selection options are generally limited. Since address selection depends on the previous contour, it cannot be shown as a schematic here. Please, refer to the Programming Aids or the Programming Instructions for further details on this subject.

On the contrary, in many cases it is possible to begin by confirming incomplete geometrical data, and to complete in the next NC block the definition of the contour segment. However, if the data available is not sufficient, the programming of the current NC block cannot be completed. This, on the other hand, also guarantees that only correct NC blocks can be programmed (see below).

Figure 12.2-4:  
WOP Interface, CNC  
Turning, Alternative  
Selection:  
Mathematically,  
certain value entries  
give two solutions for  
the contour segment;  
the desired one is  
selected in the  
Alternative Selections  
menu.

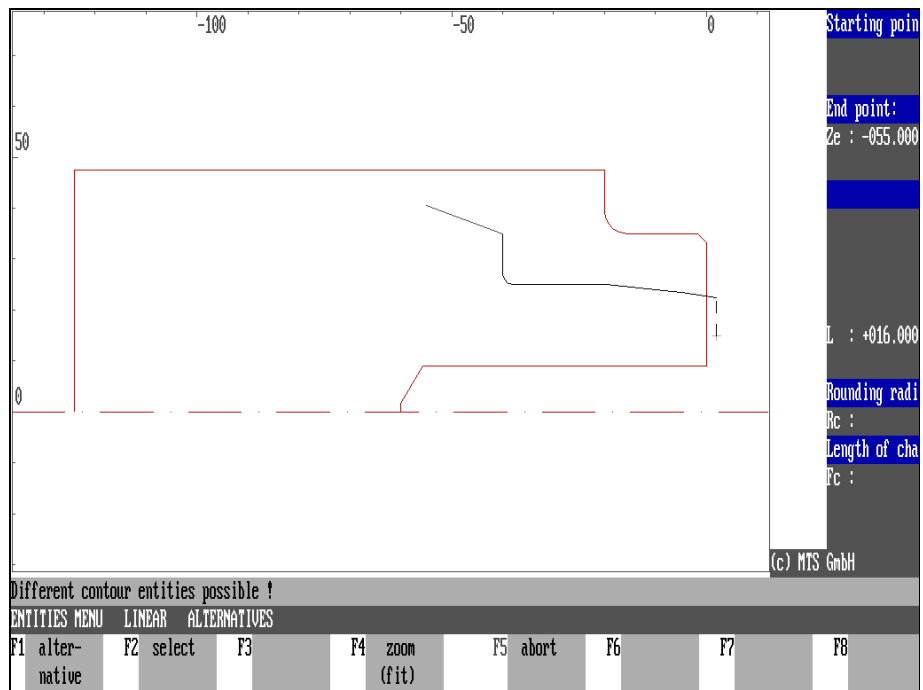
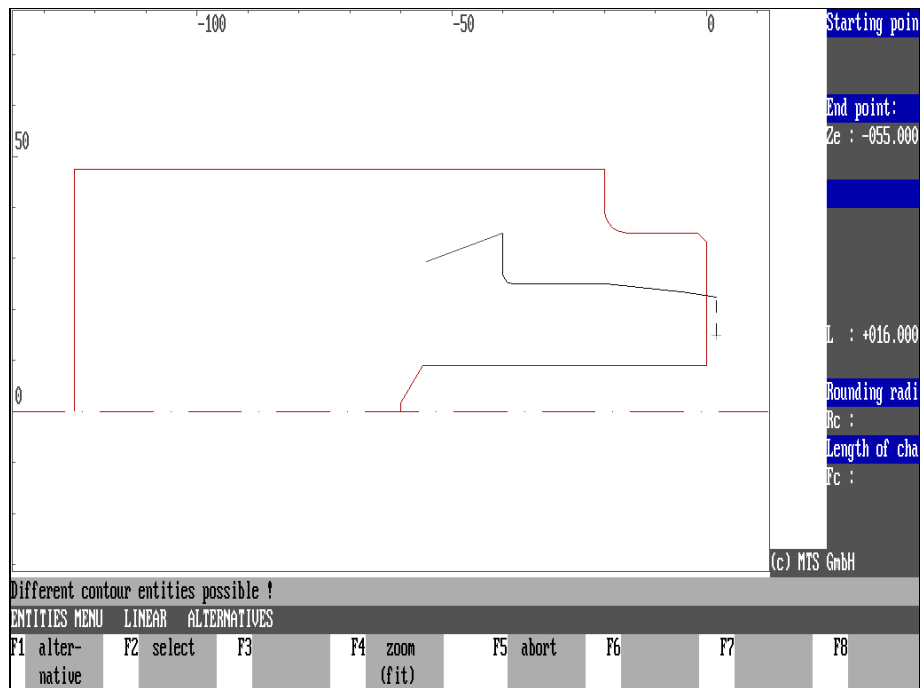


Figure 12.2-5:  
WOP Interface, CNC  
Turning,  
second alternative to  
the preceding  
example: the  
alternative selection  
refers to the current  
NC block.





## Selecting Alternatives

Frequently, the entered parameter values result in several possible variants of the contour segment being programmed. In such cases the WOP Interface switches to the Alternative Selections menu to enable you for the selection of the desired contour alternative.

The change into the Alternative Selections takes place automatically, and you cannot continue programming until after having selected one contour variant, or until after canceled the current programming session. Please note that the Alternative Selection menu displays solutions that are mathematically possible, irrespective of the fact that the solution might not be feasible from the technical or logical point of view.

After the selection you can continue your work in the Input Menu.

**F1**

**Alternate Solution:** After switching to the Alternative Selections menu, the first alternative is graphically displayed. To see the other alternative solutions, press the **F1** function key.

**F2**

**Select:** Press **F2** to select the currently displayed alternative variant. You return then automatically to the Input Menu.

**F4**

**Zoom (Fit):** Like other program modules, the Alternative Selections menu allows you to toggle between an enlarged screen cutout and the full display view (see also the chapter on Zooming).

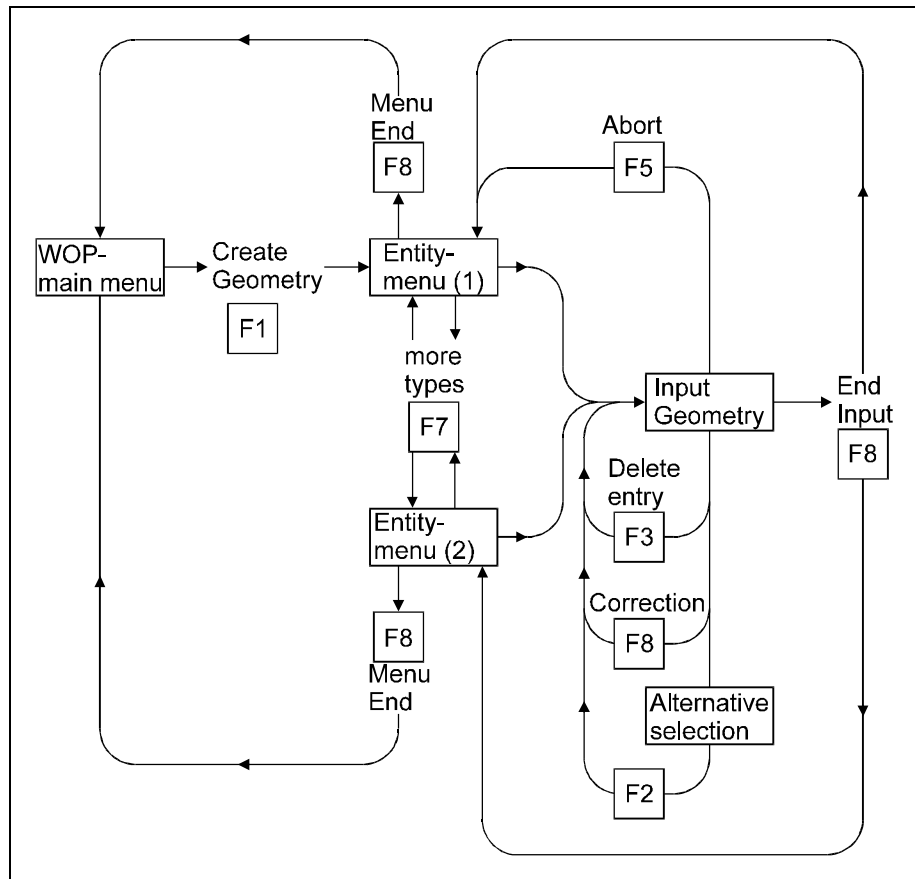
**F5**

**Cancel:** If you do not want to select any of the given alternatives, but prefer to ignore the current NC block, press the **F5** function key. This deletes all entries of the previous geometric definition, and returns you to the Entity menu.

### Note

If during work in the Alternative Selections menu you find that none of the given solutions is acceptable because of incorrect parameter values, you can accept any of the offered alternatives for the moment, and then continue by correcting the programmed parameter values.

Figure 12.3-1:  
 Schematic diagram of  
 WOP Operations:  
 Starting from the Main  
 Menu,select the  
 desired geometry  
 element in the Entity  
 menu; continue by  
 entering the correct  
 address values in the  
 input fields of the Input  
 Menu, or modify the  
 displayed data.  
 Confirmation of the  
 entry is followed by  
 selecting the next  
 geometric entity.



## 12.3 Contour Modification

For defining geometry in the Workshop-Oriented Programming Interface, the program provides several corrective functions for **modifying** or **deleting** a current contour element (geometric entity) or one or more NC blocks. Also, the WOP Interface can be used to modify the programming for a complete string of contour segments or specific contour segments thereof. In doing so, it is irrelevant which programming mode of the CNC Simulator was used to generate this contour, or whether it was generated by the INCAD CAD/CAM system or an other NC programming system. It is however necessary that the NC program is available as a program file coded according to MTS coding conventions.

After the applicable NC program is loaded, all NC blocks can be **copied** ("read in") into the WOP Interface, and the existing geometrical definitions can be checked and modified, if required. In addition, the WOP Interface gives you the option of **resetting** a programmed, copied or modified contour to a specific contour point for the purpose of adding a new geometrical definition or specifying new contour variations.

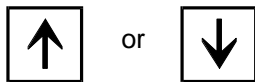
### Corrections during Parameter Input

Several correction functions are available. When working in the Input Menu, you can choose between the following options:

#### Procedure



**Correction:** In the standard editing mode, after a value has been entered, it is not possible to reactivate an input field to correct the input data. You can, however, select the Correction Mode by pressing the **F6** function key to activate the available input fields for overwriting their current values. In Correction Mode, the menu option "Correction" is not available.



or

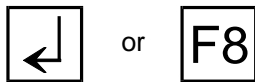
Use the cursor keys or the **TAB** key to select the input field.

[Enter Value]

A value is entered a number by typing the corresponding digits; a comma (decimal point) is represented by the <.> (period) character. Also, the normal editing functions apply (see above).



**Delete Entry:** Use **F3** to delete the entry for an address character if you do not wish to program it.



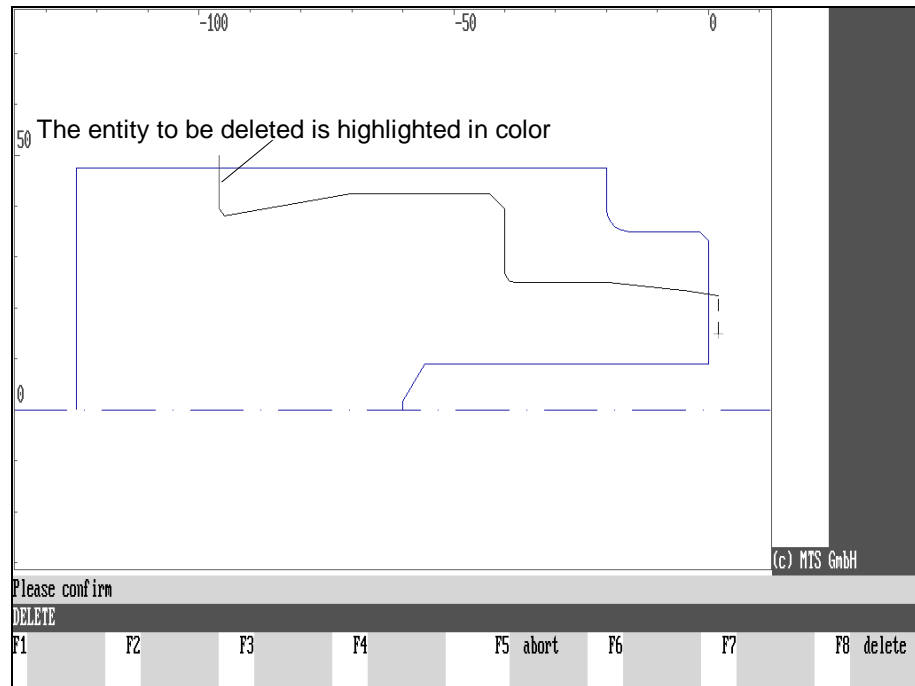
or

**End Input:** Once you have corrected the values, conclude the operation by pressing **F8**.

#### Note

Modifying existing values during Corrective Mode automatically causes a corresponding change in the generated NC block, as well as in the graphic contour display. Consequently, the deletion of a complete set of data normally causes the accompanying contour graphics to be deleted as well, and additional input fields are activated.

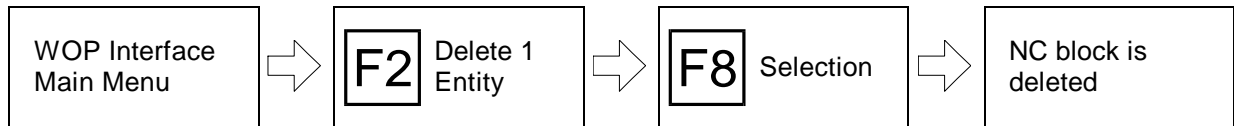
Figure 12.3-2:  
WOP Interface, CNC  
Turning,  
Delete Element:  
Before the  
programming of a  
geometrical entity is  
deleted, it is  
highlighted in color  
for  
clarity and to avoid  
any mistake.



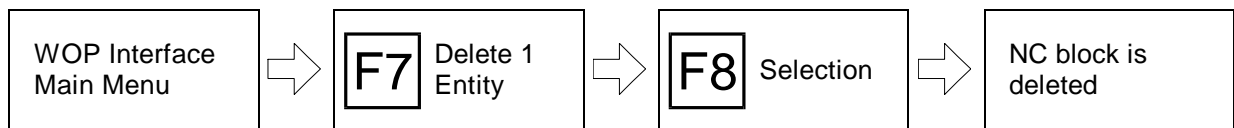
## Delete Entity

This menu option is available only if an element has already been programmed. The Delete 1 Element program function is available in the Main Menu as well as in the Input Menu.

### Sample Selection 1:



### Sample Selection 2:



### Procedure

**F2** or **F7**

**Delete Entity:** This program function is selected either in the Main Menu by pressing **F2**, or in the Entity Menu by pressing **F7**. It causes the corresponding menu to be loaded, and the last programmed contour element to be highlighted in color.

**F5**

**Cancel:** If you do not wish to delete the selected entity, press the **F5** function key. You then return to the menu level from which you initially selected the Delete Element function.

**F8**

**Select:** To delete the selected entity, press the **F8** function key. The corresponding NC block is deleted, and the system returns to the menu level from which you initially selected the Delete function.

Figure 12.3-3:  
WOP Interface, CNC  
Milling,  
Defining the "restart  
point":  
Use the cursor to place  
the displayed  
crosshairs on the  
contour element from  
which you wish to  
continue programming.

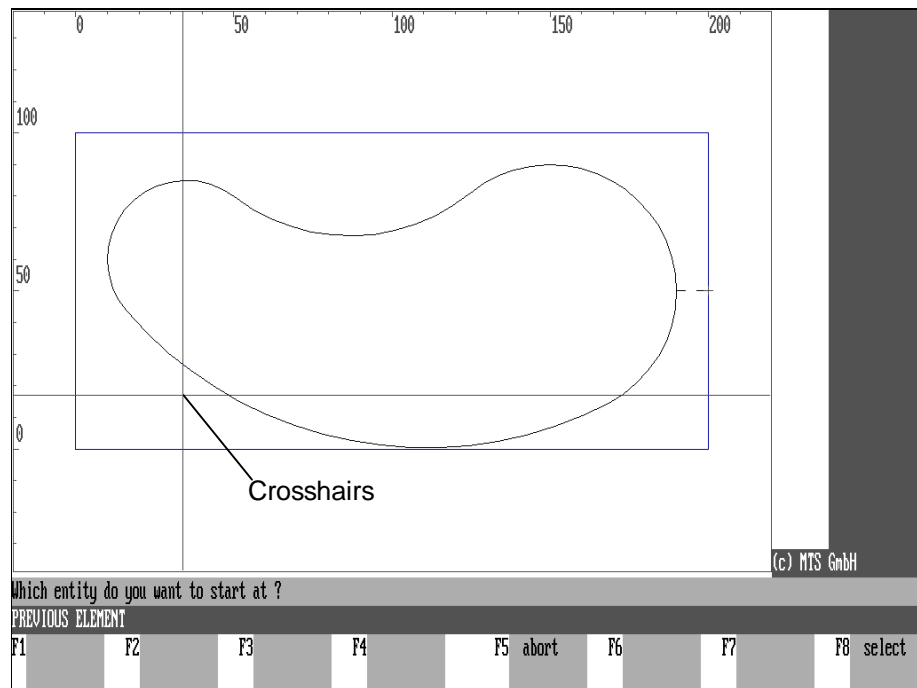
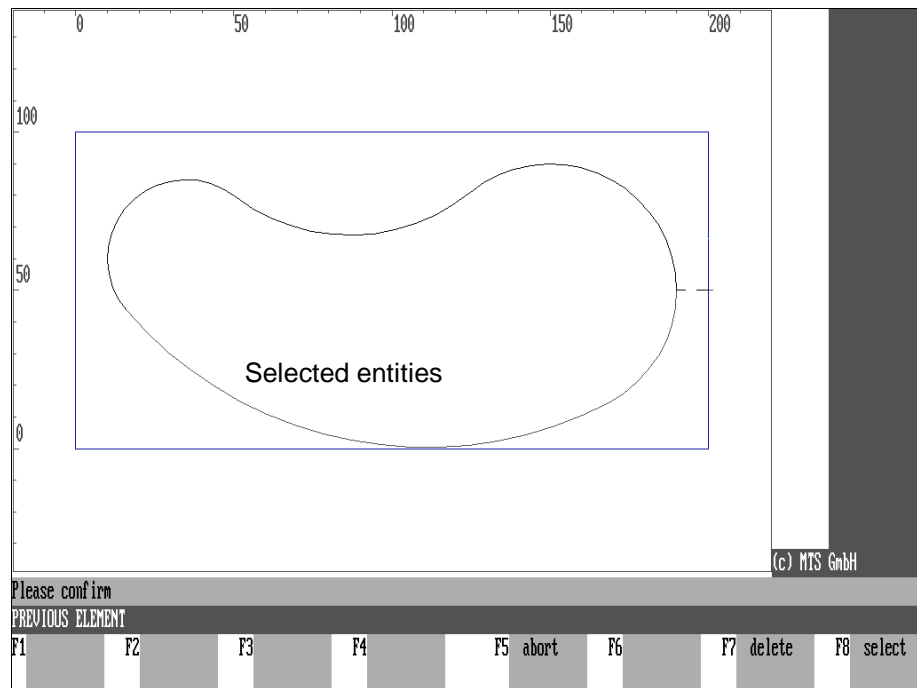


Figure 12.3-4:  
WOP Interface, CNC  
Milling,  
Reset:  
After defining the  
"restart point" you can  
either delete the NC  
blocks corresponding  
to the selected  
subsequent entities  
(colored markings), or  
place them in the  
background and  
"suspend" them.



## Reset

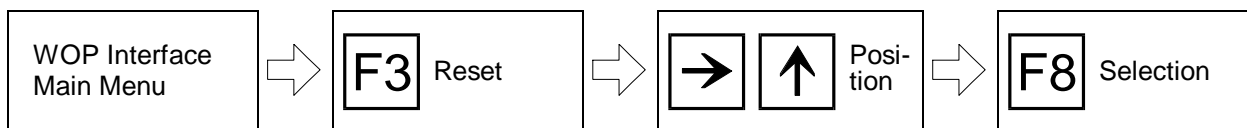
In addition to the option of deleting an NC block, you can reset a programmed or copied contour definition to a specific contour point, and then modify the contour by restarting editing from this "**restart element**". In this procedure, the following options are available:

1. **The contour definition data following the restart element is deleted**; this means that the corresponding NC blocks are no longer available, and that the contour needs to be reprogrammed from the restart point onwards.
2. As an alternative, you may wish to modify the existing geometry by changing the existing NC blocks, or by inserting new ones. In this case you can **temporarily place the contour elements following the restart point in "suspension" in the background**, to keep them available for eventual later access and reactivation.

This procedure causes the "suspended" contour elements to be identified by a different color. The newly created NC blocks are inserted into the existing program. After finishing the desired modification, you can use a program dialogue to decide whether you wish to delete the remaining NC blocks from program memory, or not.

## Defining the Restart Point

### Sample Selection:



### Procedure

**F3**

From the WOP Interface Main Menu, select the Reset program function by pressing the **F3** function key.

**→**

or

**←**

This opens a menu in which you specify the contour point to which the programming sequence is to be reset (see figure).

**↑**

or

**↓**

Use the cursor keys to move the **crosshairs** to the first element that follows the restart point.

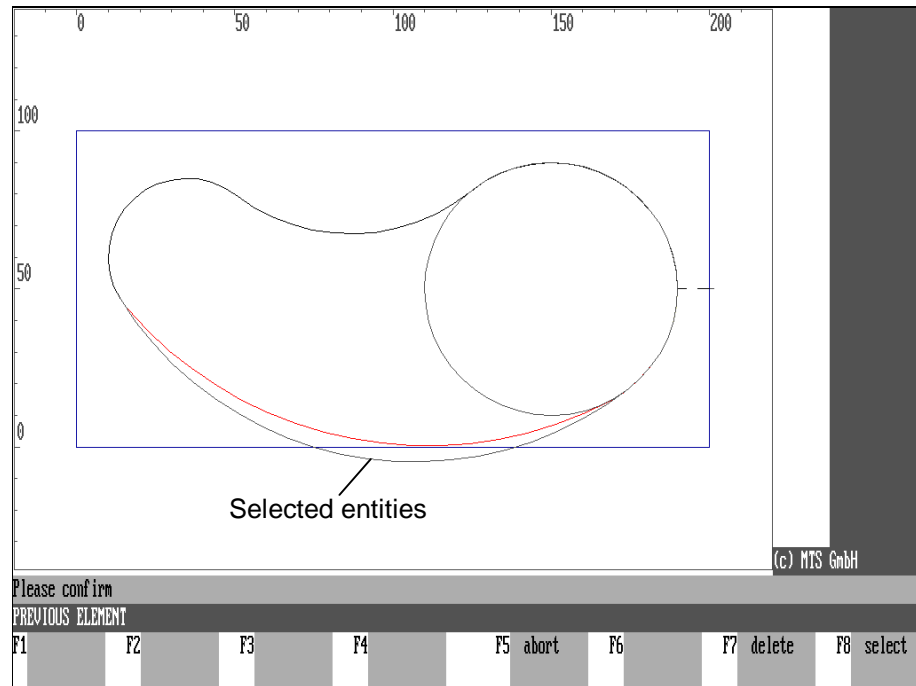
**F5**

**Cancel:** To cancel the Reset program function, press the **F5** function key. You then return to the Main Menu

**F8**

**Selection:** After placing the crosshairs on the desired point, press **F8** to confirm this "selection". The subsequent menu is loaded automatically.

Figure 12.3-5:  
WOP Interface, CNC  
Milling,  
Conclude Reset Menu:  
the programming  
sequence was reset  
and subsequently the  
contour modified; prior  
to quitting the WOP  
Interface you need to  
decide if you delete  
the suspended NC  
blocks or not.





## Deletion / Background Suspension of Elements

In the subsequent menu, the contour segment that is selected for suspension is highlighted in color (see also Fig. 12.3-5). As already described, you can recheck your restart point and decide whether the NC blocks defining the subsequent contour segments should be deleted or retained suspended in the background.

**F5**

**Cancel:** If the specification of the restart point is not correct, cancel the operation by pressing **F5**, and repeat the selection procedure.

**F7**

**Delete:** For immediate deletion of the NC blocks defining the selected contour segments, press the **F7** function key. The NC blocks is no longer available.

**F8**

**Suspend:** The Suspend function holds the selected elements (and the corresponding NC blocks) suspended in the background. You can subsequently insert new NC blocks into the existing program, or reactivate the suspended NC blocks for further modification.

## Accept Suspended Background Elements

Having returned to the Editor, or to the Interactive Programming mode, after concluding the new geometry definition, a program dialogue prompts you for a final decision whether to delete the remaining suspended NC blocks, or whether to keep them in the NC program:

### Procedure

**F1**

**Accept:** To accept these NC blocks, press the **F1** function key.

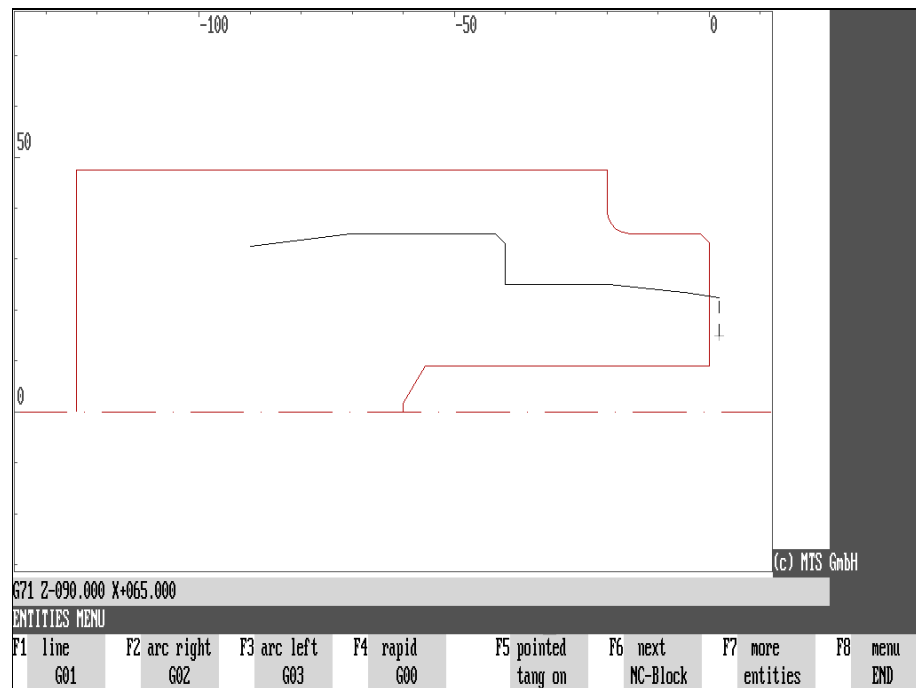
**F2**

**Discard:** To delete these NC blocks irrevocably, confirm the query by pressing the **F2** function key.

### Note

The NC blocks that were newly created as the result of a contour segment modification are taken into the NC program, i.e., placed in front of the NC blocks that were held suspended in the background.

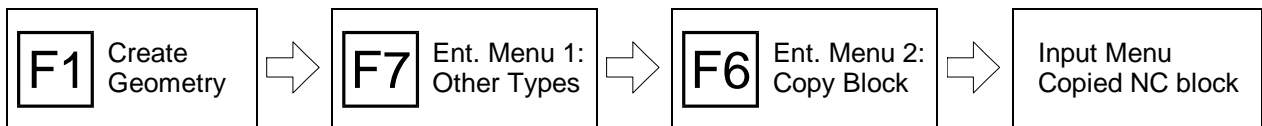
Figure 12.3-6:  
WOP Interface, CNC  
Turning,  
Entity menu:  
Starting from Entity  
Menu 2, existing NC  
blocks can be copied  
to the WOP Interface  
by pressing the **F6**  
function key.



## Copying Existing NC Blocks

As discussed in the foregoing, all NC blocks of the currently loaded NC program can be copied ("read in") into the WOP Interface, providing the WOP was started from the Interactive Programming mode. The modification options are limited to preparatory functions defining linear and circular paths.

### Selection:



### Procedure

The starting point is the WOP Interface Main Menu that you invoked while in the Interactive Programming mode:

**F1**

**Create Geometry:** Use the **F1** function key to open the Entity menu for programming a contour element, or for copying an existing NC block into the WOP Interface.

**F7**

**Other Types:** The WOP includes one Entity menu for contour definition commands, and one for DIN commands. Provided that additional NC blocks are available, the latter menu allows the option to Copy NC Blocks. Press **F7** to toggle between the two Entity menus.

**F6**

**Copy NC Block:** Press the **F6** function key to copy the following NC block. After copying the NC block, the Input Menu is available to allow editing.

[Modify Entries]

If the copied NC block defines a geometrical element, this is graphically displayed, and the entry values can be corrected.

**F8**

**End Entry:** After checking and possibly modifying the NC command, conclude the operation by pressing **F8**. Subsequently you can copy a new NC block, or define a new contour element.

### Note

Certain preparatory functions which do not define a geometrical element can be copied into the WOP Interface, however, they cannot be modified at all or only to some extent. For example, although the command " G54 X+200.000 Y+100.000 Z+065.000 " is displayed after copying, it cannot be modified.

Figure 12.4-1:  
WOP Interface, CNC  
Milling,  
"Auxiliary Contour"  
display in linear path  
definition:  
Since the starting point  
and the angle are  
known, an auxiliary  
line without end point,  
indicating the path of  
the linear string, is  
displayed

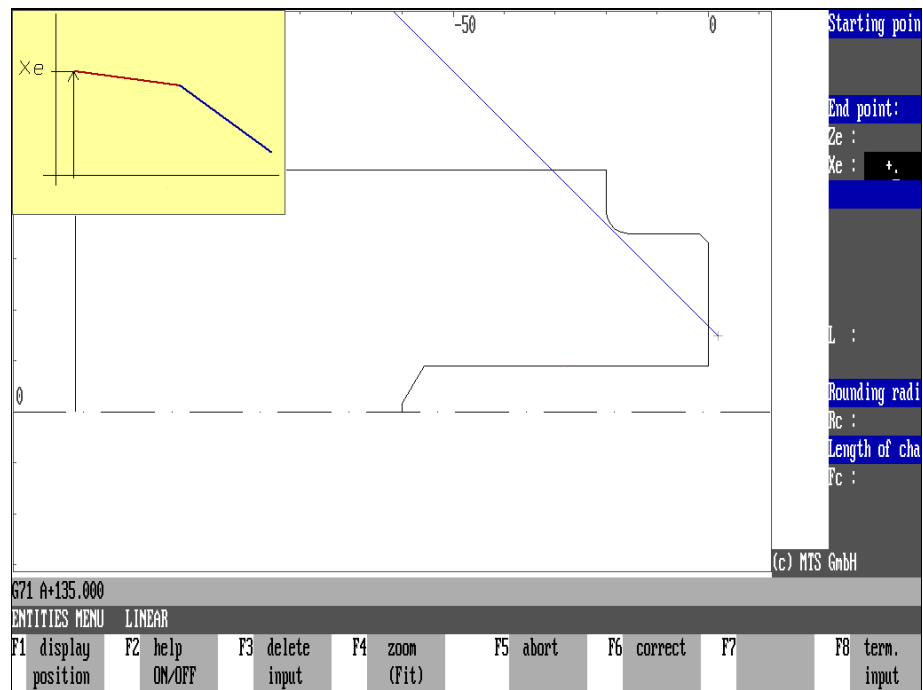
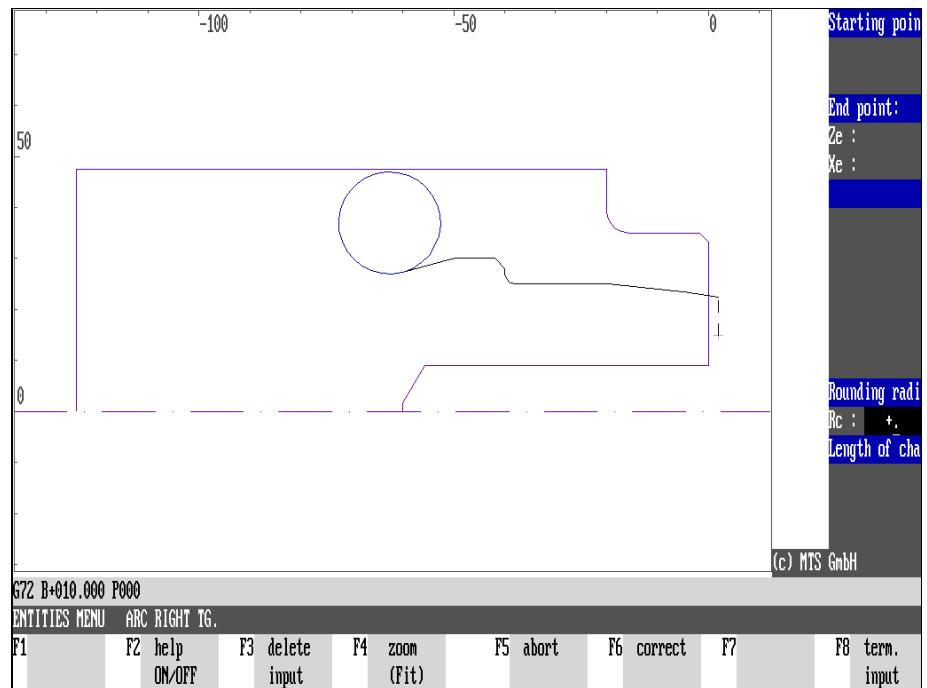


Figure 12.4-2:  
WOP Interface, CNC  
Turning,  
open contour entity:  
"Auxiliary Contour"  
in defining a tangentially  
connecting circular  
arc; based on this  
transition condition and  
the entry of the radius,  
a full circle can be  
displayed on which the  
contour entity end  
point is located.



## 12.4 Contour Definition Programming based on Non-NC Standard Dimensioned Drawings

### Summary

The definition of geometrical elements by means of the WOP Interface not only facilitates the programming of complex contours, but also helps in creating NC programs based upon drawings which are dimensioned in standards incompatible with NC conventions. Since on one hand, the application of the contour definition commands G71, G72 and G73 also allows the entry of **additional geometrical data**, such as angles, lengths, tangential transition conditions, etc., and on the other hand **the precise definition of each element is not necessary**, most problems of daily practice can be solved without additional tools.

After each data entry, the user dialogues of the Workshop-Oriented Programming Interface evaluate which further address definitions are needed, while simultaneously providing a context-dependent selection of the geometrical elements. Normally this means that the user only needs to transfer the existing drawing data.

In case of **incomplete data entries** the contour cannot be graphically displayed. However, as soon as the "path" of a geometrical entity is defined, an "**auxiliary contour line**" is displayed, even if the end point is not known.

Two examples: For the definition of a linear path, the starting or connecting point and their starting angles are known; in this case, calculation of the "path" of the linear string is already possible, and an "auxiliary line" without end point is displayed on the screen (see figure).

In case of circular arc the situation is similar: Again, the starting point as well as the center point coordinates are available. Based on the available data, the system automatically calculates a full circle on which the end point of the circular arc is located. In this case, too, the full circle is displayed as an auxiliary contour.

In the above examples the starting point of the currently edited contour element was known. It is often identical to the programmed end point of the preceding contour element. However, it is possible that the **drawing does not supply information regarding the end point**. For example, this is always the case if a fillet (based upon the programming code) is defined by a radius only.

Previous discussions of this subject have shown that even in this situation, programming in WOP can be continued. If a precise geometrical definition follows, **the last transition point is evaluated retroactively**. Accordingly, the entire string of contour elements can again be graphically displayed (in this manner, to use the previous example, the fillet can be represented only together with the corresponding succeeding element).



**This summary description of the system internal contour calculations remains necessarily somewhat abstract even if the actual use of the WOP Interface is very easy because of the numerous menus and user dialogues. Therefore, you should just start to work with the WOP Interface and get experience by using it.**

Figure 12.4-3:  
WOP Interface, CNC  
Milling,  
Segment Contour  
Programming with the  
WOP Interface.

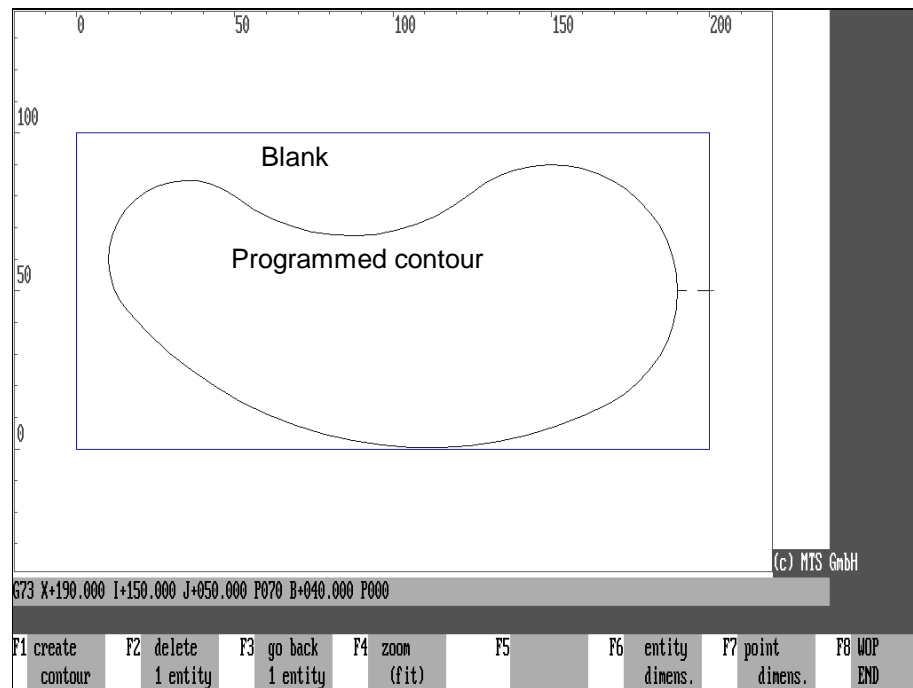


Figure 12.4-4:  
Related NC program  
(excerpt): combining  
an open and a closed  
four-point contour  
string; plus  
explanations:

```

N105 ( EXTERNAL CONTOUR WITH RADIUS COMPENSATION
N110 G42 X+190.000 Y+050.000 G46 A+005.000 G01 Z-020.000
N115 G73 I+150.000 J+050.000 P070 P001
N120 G72 B+060.000 P000
N125 G73 I+035.000 J+060.000 P070 B+025.000 P000 P001
N130 G73 B+125.000 P000
N135 G73 X+190.000 I+150.000 J+050.000 P070 B+040.000 P000
N140 G40 G46 A+010.000

```

- N110 Infeed adjustment to Z, selecting cutter radius compensation with approach condition in semi-circle
- N115 Circular arc G73 with I/J center point (absolute dimension: P070) and alternative selection P001.  
This circular arc comprises the first element of an open four-point string ["FPS"]. Together with the starting point, a circular "auxiliary contour" is calculated on which the end point must be located.
- N120 Circular arc G72 with radius B.  
Second element of the open FPS; since neither transition point nor center point coordinates are known, the contour progress cannot be calculated.
- N125 Circular arc G73 with center point I/J (absolute dimension: P070), radius B and alternative selection P001.  
Third element of the open FPS; this data defines a circle as an open element on which the end point of the circular arc must be located. Also, the program control can use the alternative selection to automatically and retroactively calculate the previous transition points from the two circles (N115 and N125) and the radius definition (N120); only the end point of the last element remains undefined. Subsequently this element will internally be used as the first element of a new FPS.
- N130 Circular arc G73 with radius B.  
Second element of a FPS; since neither the transition point nor the center point coordinates are known, the contour cannot be calculated.
- N135 Circular arc G73 with end coordinate X, center point I/J (absolute dimension: P070) and radius B.  
Third element of the FPS; this data defines a circular arc. The program control uses the two circles (N125 and N135) and the radius definition (N130) to automatically and retroactively calculate the missing transition points. Accordingly, the full contour is precisely defined.

## Open Contours and Multiple-Point Definitions

The described retroactive contour calculation might first appear to be nothing special, it creates, however, a **multiple combination of "incomplete" NC blocks** which together render a precise definition of a contour path! Of course, the condition is that, after an incomplete geometrical definition, the address data of a succeeding NC block defines the last transition points and the "basic contour profile". Any contours resulting from such NC block combinations are generally referred to as "**open contours**".

Beside the definition of open contours, the described "calculation principle" provides a second linking of incomplete NC blocks that is of special interest in professional applications: It is the programming of so-called "**multiple-point strings**". Multiple-point strings are contour definitions comprised of two or more contour elements that base the calculation of their final profile on the "combination" of the individual data. They can be chained in random sequence, and their last geometrical element may again consist of an open contour.

### Cross Reference

**At this point, the discussion of open and multiple-point contours gives you only the basic information required to use of the WOP Interface. For more detailed questions, we recommend you to refer to the relevant discussions in the Programming Instructions.**

Therefore, a brief concluding outline of the use of the WOP Interface in this special mode of NC programming is presented at this point:

### Procedure

The programming of open and multiple-point contour definitions with the WOP Interface follows the same conventions as entering geometric elements by means of complete data sets. Since the program control presents the selection options and input fields automatically, "overdefining" through entered data is not possible. It is furthermore not necessary to search permissible address assignments in the programming manual required.

In addition, the WOP Interface allows the programming of several undefined elements as long as the geometric elements may be calculated retroactively. **Only if several successive incomplete geometric definitions fail to yield a result, the current entry cannot be concluded.** During data entry such a situation is available if the **[F8]** function key (End Entry) in the Input Menu is not activated yet. If this occurs, you can either enter the pending values in the remaining address fields, or you can cancel the data entry procedure.

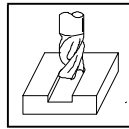
### Note

Before you conclude working with the WOP Interface, it is recommended that you use the last geometric element to render a precise contour definition. Only then the corresponding NC blocks can be executed and tested in Automatic Mode or Interactive Programming Mode. In contrast, if the simulation is started for an incomplete contour definition programming, the program displays the error message, "Contour definition not concluded".

## Appendix 1

### CNC Symbols and Miscellaneous Icons

This appendix provides an overview over the functions of the various CNC symbols and miscellaneous icons used in working with the CNC Simulator.



The right-hand side column below shows the corresponding functions for CNC Milling technology as far as these are different from those of the CNC Turning technology (left-hand column).

CNC  
Symbol

Key Function

CNC  
Symbol

Key Function **CNC Milling**

#### General-purpose Control Keys:



**Return** to your starting situation, jump to previous menu level



**Execute/Accept**, go to next procedural step



**Cancel**, return to previous editing/working situation

#### Main Menu:



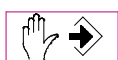
NC Editor and Program Management



Automatic Mode



Setup Mode



Automatic Setup



Colour selection and graphic display modification



Setup Sheet functions



Status Management



Exit from Simulator operation

#### NC Editor Functions:


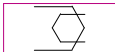











Edit Program

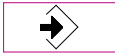


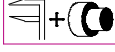
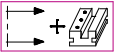






Delete Program



CNC Symbol	Key Function	CNC Symbol	Key Function <b>CNC Turning</b>
	Print program		
	Link program		
	Set editing range		
	Find Word/Character string/Error		
	Renumbering		
	Start block operation		
	Workshop-Oriented Programming		
	Help function		
	Conclude NC programming		
	End programming without saving		
	End programming after saving		

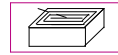
**Automatic Mode functions:**

	Accept program name		
	Select range		
	Override, Test Run and Times settings		
	Measuring and 3D Views		Sectional display and 3D Views
	Modifying graphic display		
	Continuous automatic run		
	Interactive Programming		
	Single NC block mode		

CNC Symbol	Key Function	CNC Symbol	Key Function <b>CNC Milling</b>
------------	--------------	------------	---------------------------------



Traverse path display

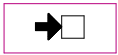


Traverse path display



Zoom function (dynamic)

### Interactive Programming and Single NC Block Mode:



Execute NC block

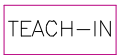


Accept NC block



Reject NC block

### Teach-In Mode:



Teach-In Mode



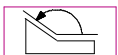
Setting the increment



Technology switching commands



Zero point programming



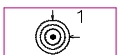
**CNC Turning only :**  
Entering angle for operation



End Teach-In Mode, return to Interactive Programming Mode



Delete last motion command generated



Increment: 1.0 mm



Increment: 0.1 mm



Increment: 0.01 mm







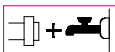




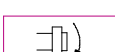








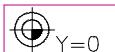

Increment: 0.001 mm



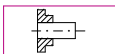
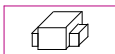


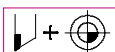
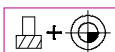
Enter feedrate in mm per revolution



RPM spindle speed coding

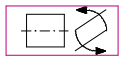
CNC Symbol	Key Function	CNC Symbol	Key Function <b>CNC Milling</b>
	Maximum RPM spindle speed coding		
	<b>CNC Turning only</b> : Enter constant cutting speed in m/min		
	Program tool change		Program tool change
	Menu call: M commands spindle and coolant		
	Coolant OFF (M09)		
	Coolant pump 1 ON (M08)		
	Coolant pump 2 ON (M07)		
	Spindle OFF (M05)		
	Spindle ON: clockwise (M03)		
	Spindle ON: counter-clockwise (M04)		
	Workpiece zero point: Define Z coordinate		Workpiece zero point: Define X coordinate
	Workpiece zero point: Define X coordinate		Workpiece zero point: Define Y coordinate
	Workpiece zero point: current Z coordinate = 0		Workpiece zero point: current X coordinate = 0
	Workpiece zero point: current X coordinate = 0		Workpiece zero point: current Y coordinate = 0
			Workpiece zero point: current Z coordinate = 0

**Setup Mode Operations:**

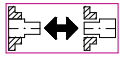
	Blank/Clamping Definition		Blank/Clamping Definition
	Referencing		
	Enter technology data		
	Tool change, turret configuration and zero point definition		Tool change, magazine configuration and zero point definition

CNC Symbol	Key Function	CNC Symbol	Key Function <b>CNC Milling</b>
------------	--------------	------------	---------------------------------

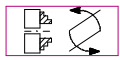
Entering technology data, see Teach-In, above



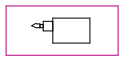
New workpiece



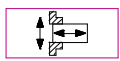
Re-clamp workpiece



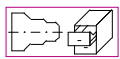
Change clamping fixture



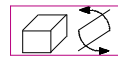
**CNC Turning only:**  
Set tailstock/sleeve



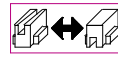
New clamping setup



Workpiece management



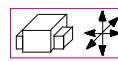
New workpiece



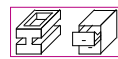
Re-clamp workpiece



Change clamping fixture

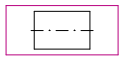


New clamping setup

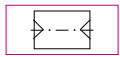


Workpiece management

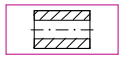
### Blank/Workpiece:



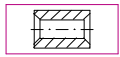
**CNC Turning only:**  
Blank: Cylinder



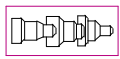
**CNC Turning only:**  
Blank: Centered cylinder



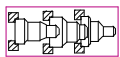
**CNC Turning only:**  
Blank: Tube



**CNC Turning only:**  
Blank: Tube with chamfer



Insert prepared workpiece, no clamping fixture



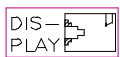
Insert prepared workpiece, use clamping fixture



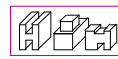
**CNC Turning only:**  
Display: Defined blank



Entry: Material



Display: Entire workspace



Insert prepared workpiece, no clamping fixture

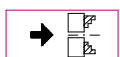


Insert prepared workpiece, use clamping fixture

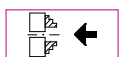


Display: Entire workspace

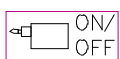
### Clamping Methods:



**CNC Turning only:**  
Select next clamping type



**CNC Turning only:**  
Select previous clamp type



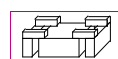
**CNC Turning only:**  
Clamping with/without tailstock



**CNC Milling only:**  
Select vice clamping



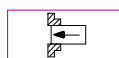
**CNC Milling only:**  
Select magnetic plate clamping



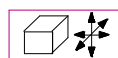
**CNC Milling only:**  
Select modular clamping

CNC Symbol	Key Function	CNC Symbol	Key Function <b>CNC Milling</b>
------------	--------------	------------	---------------------------------

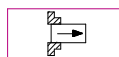
**Clamping Position:**



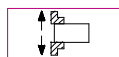
**CNC Turning only:**  
Shift workpiece toward spindle



**CNC Milling only:**  
Change workpiece position



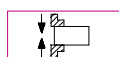
**CNC Turning only:**  
Shift workpiece away from spindle



**CNC Turning only:**  
Open jaws

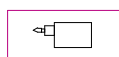


**CNC Milling only:**  
Change vice position



**CNC Turning only:**  
Close jaws

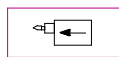
**Tailstock/Sleeve or Modular Clamping:**



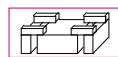
**CNC Turning only:**  
Set tailstock/sleeve



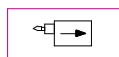
**CNC Milling only:**  
Change clamping



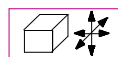
**CNC Turning only:**  
Set tailstock toward workpiece



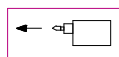
**CNC Milling only:**  
Select modular clamping



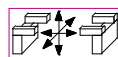
**CNC Turning only:**  
Reset tailstock



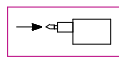
**CNC Milling only:**  
Change workpiece position



**CNC Turning only:**  
Set sleeve



**CNC Milling only:**  
Change clamping element position



**CNC Turning only:**  
Loosen sleeve



**CNC Milling only:**  
Define new clamping element

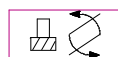


**CNC Milling only:**  
Delete existing clamping element

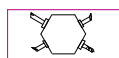
**Tools/Turret or Magazine Configuration/Workpiece Zero Point:**



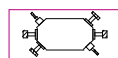
Change tool



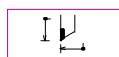
Change tool



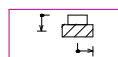
Turret configuration



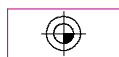
Magazine configuration



Compensation value register



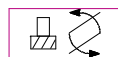
Compensation value register



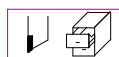
Workpiece zero point



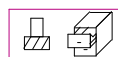
Change tool in current turret location



Change tool in current magazine location



Tool management



Tool management



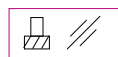
Tool: Information on current turret location



Tool: Information on current magazine location



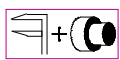
Remove tool from current turret location



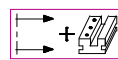
Remove tool from current magazine location

CNC Symbol	Key Function	CNC Symbol	Key Function <b>CNC Milling</b>
	Store compensation values in assigned registers		Store compensation values in assigned registers
	Display tool engagement times		Display tool engagement times
	Workpiece zero point: Define coordinate Z		Workpiece zero point: Define coordinate X
	Workpiece zero point: Define coordinate X		Workpiece zero point: Define coordinate Y
			Workpiece zero point: Define coordinate Z
	Workpiece zero point based on values saved under G54		
	Workpiece zero point based on values saved under G55		
	Workpiece zero point based on values saved under G56		
	Workpiece zero point based on values saved under G58		Workpiece zero point based on values saved under G57

### Measuring and 3D View:



Start: Measuring and 3D Views



Start: Sectional displays and 3D Views



**CNC Turning only:**  
3D Views



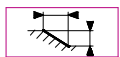
**CNC Turning only:**  
Select display



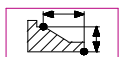
**CNC Turning only:**  
Zoom



**CNC Turning only:**  
Evaluate surface roughness



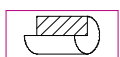
**CNC Turning only:**  
Element dimensioning



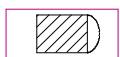
**CNC Turning only:**  
Point dimensioning



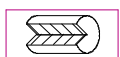
**CNC Turning only:**  
3D View: Exterior view



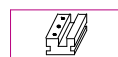
**CNC Turning only:**  
3D View: Semi-sectional view



**CNC Turning only:**  
3D View: Full sectional view



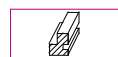
**CNC Turning only:**  
3D View: Selectable section



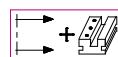
**CNC Milling only:**  
3D View: Full view





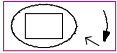





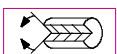


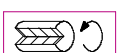

**CNC Milling only:**  
3D View: 3/4 part




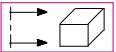

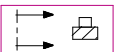






**CNC Milling only:**  
3D View: 1/4 part



**CNC Milling only:**  
3D View with sectional display

CNC Symbol	Key Function	CNC Symbol	Key Function <b>CNC Milling</b>
	<b>CNC Turning only:</b> Viewing angle, left-hand turning		<b>CNC Milling only:</b> "Z depths" dotted (based on G17)
	<b>CNC Turning only:</b> Viewing angle, right-hand turning		<b>CNC Milling only:</b> Viewing direction from front left
	<b>CNC Turning only:</b> Select section		<b>CNC Milling only:</b> Viewing direction from front right
	<b>CNC Turning only:</b> Select viewing angle		<b>CNC Milling only:</b> Viewing direction from left rear
	<b>CNC Turning only:</b> Reduce section		<b>CNC Milling only:</b> Viewing direction from right rear
	<b>CNC Turning only:</b> Enlarge section		
	<b>CNC Turning only:</b> Rotate section counter-clockwise		
	<b>CNC Turning only:</b> Rotate section clockwise		

## Modifying the Graphic Display:

	<b>CNC Turning only:</b> Display: Exterior view		<b>CNC Milling only:</b> Workpiece sectional views
	<b>CNC Turning only:</b> Display: Full sectional		<b>CNC Milling only:</b> Display: Workpiece sectional
	<b>CNC Turning only:</b> Display: Bottom semi-sectional		<b>CNC Milling only:</b> Display of axes
	<b>CNC Turning only:</b> Display: Top semi-sectional		<b>CNC Milling only:</b> Emphasizing workpiece contours
	<b>CNC Turning only:</b> Zoom		
	<b>CNC Turning only:</b> Thread display: ON/OFF		



As a rule, the text of this manual is accompanied by the corresponding PC function key symbols in the left-hand side margin of the page.

## Additional Symbols on CNC Keyboards & Graphic Tablets:






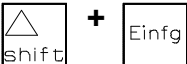

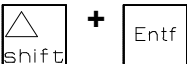












Since the CNC Simulator can be operated with the CNC keyboard or the graphic tablet alone, additional entry keys are required (NC program editing). The following section gives an overview of these keys and their corresponding PC key assignments:

+


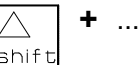




In this overview the plus character "+" is used to mark a key combination on the PC keyboard (and not the <+> key!): Similar to typing a capital letter on a typewriter by simultaneously pressing the <SHIFT> and respective character key, the respective key combinations are also entered by pressing two keys at the same time.

CNC	PC keys	key function
-----	---------	--------------

### Editing Functions:

		Insert NC word or number
		Delete NC word or number
		Insert NC block
		Delete NC block
		Page forward
		Page backward
		Cursor downward
		Cursor upward
		Cursor to right
		Cursor to left

### General-purpose Control Functions:

		This symbol indicates rapid traverse movement; on the PC keyboard the rapid traverse movement of the tool is started by simultaneously pressing the <SHIFT> key and the appropriate cursor key.
		Setting an override of + 10 %.
		Setting an override of - 10 %.
		Starting the test run.
		Ending the test run.



Rather than using the PC keyboard to control the CNC functions, they can normally be directly selected with the graphic tablet. **This means that the intermediate steps for control menus and menu options are eliminated!**



## Appendix 2

### Listing of Error Messages in Alphabetical Order

This appendix lists the error messages which are displayed in case of program execution or operating errors. This list includes most of the functions leading to an error message including hardware problems and file structure inconsistencies in NC programs and in other files...

The error messages are listed in alphabetical order in the following. Each message is commented on, specifying the possible cause of the problem and the ways of correcting it. If in doubt refer to the MTS Operating Manuals or the Programming Instructions and also check the connected hardware devices and the storage media you are using.

Information on further error messages not included here can be found in the Configuration Instructions and in the Operation Manual for Postprocessors.

#### Note

The term "unfinished" or "open contour strings" refers in the following to a sequence of standard or contour definition commands with an unspecified contour end point.

<u>Error Message</u>	<u>Cause / Correction</u>
<b>3D Movement not allowed</b>	The present configuration of the control system does not allow simultaneous movements in three axes. Make the corresponding changes in the NC program or select some other configuration setting (cf. Configuration Instructions).
<b>-A-</b>	
<b>A - Abort</b>	MS-DOS statement, occurring in various situations. Confirmation of this prompt by typing <A> cancels the current procedure.
<b>Abort editing</b>	See "The format file was modified!"
<b>Address code ... does not exist in NC-Syntax</b>	Error message during format conversion of a program file: The source program contains an address letter that is not available in the MTS programming code. Press <F1> to skip conversion of the concerned NC word and subsequently insert the command which is to be applied (according to the MTS code) in the converted program. To cancel the conversion process, press <F8>.
<b>Address combination invalid</b>	The programmed addresses / identification characters of a segment contouring command are not consistent with the contour strings of the MTS programming code. Please re-program this contour string according to the Programming Instructions.
<b>Address not valid</b>	Internal definition of auxiliary values is not possible. If this error occurs, send the defective NC program to MTS, or contact us per phone.
<b>Analyzing BEM.KNF file error:</b>	The configuration file for the program parts Workshop-Oriented Programming and Measuring could not be loaded. Check (on the DOS level) whether the file "BEM.KNF" exists in the directory in which the CNC Simulator has been installed. If the configuration file is not found, repeat system installation.

<b>A new cycle cannot be defined while radius compensation is active</b>	The attempt to call a cycle while radius compensation is active failed. Disable radius compensation before invoking a cycle or make sure that the cycle is programmed after the radius compensation has been switched off.
<b>Arguments list of the selected program too long (environment)</b>	See "Program was aborted with an error ..."
<b>-B-</b>	
<b>Before calling cycle, please deactivate tool nose radius compensation</b>	Roughing cycle cannot be invoked if tool nose compensation is active. Deactivate tool nose compensation before calling the cycle.
<b>Beginning and end of blank contour are not identical.</b>	Errors in a Setup Sheet: beginning and end point of a specified workpart contour must be identical. Please make the corresponding corrections in the contour definition or generate a new setup sheet (after preproducing the workpart).
<b>Blank dimensions inconsistent with the configured min/max values!</b>	See "Blank not permissible!"
<b>Blank not permissible !</b>	The minimum and maximum blank geometry values must be within the values specified in the configuration.
<b>Block number overflow !!! Abort renumbering. Press any key</b>	According to the MTS programming code the acceptable block number entries range from "N000" to "N999". Reduce the increment in block numbering. If an NC program contains more than 1000 NC blocks, you can edit the numbers manually.
<b>-C-</b>	
<b>Calculation of an entity suspended</b>	Computing the surface depth-and-valley heights of an entity may be time-consuming, esp. if the specified feedrate is very small. You can cancel this calculation with <ESC> and the corresponding error message is then be displayed.
<b>Can only be edited in free format mode</b>	See "The format file was modified !"
<b>Cannot overwrite the target file ! Press any key ...</b>	The specified target file name (and path) for conversion is locked. Select a new file name and try again.
<b>Cannot find "FROM"-number. Abort renumbering. Press any key</b>	The specified start number is not available. Correct your entry and call the "renumber" function again.
<b>Cannot open target file ! Press any key...</b>	The specified target file name (and path) for conversion is not correct. Check your entries and try again.
<b>Cannot read Floppy Disk! To try again, press any key</b>	Make sure that the drive selection is correct and the diskette has been properly inserted. Should this error occur again, the diskette may be defective or initialized in a foreign format. Press any key to retry reading the diskette...

<b>Centere sleeve cannot be set/disengaged !</b>	See "Spindle sleeve did not catch !"
<b>Centering does not make sense</b>	The centering of the turning workpart as specified in the blank definition, is not consistent with the blank geometry. Modify the entries regarding to centering ( cf. Chapter 4 in this Manual).
<b>Chamfer does not make sense</b>	When defining a blank, the width of the chamfer and its angle to to positive X-axis need to be specified. Correct the entries correspondingly (cf. the detailed instructions in this Operation Manual).
<b>Character-number-principle of the ISO code was ignored</b>	See "ISO blocks for blanks contain different parameters..."
<b>Chuck and spindle data inconsistent</b>	The values specified for chuck and spindle geometry are inconsistent or contradictory. Check the entries; when in doubt refer to the explanations in this Manual.
<b>Chuck does not match part.</b>	The workpart could not be clamped with the clamping device specified in the setup sheet. Either edit the entries in the setup sheet, or insert a new clamping fixture in manual mode, clamp the workpart and then select automatic generation of the setup sheet.
<b>Chucking depth impossible</b>	The workpart could not be clamped at the desired depth with the clamping device specified in the setup sheet. Change your entries in the setup sheet or select automatic generation of the setup sheet.
<b>Circular interpolation error</b>	In a G02 or G03 block either the end point or the centre point has not been calculated correctly. The end point is not located on the circle line which has already been mathematically specified by the centre and starting point. As only three places behind the decimal point are accepted for the entered coordinate values, this error may occur despite of the fact that very small circular arcs have been computed correctly. Basically, this message indicates a coordinate input error. The CNC Simulator checks coordinate values at the tolerance of 10 µm.
<b>Clock device error</b>	Hardware error: Check the employed hardware and contact your local dealer, if necessary.
<b>Codeword CHUCKING DEVICE / TOOLS is missing</b>	The respective keyword is not specified in the setup sheet. Correct the setup sheet or generate a new one.
<b>Collision during rapid traverse</b>	The programmed NC block effects a rapid tool movement which leads to a collision with the workpart. Correct the NC block so that the end point of the rapid traverse movement is located far enough from the workpart.
<b>Collision of tip chucking and jaw chucking</b>	Both jaw chucking and clamping between tips has been specified in the setup sheet. Delete one of these entries.
<b>Collision of tool with chuck</b>	The movement effected by the programd NC block leads to a collision of the tool with the clamping device or tailstock or centre sleeve. Change the values of the NC block to enable collision-free tool movement.

<b>Collision of turret (tool, tool adapter) with sleeve tip / machine space / quill / tailstock / chuck / spindle / face driver / step jaws / workpart</b>	The tool movement effected by manual control or by the programd NC block leads to a collision. Change the values of the NC block to enable collision-free movement.
<b>Collision of spindle with part</b>	The movement of the spindle head effected by the programd NC block leads to a collision with the workpart. Change the values of the NC block to enable collision-free movement.
<b>Collision while lifting</b>	The programd NC block effects a tool movement leading to a collision with the workpart "from below". Program first "tool retract" and then "lifting"
<b>Collision while tool-changing</b>	Moving the selected tool to working position into the turret leads to a collision (because of adjacent tools!). First increase distance between the turret and the other objects in the machine space, then program this tool change.
<b>Collision with desired workpart contour</b>	The movement effected by the programmed NC block leads to a gouging of the desired final contour. Change the values of the NC block to avoid gouging.
<b>Collision with machine table</b>	The programmed NC block effects a tool movement leading to a collision with the machine table. Modify the NC block to enable a collision-free tool movement.
<b>Collision with part</b>	See "Collision during rapid traverse"
<b>Collision with shank</b>	The programmed NC block leads to a collision of the tool shank with the workpart. In a multipass roughing cycle, the broaching of a falling contour span would lead to a collision. Change the values of the NC block to enable collision-free tool movement. Also check the programmed contour, the selected tool and, if applicable, the starting point of the multipass cycle. You may have to choose a tool with a larger infeed angle.
<b>Collision with spindle while sinking</b>	The movement of the spindle head effected by the programmed NC block leads to a collision with the spindle. Change the values of the NC block to enable collision-free movement.
<b>Collision with tool</b>	See "Collision during rapid traverse"
<b>Collision with tooth</b>	The programmed NC block effects a non-centred (not X=0) drill movement. Change the values of the NC block to enable centred drilling.
<b>Collision with turret head</b>	The movement effected by the programmed NC block leads to a collision of the turret head and the workpart (occurs frequently when machining inside the workpart, also when the workpart is approached preaxially by a tool for postaxial cutting, and vice versa). Change the values of the NC block to enable collision-free movement.
<b>Collision with work space</b>	The attempt to move the tool carrier beyond the configured work space failed. The dimensions of the work space are specified in the machine configuration.
<b>Collision: End of travel limits</b>	The attempt to move the tool carrier beyond the configured travel range failed. Check the specification of the travel range limits in the machine configuration.

<b>Collision: Please activate spindle</b>	The attempt to machine the workpart while the spindle is deactivated is not possible. Switch the spindle on prior to carrying out any tool movements.
<b>Colour configuration impossible in this graphic mode</b>	The colour display can only be specified or changed if the applicable colour display mode is part of the system configuration (Cf. Configuration Instructions).
<b>Compensation values cannot be processed</b>	Error message caused by various situations: a.) Recessing cycle: the tool width value in the current compensation register is too small. b.) Threading cycle: the quadrant specified in the current compensation register is not compatible with the location of the thread. c.) RADIUSING/chamfering cycle: the quadrant specified in the current compensation register is not compatible with the actual position of the tool in relation to the cycle end point. Check the tool storage and the compensation value register.
<b>Configuration aborted because of invalid password entry</b>	The configuration program has been protected by a password. To proceed, the correct password must be entered.
<b>Configuration already exists</b>	The specified file name already exists as a configuration file. Specify a new file name.
<b>Configuration could not be actualized. Press any key!</b>	The actual selection of configuration files could not be stored in the main selection menu of the MTS programs. Check (on the DOS level) whether the files "DNC.KNF" and "FNC.KNF" are available (and unlocked) in the directory in which the CNC Simulator has been installed. If the files are locked change the file attributes. You may also import the selected configuration files by activating the configuration management (cf. Configuration Instructions) and subsequently repeat your configuration selection.
<b>Configuration file not found : Data is not saved !</b>	Error in the colour configuration of the CNC Simulator. Check whether the selected configuration files of the turning/milling control are available and properly embedded (cf. Configuration Instructions). When in doubt contact MTS.
<b>Contour entity invalid or incomputable</b>	The programmed values are not logical, therefore the contour segment could not be computed. Examples: a.) Parallel non-identical straight lines have no intersection point. b.) The rounding radii of two non-intersecting circles is less than half the distance between the two circles. c.) The value of a circle radius or of the length of a line is "extremely small" (almost zero). Consequently internal computing is not possible. Check the programmed values, including those of the preceding NC blocks.
<b>Contour not available</b>	In a roughing multipass cycle no contour has been programmed between the invocation and termination of the fixed cycle. Program a contour specifying the roughing passes.
<b>Contour point cannot be processed</b>	The maximum number of contour points allowed in a roughing multipass cycle has been exceeded. To increase the number of the contour points allowed in cycle programming activate the configuration programme (cf. Configuration Instructions).

<b>Contour points arranged in wrong order (ESC to Abort)</b>	The contour points in a roughing cycle have been programmed in sequential order opposite to the machining direction (e.g. from the upper left to the lower right with postaxial external roughing). Pressing <ESC> cancels program execution. Pressing any other key starts the automatic rearrangement of the contour points (referring to the above example, the sequential order should be changed to "lower right to upper left").
<b>Contour string is not finished</b>	No standard or contouring command has been programmed within an open contour string. Conclude this contour string before you proceed to program further G commands.
<b>CRC-Check error !</b>	Hardware error: Diskette reading error: Either the diskette or the floppy disk drive is defective.
<b>Cycle address codes incomplete</b>	An obligatory address letter has not been programmed in this cycle. Program a complete cycle as described in the applicable "Programmer's Guide".
<b>Cycle starting point cannot be processed</b>	The current tool position cannot not processed with the programmed cycle: a) The current tool location is too close to the programmed cycle end point. Move the tool to a better location before calling the cycle. b) The tool quadrant is not consistent with the location of the cycle end point relative to the current position. Check the quadrant statement in the current compensation register or select an appropriate tool.
<b>-D-</b>	
<b>Degression or radius compensation invalid</b>	Cutter radius compensation is not possible for the active tool: a) Check whether the desired tool is in working position. b) Check whether it is possible to go without collision to the specified starting point of the cutting operation c) Check whether the 'go to' statement is compatible with the current tool (frequently the radius given in the quadrant/semicircle for 'go to' is not "compatible" with the tool geometry). See also "tool tip radius larger than programmed radius"
<b>Depth of roughness could not be displayed (short of memory)</b>	The available working memory is not sufficient to compute the surface peak-to-valley heights. Terminate the current processing and restart the Simulator. Make sure that driver files etc. which are not currently needed do not occupy memory space.
<b>Depth of roughness incomplete</b>	See "Surface roughness data incomplete !"
<b>Device not ready !</b>	Hardware error: Check whether the device is ready to receive and properly connected and set up as specified in the hardware configuration.
<b>Different planes have been selected in the system and in the setup form</b>	The CNC milling Simulator can be configured to simulate a 3-axis milling machine with vertical or horizontal spindle position and different axes designations. Either change the selection of planes in the configuration or make change the entries in the setup sheet.
<b>Directory does not exist / already exists</b>	An already existing or a non-existent directory has been selected. Check the entry and consider the MS DOS conventions.
<b>Division by zero</b>	The programmed NC block has effected the system attempt to execute a division by zero or a very small value. Check the plausibility of the NC block or slightly change the programmed values.

<b>Dongle not found</b>	The software ID module (dongle) could not be found during startup. Please check whether the port for the dongle has been correctly selected in the configuration. If the dongle is attached to a port where an external device (e.g. printer) is connected, this must be either switched off or set to the "on line" mode (depending on the type of printer).
<b>Double entry of a part geometry!</b>	Workpart geometry data has been specified several times in the setup sheet. Delete double entries, or select automatic generation of a setup sheet (after pre-machining).
<b>-E-</b>	
<b>Error calculating the depth of roughness</b>	Mathematical error during calculating the surface roughness. This may occur if the tool used in the machining is not available any more. To guarantee the correct calculation of the peak-to-valley height, all working tools must be available in the tool manager.
<b>Error in input/ output unit</b>	Hardware error: please check the device in question.
<b>Error in field operation. Press any key...</b>	An error during field operation. Check whether the NC program is error-free. When in doubt, first save the results of your session, then repeat the operation.
<b>Error in graphic contouring Press any key</b>	The Workshop Oriented Programming cannot be loaded. Save your work results and start the Simulator again. If this error occurs again and again, incorrect data structure may be the cause: Check the working sequence of your hardware and try re-installation of the MTS system.
<b>Error in nested routine</b>	Error in the turning tools management: Please check the memory space available (see "Memory overflow"). When in doubt, contact MTS. You can also try to re-install the system. Note: Newly defined tools, clamping devices etc. get lost during re-installation!
<b>Error in subsequent mask</b>	An input mask could not be loaded. Save your results and start the system anew. Should this error occur again, check whether your computer functions properly; if necessary repeat the installation of the CNC Simulator in your system.
<b>Error in working memory!</b>	General hardware error: Please make sure that your computer functions properly, for questions consult your dealer.
<b>Error on floppy !</b>	Hardware error: please check the selected drive and make sure that the diskette has been properly inserted.
<b>Error reading Simulator configuration</b>	A configuration file could not be read during initialization of the CNC Simulator. Check whether the configuration files are available in the subdirectory ".\KNF". If necessary, repeat the installation of your MTS system.
<b>Error while initializing graphic mode</b>	The selected graphic mode is not compatible with your hardware. Check the entries in the hardware configuration.
<b>Error while initializing keyboard</b>	The keyboard driver for the configured keyboard could not be loaded. Check whether the keyboard is properly connected and the configuration data are valid.

<b>Error while initialising the graphic tablet</b>	The graphics tablet is not properly connected or not compatible with hardware configuration. Please check entries in the hardware configuration and also make sure the required driver files are available and have been selected (Cf. Configuration Instructions).
<b>Error while initializing the NC Editor</b>	The Editor could not be loaded. Check the available memory space (see "Memory overflow") and make sure the control configuration is error-free. The corresponding format and syntax files must be available as well as the selected programming code. When in doubt rerun the system installation.
<b>Error while initializing the Simulator</b>	General error: The Simulator could not be initialized. Please check your hardware and re-install the MTS system on your computer.
<b>Error while initializing the status file !</b>	A software module that is part of the Workshop Oriented Programming could not be loaded. Save the results of your work session and restart the CNC Simulator. Should this error occur again, check the system functions of your computer; if necessary re-install the MTS software.
<b>Error while initializing tools</b>	The configuration of the turret/magazine could not be set up while loading the system. Check whether the tool files are available in the the subdirectories ".\DWZ" or ".\FWZ". If necessary, repeat the system installation. Note: newly defined tools get lost during re-installation!
<b>Error while opening data input file</b>	Internal error during WOP processing. Conclude the current work session and save the results, then start the system again. Should this error occur again, the data structure of Workshop Oriented Programming may be the reason. Make sure that your computer functions correctly, then re-install the software.
<b>Error while opening / closing the library</b>	The tool file could not be read or saved. Please check whether the tool files are available in the subdirectories ".\DWZ" and ".\FWZ". If necessary, repeat system installation. Note: newly defined tools get lost during re-installation!
<b>Error while printing or writing</b>	The screen dump (the print file) could not be generated. Check the printer connection as well as the file name and printer configuration. When in doubt, contact MTS.
<b>Error while reading !</b>	General hardware error: reading operation failed. Check eventual problems during accessing the hard disk or other drives.
<b>Error while reading language file</b>	The language file could not be read-in during initialization of the Simulator. Check the entry of the language file in the configuration and repeat conversion if necessary. Note, that sufficient memory space must be available on your hard disk for this operation (cf. Configuration Instructions, Language Files).
<b>Error while reading programming code file</b>	The programming code file specified in the configuration could not be read. Check the entries in the control configuration. When the "Free Definition Programming Code" is applied, the corresponding files with extensions ".DPS" or "FPS" must be available. If necessary, modify or replace the assignment file in the configuration (cf. also the Instructions for the Free Definition Programming Code).
<b>Error while searching !</b>	An error during search operation. Make sure the search string and the NC program have been specified correctly. If necessary, save your results and repeat the procedure.



<b>Error while writing !</b>	General hardware error: this message is displayed each time a write operation could not be executed (error while writing to the buffer file). Make sure your storage media are not defective, that the hard disk or other drives can be accessed error-free and that sufficient storage capacity is available.
<b>Error while writing input data</b>	See "Error while opening data input file"
<b>Error with consistency or data storage</b>	See "Internal inconsistency" and "Internal data error"
<b>Error: Closing bracket not found</b>	The character "]" is missing in the calculation program. Insert the missing character in the NC program (please note that calculations can only be made in the Free Format Mode).
<b>Error: Division by 0</b>	See "Division by zero"
<b>Error: Incorrect syntax / invalid value</b>	There is a syntax error in the current NC block or one of the address values violates the defined value range. Change the NC block (see also Ch. 7 of this manual)
<b>Error: Invalid argument of an operator</b>	A non-permissible value (such as root of a negative number) within programmed calculation in the NC program has not been accepted. Check the NC block and verify parameter assignments, if necessary. (Please note that arithmetical operations can only be programmed in the Free Format Mode.)
<b>Error: Invalid parameter type</b>	Either no number (or, with recurring parameters, no following character/word) has been specified after a programmed parameter letter (or parameter word, depending on the control configuration) or the parameter number refers to a parameter not yet specified. Check the correctness of the entries next to the parameter letter (cf. the Configuration Instructions).
<b>Error: Invalid parameter- or register-index</b>	The parameter number refers to an invalid or not yet specified parameter. Check the correctness of the entries next to the parameter letter (cf. the Configuration Instructions).
<b>Error: Memory overflow</b>	The available memory capacity is not enough for the calculation. Conclude the current processing and restart the CNC Simulator. Check that driver files etc. not currently needed do not occupy memory space.
<b>Error: NC-block too long</b>	The allowed maximum length of the NC block has been exceeded when replacing a text parameter. Check parameter definition or edit the NC block.
<b>Error: Overflow</b>	The value resulting from an arithmetical operation is too large (e.g. with trigonometrical functions). Check the NC block and parameter assignments, if necessary. (Please note that arithmetical operations can only be programmed in the Free Format Mode.)
<b>Error: Parameter exchange impossible</b>	Either no number (or, with recurring parameters, no following character/word) has been specified after a programmed parameter letter (or parameter word, depending on the control configuration) or the parameter number refers to a parameter not specified yet. Please check that only valid entries have been made next to the parameter letter (cf. the Configuration Instructions).

**-F-**

<b>Face driver and spindle data inconsistent</b>	See "Chuck and spindle data inconsistent"
<b>Face driver cannot be loosened</b>	Error during setup, workpart changing, re-chucking ... After the face driver has been deactivated, the clamping elements cannot be positioned in a consistent way. Check the combination of clamping devices, change the clamping if necessary.
<b>Fatal error while reading status file / workpart</b>	See "Error while reading"
<b>Feed adjustment or rounding incorrect</b>	The infeed depth or filleting programmed with this cycle cannot be executed with the active tool. a) Make sure the right tool is engaged. b) Check whether the currently active tool can be applied to machine the desired fillet (frequently the specified radius is "not compatible" with the tool geometry). c) Check whether the feed adjustment of the active tool can be executed as programmed. See also "Contour point cannot be processed"
<b>Field operations not possible. Press any key...</b>	It is not possible to copy or move a block to itself. If you specify a target position within the selected block, the appropriate error message is displayed.
<b>File could not be opened!</b>	No workpart data is stored under the specified file name in the subdirectory. Check the listing of available workpart files in the selection window and make sure the name is spelled correctly.
<b>File does already/not exist!</b>	The specified file name already exists or does not exist. Check your entry or enter some other name.
<b>File reading error or invalid file format</b>	The selected file could not be loaded. Check your file information or the file format (on the DOS level).
<b>Fixed cycle not available</b>	No machining cycle has been programmed with this cycle invocation. Insert the cycle data in the corresponding program line, prior to the invocation.
<b>Floppy Disc write protected !</b>	The target disk is locked. Unlock the disk or insert an other.

**-G-**

<b>General error !</b>	General hardware error: please check that your computer works properly; if in doubt, consult your dealer.
<b>Generated line too long, line has been split</b>	Error during format conversion of a programme file: according to the MTS format an NC line of more than 80 characters is generated. The program cannot be run. Use the Free Format Mode to make the relevant corrections in the NC line, either before or after conversion.

**-H-**

<b>Help screen not available! Press any key</b>	See "Help screen not found"
---	-----------------------------

<b>Help screen not found</b>	A help screen of the specified name is not available. Check your entry. You may also select the desired help screen from the help screen list.
<b>- -</b>	
<b>I - Ignore</b>	MS DOS prompt, caused by various reasons: When < > is pressed, the system ignores the current error and makes a new try to execute the function.
<b>Impossible to change the jaws</b>	The selected clamping devices are not compatible with the current machine configuration. Please select some other jaws (or define smaller dimensions of the blank/workpart). Alternatively you can also edit the configuration file.
<b>Impossible to continue in undefined contour or cycle</b>	This error message is shown if you specify a part of an NC program to be executed and its upper limit is within an open string of contour segments or a cycle. If so, further processing is not possible. Re-specify the program range.
<b>Impossible to flip the part</b>	There is no logical way to position the re-clamped workpart. Check and change if necessary the selected clamping fixtures and the tool position.
<b>Inconsistency</b>	See "Internal inconsistency"
<b>Inconsistency in calculation</b>	See "Internal inconsistency"
<b>Inconsistent termination of a sequence</b>	See "Unexpected program end"
<b>Index of NC cross-reference file is invalid</b>	This error message appears if you apply some other than the MTS programming code: The cross-reference file contains invalid data. Use the MTS editing program to check the cross-reference file (cf. instructions for the Free Definition Programming Code). Carefully check that no undefined references are included on the foreign syntax side.
<b>Internal data error</b>	An error has occurred during a CNC Simulator run. Save the results of your work session and restart the MTS system. Should this error recur, check the system functions of your computer, then try to proceed by repeating the installation of the CNC Simulator. In case of recurring problems, contact us for help.
<b>Internal error: Part information incomplete or invalid</b>	The workpart geometry could not be computed (beginning or end point not defined or values too small). Try again to machine the workpart before you call up the 3-D view function. Should this error occur again, check the system functions of your computer, then try to proceed by repeating the installation of the CNC Simulator. In case of recurring problems, contact us for help.
<b>Internal inconsistency</b>	Inconsistency of data. Check the movement command or the programmed values.
<b>Intersection cannot be computed</b>	See "Contour point cannot be processed"
<b>Invalid centre tip chucking</b>	Clamping between centres as specified in the setup sheet is not possible. Check your entries in respect to clamping devices and workpart geometry.

<b>Invalid contour entity</b>	The contour to be roughed in a multipass cycle contains a loop. Delete the loop (e.g. by block skipping).
<b>Invalid initialization!</b>	Error message caused by various reasons: a.) It is not possible to generate an NC program. The number of file locators (pointers) allocated to this program may be too small. Check the entry at FILES=xx in the CONFIG.SYS file which is read-in after startup (at the DOS level). Change the value statement to FILES=20 and reboot the system. b.) A roughing or contouring cycle (Simulation of turning or dressing) cannot be initialized. Invalid modification of a configuration file may be the cause. Check your entry at the menu item "roughing cycles/number of contour points" in the Configuration Program and enter an acceptable value (Cf. Configuration Instructions). c.) Resetting of contour segments is not possible. The current working memory of your computer may be insufficient. You should not continue the program run. You can add available memory space e.g. by deleting all driver files not currently needed.
<b>Invalid inner diameter</b>	It is not acceptable that the internal diameter of the blank is longer than the external one.
<b>Invalid subprogram name</b>	No valid subprogram name could be found. Check the subprogram invocation. According to the MTS format, next to the invocation address a number of maximum six digits must be specified. In the Free Format mode a character string of maximum seven alphanumerical characters is acceptable.
<b>Invalid tool compensation</b>	(Input) error with tool offset values. Check your entries. If in doubt, refer to the magazine configuration or the tool file manager for information on the applicable tool geometry.
<b>Invalid turret number</b>	The turret position (or magazine location) specified in the setup sheet is not available. Check the machine configuration, if necessary edit the acceptable number of tool positions.
<b>Invalid value of an address</b>	The values entered next to the address letter do not conform to the MTS programming code. In case of departure commands, machine commands, tool change commands and block skipping only integer values may be entered next to the address letter. Subprogram names and character strings next to a parameter assignment are exceptions. Check the syntax of the NC block in question. If you do not apply the MTS code, use the editor of the Free Definition Programming Code to check the cross-reference file applied for conversion of this NC file (or NC files). Cf. the Instructions for the Free Definition Programming Code.
<b>?ISO-blocks for blanks contain different parameters than G,X,Z,I,K,M</b>	For the definition of workpart geometry in the setup sheet, only DIN blocks are permitted. Please change the commands in the setup sheet, or choose automatic setup sheet generation (after pre-machining).
<b>-J-</b>	
<b>Jaws cannot be opened</b>	Opening the jaws would cause a collision. Move the tool carrier to increase the distance for opening the clamping device.
<b>Jaws invalid, maximum blank length ...</b>	An internal check of blank length and clamping fixture geometry is automatically carried out. Some other clamping device must be selected.

**-L-****Lathe centre and spindle data inconsistent**

The entered values relating to spindle and lathe centre geometry are invalid or contradictory.  
Check the entries and, if in doubt, refer to the corresponding explanations given in this manual.

**Line segment incomputable**

See "Contour point cannot be processed"

**Line segment not properly defined**

Unacceptable values have been entered when programming a hole pattern on a straight line entity.  
Check the conformity of the address values.

**-M-****Machine function cannot be executed**

More than four machine commands have been programmed within the same NC block.  
Insert the machine commands in different blocks.

**Mathematical error**

See "Contour point cannot be processed"

**Maximum /minimum blank dimensions exceeded**

The values specified for blank geometry exceed the defined range of acceptable values.  
Modify your entries or check the machine configuration.

**Memory overflow**

The memory space required by the software could not be located.  
Please use the commands "CHKDSK" or "MEM" at DOS level to check the working storage capacity of your computer. You may obtain additional memory space by deleting driver files etc. that are not needed currently.  
In some cases the problem can be eliminated by editing the NC block which caused the prompt. However, it is recommended not to continue the program run after this error message has been displayed.

**'MiniMask': Zeropointer!**

See "Error while initializing the status file"

**Movement not possible**

The movement effected by this NC block would result in one coordinate exceeding the max value of "999.999".  
Please edit the NC block so that the movement does not violate the configured travel limits.

**-N-****NC-Block not permitted in this situation**

Error message caused by various reasons:  
a.) machine travel limits function has been programmed although no roughing cycle is selected. Please delete the limits function.  
b.) a roughing cycle has been programmed while a roughing cycle is already selected.  
Please delete the second cycle invocation.  
c.) a single pass cycle of less than two contour points has been programmed while a roughing cycle with undefined starting point is selected.  
Please program the contour beginning first, then check the values of the single pass.

**NC-block not complete**

No start block number has been programmed for a jump instruction or routine.  
Please program the start block number.

<b>NC-command or coordinate movement could not be established</b>	<p>Error message caused by various reasons:</p> <p>a.) The NC block contains a standard movement command without coordinate value. You can either program the coordinate or eliminate the movement command.</p> <p>b.) The departure command is not available (because no modal command is configured, cf. "modal commands") In such a case, program a command or edit the configuration.</p>
<b>NC program not found</b>	See "Program does not exist / not available"
<b>No help screen available !</b>	See also "Help screen not found"
<b>No help screen available !</b>	<p>The auxiliary graphics for tool and clamping fixture management have either not been installed in your system or their data structure is defective. Try to re-install the MTS system in your computer.</p>
<b>No intersection with the previous contour entity</b>	<p>A contour element could not be computed because the programmed values are inconsistent. Examples:</p> <p>a) Parallel non-identical straight lines have no intersection point. b) The rounding radii of two non-intersecting circles is less than half the distance between the two circles. c) The value of a circle radius or of the length of a line is "extremely small" (almost zero). This renders internal computation impossible. Please verify the programmed values, including those of preceding NC blocks.</p>
<b>No more memory available !</b>	<p>No working memory available (see "Memory overflow"). Terminate your current work session and restart the CNC Simulator. Make sure that driver files etc. not currently needed do not occupy working memory space.</p>
<b>No printer configured! Print job aborted</b>	Please change the setting in the hardware configuration.
<b>No source or converted program selected</b>	<p>Error during format conversion of a program file. To generate a NC program according to the MTS programming code, the file names of the (existing) source program and the target (object) program (to be generated) must be specified prior to the conversion.</p>
<b>No such setup file available</b>	<p>The program for automatic setup of the Simulator, as specified in the configuration, is not available. Check your configuration entries or generate the Setup Sheet.</p>
<b>No workpart has been selected!</b>	<p>If no status file or setup sheet is read in during the startup procedure of the CNC Simulator, then the simulated machine must be completely set up in the manual mode. Please note that you need to specify a blank or insert a pre-produced workpart before you can change clamping devices.</p>
<b>Note: After touching the workpart, return to a suitable starting point !</b>	When working in Teach-In mode, after having specified the zero point by touching the workpart, take the tool back to its starting position before you conclude the submenu.
<b>Not enough memory available !</b>	See "Memory overflow"

<b>Not enough memory to list all</b>	There is not enough free memory space to display the complete list of tools or clamping fixtures. Terminate your current session and restart the CNC Simulator. Make sure that driver files etc. not currently needed do not occupy memory space.
<b>Null device error</b>	Hardware error: please check your hardware; if in doubt, consult your dealer.
<b>-O-</b>	
<b>One of the tools is not available</b>	One of the tools specified in the setup sheet is not available (any more). Define the desired tool or specify some other tool in the setup sheet.
<b>Only G00/01/02/03/0/1/2/3 accepted!</b>	For the definition of workpart geometry in the setup sheet, only DIN blocks are accepted. Please edit the commands in the setup sheet, or choose automatic setup sheet generation (after pre-machining).
<b>Out of paper !</b>	Your printer is out of paper
<b>-P-</b>	
<b>Part cannot be removed</b>	Re-chucking the workpart would lead to a collision. Please move the tool carrier to a position that enables collision-free re-clamping of the workpart.
<b>Part contour contains not just one single contour</b>	Workpart geometry data describing a preaxial/postaxial closed contour can be specified in the setup sheet. Only one contour definition is allowed. Edit your geometry entries or select automatic generation of the setup sheet after the first machining process of the inserted part has been carried out.
<b>Part contour contains only three different points!</b>	Workpart geometry data describing a preaxial/postaxial closed contour can be specified in the setup sheet. At least four corner points must be defined. Edit your geometry entries or select automatic generation of the setup sheet after the first machining process of the inserted part has been carried out.
<b>Part contour crosses rotation axis!</b>	Workpart geometry data describing a preaxial/postaxial closed contour can be specified in the setup sheet. The contour line must not cross the rotation axis (centre line). Edit your geometry entries or select automatic generation of the setup sheet after the first machining process of the inserted part has been carried out.
<b>Part does not exist !</b>	No workpart is stored under the specified file name. Check the file name or select an existing file from the list in the workpart manager.
<b>Part geometry too complex !</b>	This workpart cannot be represented in 3-D view because of too complex geometry information. This may apply to certain stored parts as well. Please check the available free memory space in your system (cf. "Memory overflow"). If the applicable workpart has been generated with the MTS system check that driver files or programs which are currently not used do not occupy memory space. Make sure that only the currently needed program and driver files are being loaded, then reboot the system. (See also "Error while reading / writing")
<b>Part is not clamped</b>	The workpart is not (or cannot be) clamped with the selected clamping device (examples: the driver teeth do not engage in the workpart or the tailstock/quill has not been positioned). Please select an other clamping fixture or specify an appropriate clamping method.
<b>Part updating impossible</b>	Data inconsistency during contour computing (rounding error). Send the applied program to MTS or contact us per phone.

<b>Part was saved in a graphic mode different from the active mode</b>	The system failed to read in a stored workpart which was generated in a different graphic mode. Only parts created in the currently active graphic mode can be loaded.
<b>Plane selection in the setup form inconsistent with the current control system</b>	See "Different planes have been selected..."
<b>Please program the angle</b>	The value "A" for the angle to the X axis has not been specified in the geometry definition by polar coordinates. Enter the missing data into the NC block.
<b>Please program the distance from the polar centre</b>	The value "B" for the distance to the polar centre has not been specified in the geometry definition by polar coordinates. Enter the missing data into the NC block.
<b>Please enter program name first</b>	To launch an NC program in the Automatic Mode, first enter the desired program name.
<b>Please move to the reference point</b>	Each time when starting the CNC Simulator the reference point needs to be approached to set the position measuring systems to the machine datum. This is part of the setup procedure and is required for running NC programs. (For information on the option of Automatic Setup, cf. the Configuration Instructions.)
<b>Please remove the tool</b>	Changing the blank would result in a collision with the working tool. Move the spindle head to a position where it does not collide with the new blank to be inserted.
<b>Please remove tailstock before loading a saved part !</b>	A workpart cannot be loaded or stored until the tailstock has been removed.
<b>Please select tailstock/ centre sleeve in the clamping menu</b>	The tailstock cannot be positioned until one of the clamping methods "with tailstock" in the clamping fixture menu has been selected.
<b>Please turn the spindle off</b>	Changing tool or blank is not possible if the spindle is still active. Turn the spindle off first. See also : "Warning: Spindle still activated"
<b>Point on circle cannot be computed</b>	See "Contour point cannot be processed"
<b>Positioning / removing of the tailstock not possible!</b>	After reclamping or changing the blank, the tailstock cannot be set to the initial position again, because this would result in a collision when the "G28" command is executed. Please check workpart geometry, tool position etc.
<b>Precision loss</b>	Error during format conversion of a program file. According to the MTS code the indicated value would appear in the target program. You can either confirm this value conversion (with <F1>) or press <F8> to cancel the conversion.
<b>Printer not responding!</b>	Check that the printer is switched on and ready to receive. Or check the printer port selection in the hardware configuration of the CNC Simulator (Cf. Configuration Instructions).



<b>Printing impossible in this graphic mode</b>	To print out the graphic display (screen dump) select appropriate graphics mode. (Cf. Configuration Instructions)
<b>Program not available</b>	No program file is available in the specified search path. Please check the directory and file names in the NC program management. Another possible cause: the program selection path has been edited (cf. the Configuration Instructions).
<b>Program termination not possible in this situation</b>	A program end statement has been programmed in an open contour string, in a subprogram, in a routine, in a multipass roughing cycle or during the operation of the tool nose compensation. Note that the program end command can only be stated at the actual point of program termination.
<b>[Program] ... could not be installed / deleted / printed</b>	Check the file identification entries. File extensions and file names must conform to the MS DOS conventions.
<b>Program could not be started (error ...)</b>	An error during launching an MTS program. Check the system functions of your computer and rerun the installation of the CNC Simulator.
<b>Program does not exist</b>	Error message of the PAL Marking Program: No NC Program is stored under the filename you have entered. Use the file selection window for an overview of available programs.
<b>Program file already exists</b>	The specified file name has already been used for a program. Either overwrite the existing program or enter a new file name.
<b>[Program] ... not found</b>	See "[Program] ... could not be installed "
<b>Program ... not found! Press any key...</b>	Internal error while starting an MTS Program.
<b>Programmed halt impossible in this situation</b>	A program halt has been programmed in an open contour string, in a multipass roughing cycle or during the operation of the tool nose compensation. A program halt can only be programmed at a point where the above procedures have been concluded.
<b>Programmed value not plausible</b>	One of the address letters of the current cycle is not acceptable. Check the value statements assigned to the address letters.
<b>-R-</b>	
<b>R - Repeat</b>	MS DOS prompt caused by several reasons. Type <R> for another try to execute the previous operation.
<b>Reading error</b>	See also "Error while initializing the status file !"
<b>Reading/Writing Error !</b>	A file cannot be read or stored. Check the specified file name; a file whose data structure is incompatible with the present program function may have been stored by mistake in the respective directory and later selected; also a data error may have occurred. Storage problems may be due to the storage medium being defective or full, also a hardware problem may be the cause.
<b>Reversible tip not yet available</b>	This type of reversible tip is not implemented in the current version of the Simulator.

<b>Rounding error during time calculation</b>	The computer clock device works with a certain tolerance range; a rounding error within this tolerance has resulted in inconsistency. Restart the NC program or repeat the execution of the NC block.
<b>Rounding impossible</b>	See "Contour point cannot be processed"
<b>Rounding radius or chamfer too small</b>	During parameter programming a non-permissible value has been entered. The length of a chamfer or the filleting radius must be larger than 0.0005 mm.
<b>-S-</b>	
<b>Sector not found !</b>	Hardware error: during disk access a sector could not be found. Either the disk or the floppy drive is defective.
<b>Set-up of help screen inconsistent</b>	The data structure of the help screen file is defective. Please re-install the help screen files in your computer.
<b>Short of memory: the selected program cannot be loaded</b>	See "Memory overflow"
<b>Simulation cannot begin from within a subprogram or a cycle</b>	A jump instruction or a program routine has been programmed within a subroutine. Jump instructions and program routines can only be programmed in the main program. You may try to block skipping.
<b>Source- and target program must have different names !</b>	Error during format conversion of a program file. To generate an NC program conforming to the MTS programming code, the (existing) source program and the target program (to be created) must have different names.
<b>Space problem : No intersections available !</b>	Available memory space is not enough to compute the intersection points. In most cases you will be able to continue processing without computing the intersections. However, it is advisable to save the results of your session and check the available memory space (see <i>a/so</i> "No more memory available").
<b>Spindle sleeve did not catch !</b>	To clamp the workpart, the tailstock must be positioned closer to the workpart.
<b>Stack overflow</b>	This error message may have various origins: The NC program contains more than five jump instructions; a jump instruction causes an infinite loop; the NC program contains more than fifteen routines, subprogram invocations or jump instructions. Reduce the nesting in your NC programming and pay special attention with jump instructions prior to the current NC line.
<b>Status could not be read</b>	The required status file could not be loaded while initializing the simulator. Skip the status file in the initialization process (by appropriately editing the configuration) and subsequently store the status again.
<b>Status could not be saved</b>	The current processing status could not be saved. Check the file identification. Other causes of this error may be: The storage medium is full or defective, or a hardware error has occurred (see <i>a/so</i> "Error while reading/writing")
<b>Step jaw and spindle data inconsistent</b>	See "Chuck and spindle data inconsistent"

<b>Subroutine not available</b>	<p>When searching the configured path (cf. Configuration Instructions) the simulator system could not find the specified subprogram. If the subprogram name consists only of digits (e.g. G22 U000123) and the number of the subprogram repetitions has not been specified in the corresponding configuration file, the system searches the program path for the specified routine, first with and then without the zeros as a search criterion (e.g. U000123 in the first pass, then U123). Also check the subprogram identification letter and the default extensions as specified in the configuration.</p> <p>Example: If the subroutine identification is "%" and the default extension is ".din", the program path search is successful for subprograms "%000123.din" and "%123.din".</p> <p>To avoid this kind of problem a subroutine invocation should be programmed with reference to a routine which is in accordance with the configuration settings.</p>
<b>Subroutine nesting limited</b>	<p>Too many subroutine invocations have been nested. For the maximum nesting depth check the program configuration (cf. Configuration Instructions). Edit, if necessary, nesting depth in the configuration. Up to 11 nested routines can be specified in the configuration. Consider this during programming.</p> <p>See also "Error in nested routine"</p>
<b>Surface roughness data incomplete !</b>	<p>There is not enough memory space available to complete the peak-to-valley heights computation (see also "Calculation of an entity suspended").</p>
<b>-T-</b>	
<b>Tailstock and blank are overlapping!</b>	<p>According to the present work space configuration, the workpart geometry as specified in the setup sheet is not compatible with the selected tailstock. Please amend either the setup sheet or the machine configuration.</p>
<b>Tailstock not available!</b>	<p>The machine configuration specifies whether the simulated machine tool is equipped with a tailstock. It also includes the information on tailstock geometry, if applicable.</p> <p>Check the settings in your machine configuration.</p>
<b>Tailstock or chuck not defined.</b>	<p>See "Tailstock not available"</p>
<b>Tangential transition incomputable</b>	<p>See "Contour point cannot be processed"</p>
<b>The converted block is too long</b>	<p>Conversion of an NC block would result in a block exceeding the acceptable block length according to the MTS programming code. This error message appears only when working with foreign control codes.</p> <p>Please check the current cross-reference file (cf. Instructions for the Free Definition Programming Code). Reduce the number of addresses per block, e.g. by programming optional words or standard conversion words in subsequent blocks.</p>
<b>The face driver / lathe centre ... is not available in the chuck management</b>	<p>The face driver / lathe centre specified in the setup sheet is not/no longer available in the clamping fixture manager.</p> <p>Modify the setup sheet or define a new clamping device to be stored under this name.</p>
<b>The first line of the geometry accepts only G, X and Z as parameters.</b>	<p>In the setup sheet the first block for definition of the workpart geometry must be the DIN block "G01".</p> <p>Change the entries in the setup sheet or select automatic generation of a setup sheet (after pre-machining).</p>

<b>The first line of the geometry accepts only G0 and G1.</b>	In the setup sheet the first block for definition of the workpart geometry must be one of DIN blocks "G00" and "G01". Change the entries in the setup sheet or select automatic generation of a setup sheet (after pre-machining).
<b>The following tool compensation makes no sense: ...</b>	Values in this compensation value storage are not correct or logical. Check the entries in the setup sheet or activate automatic generation of a setup sheet.
<b>The following turret position is not available: ...</b>	A turret has a maximum of 16 tool positions, a magazine has a maximum of 99 tool pockets. Change the entries in the setup sheet or check the machine configuration.
<b>The format file was modified !</b>	The selected NC program is not based on the current format file or the checksum is invalid. Convert this program before starting to edit.
<b>The NC-program contains no format information!</b>	It is not possible to load an NC program to the NC Editor which has been generated or edited in a different system. Convert this program first or try to edit in the Free Format Mode.
<b>The NC-program does not match the format information!</b>	<i>See above</i> "The NC program contains no format information!"
<b>The NC-program was not created with the current format file !</b>	The selected NC program is not based on the current format file. Convert the program before starting to edit.
<b>The path to the NC-Programs could not be established</b>	This program selection path has not been configured. Check the program selection path stated in the configuration (cf. Configuration Instructions).
<b>The printer is busy ! Try again ?</b>	The printer is not ready to receive. Enter <Y> to try again, <N> to cancel the procedure.
<b>The printer is not ready!</b>	The printer does not respond. Make sure the printer is switched on and ready to receive.
<b>The reach of the sleeve tip is too small</b>	Change the input data regarding the reach of the sleeve (quill stroke) in the Configuration.
<b>The selected program arguments list is too large (environment)</b>	<i>See below</i> "The selected program was aborted with error ..."
<b>The selected program could not be started !</b>	An error has occurred during the launching of an MTS program. Terminate other applications before you start the CNC Simulator. Should this error occur again, check the system functions of your computer and repeat the installation of the CNC Simulator.
<b>The selected program file cannot be executed (format error)</b>	<i>See</i> "The selected program was aborted with error ..."
<b>The selected program is not available</b>	You have selected a program module that has not been installed in your system.

<b>The selected program was aborted with error ...</b>	Internal error while starting an MTS Program. Conclude all applications and start the program again. Should this error occur again, contact us for help. If you decide to install the MTS system anew, make sure to save the tools, clamping fixtures etc. that you have generated, to avoid them not being overwritten.
<b>The setup form syntax is not as expected!</b>	A setup sheet must be structured according to certain conventions. Correct the entries to conform to the syntax, or have the setup sheet generated automatically.
<b>There is a loop in the contour!</b>	The workpart geometry data specifying a preaxial/postaxial closed contour can be entered in the setup sheet. The defined contour segments are not allowed to intersect any other segment (contour loop). Change the geometry entries or select automatic generation of a setup sheet after the first machining process of the inserted part.
<b>There is no entity with a calculable depth of roughness !</b>	The surface roughness (peak-to-valley height) can only be calculated for machined parts.
<b>This combination of NC-Commands is not permissible</b>	See "Too many addresses in NC block..."
<b>This graphic mode is not supported</b>	The auxiliary graphics cannot be displayed in the selected graphic mode (cf. Configuration Instructions, Hardware)
<b>This program contains no address. Press any key...</b>	The NC program cannot be processed because it contains no address letters. Check the program on the DOS level or try to edit in the Free Format Mode.
<b>Too many addresses in NC-block / This combination of NC-Commands is not permissible</b>	Error messages due to various reasons: a) Multiple standard commands or contour string commands have been programmed within the same NC block. b) Program halt as well as program end have been programmed within the same NC block. c) A subprogram invocation as well as a routine have been programmed within the same NC block. d) Tool nose compensation has not been programmed in a separate NC block. e) In the Simulator for turning multipass roughing cycles have not been programmed in separate blocks. To eliminate these problems split the applicable NC block into several blocks.
<b>Tool adapter and spindle data inconsistent</b>	See "Chuck and spindle data inconsistent"
<b>Tool already exists</b>	Error message referring to the tool file management. The specified name has already been used for a tool. Give some other name to designate the tool.
<b>Tool and spindle data inconsistent</b>	See "Chuck and spindle data inconsistent"

<b>Tool change impossible</b>	<p>Error message due to various reasons:</p> <p>a) The programmed tool number exceeds the maximum number of tools in the turret (the latter being a default parameter which is specified in the configuration). Enter an acceptable tool number or edit the applicable entry in the configuration file (cf. the Configuration Instructions).</p> <p>b) Removing the active tool or inserting a new tool would cause a collision. Before changing the tool move the tool slide to a position allowing a collision-free tool change.</p>
<b>Tool tip radius larger than programmed radius</b>	<p>This error may occur while the tool nose radius compensation is active. The programmed NC block provides compensation of a circular arc whose radius is smaller than the tool nose radius.</p> <p>You must program an arc of larger radius or apply a different tool.</p>
<b>Tool too close to the jaw</b>	See "Tool too close to the part"
<b>Tool too close to the part</b>	<p>Error when inserting a new workpart from the workpart file.</p> <p>Please move the tool to a different position.</p>
<b>Tools for the following turret positions not available:</b>	<p>A tool that does not / no longer exist has been specified in the setup sheet.</p> <p>Please amend your entries in the setup sheet or define a new tool for this name.</p>
<b>Transformation in this situation impossible</b>	<p>A zero offset has been programmed in a multipass roughing cycle or concurrent with tool nose compensation.</p> <p>Deactivate tool nose compensation or conclude the cycle before you can program the zero shift.</p>
<b>Turret already exists</b>	See "Magazine already exists"
<b>Type of tool presently not implemented</b>	The selected type of tool is not implemented in the current version of the MTS system.
<b>-U-</b>	
<b>Unable to read status index</b>	<p>The status index contains information for segment contour programming. It is stored as the file "ZT" in the main directory of the Simulator.</p> <p>Make sure this file is available; also check your hardware. If necessary install the system anew.</p>
<b>Unanticipated end of program</b>	<p>Error message due to various reasons:</p> <p>a) The NC program is not terminated by a "program end command" (M02 or M03 congruent to the MTS programming code). Make sure the program is properly concluded.</p> <p>b) Start block number or end block number of a jump instruction or routine could not be found in the program.</p> <p>Change the programming of the block numbers.</p>
<b>Unidentified error!</b>	<p>An unidentified error during printing.</p> <p>Check whether the printer is properly connected and ready to receive. Also check the printer selection in the hardware configuration (cf. Configuration Instructions).</p>
<b>Unknown command !</b>	<p>Hardware error: the selected device does not support the actual application.</p> <p>Check your command entry and try again.</p>
<b>Unknown device !</b>	<p>Hardware error: system access to a peripheral device failed.</p> <p>Check the status of the selected device.</p>

<b>Unknown Floppy Disk format !</b>	Hardware error: the inserted disk has no MS DOS format. Check whether this disk is needed for application in another system. If this is not the case you may use the MS DOS "FORMAT" command to format the disk.
<b>Use of parameter ... invalid</b>	According to the MTS programming code it is not acceptable to insert an identification letter immediately after a parameter. Either interrupt the file conversion to change parameter assignation, or make the necessary corrections after the program has been converted.
<b>User directory is occupied</b>	Error message for network operation. The CNC Simulator cannot be started until the current processing is terminated.
<b>-V-</b>	
<b>Vice capacity too small for the workpart! Accept anyway?</b>	In the Simulator for milling the workpart may be turned 90° in a single step. Should the vice capacity prove insufficient for clamping the turned part, either rotate the part by another 90° or select an other vice.
<b>Vice not available in the chuck management</b>	The vice specified in the setup sheet is not/no longer available in the clamping fixture management. Either change the setup sheet or define a new vice to be stored under the given name.
<b>-W-</b>	
<b>Warning: Coolant still activated</b>	Please deactivate the coolant function before you proceed to change the blank or workpart.
<b>Warning: Different configuration files active!</b>	The currently selected configuration files differ from those specified in the setup sheet. Check whether the selected files are available; if necessary, modify the entries in the setup sheet.
<b>Warning: Failure in creating correct setup form!</b>	An error has occurred during the setup sheet generation. Check the current setup status and the NC program file name to which the setup sheet is to be assigned.
<b>Warning: Invocation of cycle before subprogram end or program section end</b>	The invocation of a roughing cycle has been programmed as part of a subprogram or routine. This message is meant to prompt the user to check eventual inconsistencies in the current program structure. Check the NC program on inconsistencies. Strike <ESC> to cancel processing, press any other key to continue.
<b>Warning: Spindle still activated</b>	Deactivate the spindle before you proceed to change the tool.
<b>Warning: You cannot start at this position</b>	This error message may appear when attempting to run an NC program, starting from a certain block number. If the starting block is part of an open contour string or a multipass roughing cycle, this may cause problems in the subsequent program run. Press <ESC> to interrupt the processing, then select a block number before or after the applicable contour string or multipass cycle.
<b>When turning between centres, tailstock/sleeve movements are not permissible!</b>	When clamping between centres, the tailstock and sleeve are not allowed to be deactivated in the NC program

<b>Window too small !</b>	The detail area to be zoomed has been specified too small: the area must be at least 8µmx by 6µm. Specify a larger window for enlargement.
<b>Withdraw spindle sleeve first !</b>	The spindle sleeve must be withdrawn before setting back the tailstock.
<b>WOP-shell not available with open contour</b>	Workshop Oriented Programming may only be invocated before beginning or after completing a contour definition.
<b>WOP-shell only available with MTS-Control</b>	The MTS programming code must be applied in Workshop Oriented Programming. A further requirement is the plane selection G17 in the configuration.
<b>Wrong buffer value</b>	Hardware error: internal buffer size statement is invalid. Reboot the system and check the system configuration: the "buffer statement" in the MS-DOS CONFIG.SYS file determines the number of hardware (disk/diskette) queue space segments in the working memory to be allocated to the buffer pool at system startup. To run a CNC Simulator this value should be set to 8 at minimum (Cf. your MS-DOS Operation Manual).
<b>Wrong chuck or chucking length</b>	Entries in the setup sheet referring to the clamping device and clamping depth are not consistent. Change your entries or select automatic generation of a setup sheet.
<b>Wrong password ! Changes are not permanent.</b>	In the program modules "Workshop-Oriented Programming" and "Measuring and Zooming" this interactive error prompt is displayed when, after concluding the editing of the configuration, a wrong password is entered. In such a case, the changes of the configuration are only considered for the current processing. After restarting the program module, these values are reset to the "previous state".
<b>Wrong sense of spindle rotation</b>	An attempt to machine the workpart with wrong sense of spindle rotation has failed. Change the sense of spindle rotation before executing the tool movement.
<b>WOP-shell only available with MTS-Control</b>	The MTS programming code must be applied in Workshop Oriented Programming. A further requirement is the plane selection G17 in the configuration.
<b>-Z-</b>	
<b>Note:</b>	Error messages are subject to modifications. If you are not able to find the message displayed in the prompt line in this list refer to the entries "Error: ..." or go through the list to find a similar situation.



## Register

3-D Views, 1-15, 6-25, 10-3

Turning, 10-5

Milling, 10-11

Menu, 10-9, 10-15

### -A-

Absolute Dimensioning, 9-13

Automatic Mode, 1-13, 6-1

automatic setup, 3-17

cancel, 6-13

interrupt/resume, 6-11, 6-17

menu structure, 6-3

screen layout, 6-5

start, 6-1, 6-9, 7-11, 8-17

Automatic Run, 6-11

### -B-

batch file, 2-3

blank, 4-9

change, 4-11, 4-15

definition, 4-9

dimensions, 4-13, 4-15

Block Group Handling, 7-25

### -C-

CAD/CAM system, 1-7, 12-25

cavalier perspective, 4-17

checksum, 7-7

clamping fixtures, 1-9, 1-11, 4-23, 4-55

change, 4-23, 4-27, 4-55

copy data, 4-47, 4-75

define, 4-41, 4-75

delete, 4-51, 4-81

Manager, 1-15, 4-27, 4-37, 4-71

modify, 4-49, 4-79

select, 4-25, 4-55

clamping, 3-19, 4-31, 4-67

define, 4-31

display, 4-17

jaw chuck, 4-31

tailstock, 4-33

CNC control, 7-3

CNC keyboard, 1-3, 1-7, 2-15, 2-22

CNC symbols, 1-7, 2-11

CNC template, 1-7

collision monitoring, 1-13

collision, 6-1, 6-13, 8-13

error message, 4-87, 4-127, 6-7, 8-3, 8-27

colour setting, 2-9, 9-35, 10-31, 12-5

assignation, 10-37, 10-40

palette, 10-35, 10-40

compensation values, 4-85, 4-105

register, 4-5, 4-87, 4-91, 4-105,  
5-13, 6-5

configuration, 2-3, 2-21, 3-5, 6-1, 9-21, 9-35

control, 2-1, 7-9, 7-35, 11-11

hardware, 1-7, 2-1, 7-13, 11-13

machine, 2-1, 3-19, 4-109, 4-119,  
4-127

selection, 2-5

contour element, 9-9, 12-7, 12-11, 12-17

chamfer, 12-7, 12-18

circular arc, 9-9, 12-19

contour point, 9-13, 12-18, 12-29,  
12-35

display, 12-5, 12-27, 12-31,  
12-35

fillet (rounding), 12-7, 12-18

linear path, 9-9, 12-18

multi-point strings, 12-21, 12-37

open contour, 12-21, 12-37

segment contour programming,  
7-1, 12-1

tangential transition, 12-11

control, 1-9, 1-11

conversion programme, 7-5

coolant, 1-13, 4-119, 8-23

cutter radius compensation (CRC), 8-11

cutting edge point, 4-105, 5-13

cutting point, 4-105, 5-13

cycle, 7-3, 8-11

### -D-

datum, 4-111

dialogue, 8-9

Interactive Programming, 8-13

Teach-In, 8-29

Workshop-Oriented Progr., 12-1

dimension reference point, 9-5, 9-13, 9-17

dimensioning framework, 9-13, 9-23

dimensioning text, 9-25

DIN, 2-5, 7-3, 12-1, 12-7

### -E-

edges display

clearance edges, 9-1, 10-33

visible edges, 10-33

editing functions, 7-18, 8-9

Editor, 1-13

entering angle, 5-7

exit

CNC Simulator, 2-22

MTS System, 2-3

### -F-

face driver, 4-25

- define, 4-25
- feed motion, 1-13, 4-119, 4-125, 6-5, 6-23, 8-23, 8-27
  - rapid traverse speed, 4-5
- file, 2-19
  - extension, 2-21, 7-1, 11-13
  - format, 7-5, 7-15
  - management, 1-15, 3-19, 7-11
  - open strings, 2-22
  - selection, 2-19
- finishing allowance, 4-107, 5-15
- format conversion, 2-3, 2-5, 7-5
- Free Format Mode, 7-9, 7-15, 7-17, 7-35
- function keys, 2-7

**-G-**

- G-commands, 7-3
- graphic tablet, 1-3, 1-7, 2-15, 2-22
- graphics adapter, 1-3
- graphics printout, 11-13
- graphics standard (video mode), 1-3, 10-35, 11-13

**-H-**

- hardcopy, 11-13
- hardware, 1-3
- helical interpolation, 12-5
- Help Function, 6-31, 7-31, 8-16
  - cancel, 11-9
  - start, 11-1

**-I-**

- image region (detail display), 10-29
- INCAD, 2-3, 7-5
- increment, 4-5, 4-119, 8-23, 8-30
- incremental dimensioning, 9-19
- infeed angle, 4-105
- input media, 1-7
- input routine, 2-17, 7-18
- installation, 2-1, 7-11, 11-11, 12-3
- Interactive Programming, 1-13, 6-1, 7-1, 8-1
  - dialogue, 8-9
  - end, 8-17
  - input, 8-9
  - interrupt, 8-11
  - screen layout, 8-3
  - start, 6-19, 8-5
- interface, 1-5, 1-7, 7-13
- interruption of simulation, 6-11
- intersection point, 9-13

**-L-**

- lathe centre, 4-25
  - define, 4-45
- lathe chuck 4-25
  - define, 4-45

- lathe, 1-9

**-M-**

- machine functions, 4-119, 7-3
- machine table, 4-123
- machine tool, 1-9, 4-111
  - functions, 1-7, 4-1
  - machining, 1-13
  - setup, 4-1
  - status, 4-5, 6-5
  - work space, 2-7, 3-1, 4-5, 4-111, 6-5
- machine zero, 4-111
- machining planes, 1-11
- machining simulation, 1-13, 6-3, 6-11, 8-11
- machining status, 3-19, 4-7
  - file, 2-19, 3-20
  - management, 3-20
- magazine, 1-11, 4-85, 4-99
  - tooling, 4-7, 4-85, 4-89, 4-99
- magnetic plate clamping, 4-55, 4-61
- Main Menu, 2-7, 3-1
  - functions, 3-3
- manual control, 4-125, 8-27
- measuring, 6-25, 9-1
  - configuration, 9-21
  - display, 9-3
  - element dimens., 9-9
  - incremental, 9-13, 9-19
  - invoke, 9-1
  - point dimens., 9-13
- menu operation, 2-1, 2-11
  - disable functions, 2-15
- menu, 2-7
- military perspective, 4-17
- milling machine, 1-11
- milling tools, 1-11, 5-27
  - parameters, 5-29
  - selection, 5-35
  - tool adapter, 5-41
- modal commands, 6-3, 6-5, 12-18
- modular clamping, 4-55, 4-63

**-N-**

- NC block, 7-15, 12-11, 12-33
  - block number, 6-15, 7-9, 7-29
  - display, 2-19, 6-7, 7-15, 8-3, 10-23, 12-5
  - format, 7-15
  - special char., 7-19
- NC Editor, 2-19, 3-7, 7-9, 7-15, 8-16
  - block group handling, 7-25
  - display, 2-19, 7-15
  - editing range, 7-27
  - exit, 7-11, 7-33

functions, 2-19, 7-17  
 invoke, 7-9  
 Search & List, 7-31  
 NC programme, 1-13, 7-9, 8-7  
   convert, 7-5  
   delete, 7-11  
   edit, 7-9, 7-15, 8-9, 8-19, 12-17, 12-25  
   file, 2-19  
   link, 7-23  
   Manager, 1-15, 7-11  
   print, 7-11  
   range, 6-15, 8-7  
   renumber, 7-29  
   save, 7-33, 8-16, 12-31  
   select, 2-21, 6-3, 7-7, 7-12  
   Setup Sheet, 3-15  
   test, 1-13, 6-1, 8-3, 8-13  
 NC programming, 1-13, 2-17, 6-19, 7-1, 8-1  
 NC word, 7-9  
   addresses, 7-15, 12-18  
   value range, 7-15, 7-33  
 network, 1-5  
 nominal contour, 8-21  
  
**-O-**  
  
 office perspective, 4-17  
 operating system, 1-5, 2-1, 2-21, 2-22, 7-12  
 operation modes, 1-13  
 override, 4-125, 6-23  
  
**-P-**  
  
 PAL Marking Programme, 2-3  
 PAL Programming Code, 2-5, 7-35  
 parameters, 12-25  
   circular arc, 12-19  
   linear path, 12-18  
 password, 9-21  
 PC keyboard, 1-3, 1-7  
 peak-to-valley height, 1-15, 9-1, 9-27  
   acquisition, 9-27  
   display, 9-29, 9-33  
   maximum roughn., 9-27  
   mean roughn., 9-27  
   measuring distance, 9-31  
   select element, 9-29  
 perspectives, 4-17  
 plane selection, 4-5, 6-5  
 port, 1-5, 1-7, 7-13  
 postprocessors, 2-3, 2-5, 7-3  
 preparatory function, 7-3, 8-19, 12-7  
 printer, 1-3, 7-13, 11-13  
 programme dialogues, 2-7  
 Programming Aids, 1-15, 6-31, 7-31, 8-16, 11-1  
   names & designations, 11-3

programming code, 7-3, 12-7  
   file assignment, 7-3  
   PAL, 7-3  
 programming modes, 7-1  
   switch, 7-29, 8-15, 8-30

**-Q-**

quality control, 6-25, 9-1  
 quality testing, 6-25, 9-1

**-R-**

radius compensation, 8-11  
 rapid traverse, 4-127, 4-128, 8-27  
 re-clamping, 4-23, 4-35, 4-69  
 realtime, 4-125, 6-23  
 reference point, 4-111, 4-113, 6-1  
 reversible tip, 5-7  
   DIN identification, 5-7  
 rotation speed, 1-13, 4-119, 6-5, 8-23

**-S-**

screen display, 2-7, 3-1, 4-5, 6-5, 10-1, 12-5  
   contour display, 10-21  
   display of axes, 2-9  
   dynamic sectional displays, 10-21  
   modification, 10-17  
   sectional display, 10-11  
   work space, 10-27  
 section display, 2-9  
   milling, 10-19  
   turning, 10-17  
 Setup Mode, 1-13, 4-1  
   functions, 4-3  
   screen layout, 4-5  
   start, 4-1, 7-11  
 Setup Sheet, 3-5, 4-7, 6-1, 8-1  
   create, 3-15  
   delete, 3-15  
   edit, 3-7  
   Interpreter, 3-5  
   keywords, 3-7  
   magazine config., 4-89  
   turret config., 4-89  
 setup, 4-1  
   automatic setup, 3-5, 3-17, 6-1, 6-8  
 Simulation Mode, 6-17, 8-17  
 Single Block Mode, 6-13, 6-17, 8-17  
 sleeve, 4-33  
 spindle, 4-119, 4-127, 8-23, 8-27  
 start  
   CNC Simulator, 2-1  
   Configuration Progr., 2-3

status manager, 3-19  
 stepped jaws, 4-25  
     define, 4-43  
 subprogram, 7-3  
 surface roughness. *See peak-to-valley height*  
 switches, 4-121  
 syntax check, 1-13, 7-1, 7-9, 7-33  
 system information, 2-7, 4-5, 6-5

**-T-**

tailstock, 4-23, 4-33, 4-35  
 tangential transition, 12-13  
 Teach-In Mode, 1-13, 7-1, 8-16, 8-19  
 technology functions, 4-119, 8-23  
 test run, 4-125, 6-23  
 text bar, 9-25  
 threads, 1-15, 9-1, 9-7  
 time calculation, 4-125, 6-5, 6-23, 8-3  
 tool carrier, 4-123  
 tool change point, 4-111  
 Tool Manager, 1-15, 4-85, 5-1  
     change tool, 5-23, 5-37  
     copy data, 5-9, 5-33  
     define tool, 5-5-, 5-29  
     tool adapter, 5-19, 5-41  
     tool holder, 5-17  
 tool nose compensation (TNC), 4-105  
 tool nose compensation vector, 4-105  
 tool nose compensation vector, 4-105  
 tool path indication, 6-21  
 tool, 3-19, 4-5, 4-85, 6-5  
     adapter, 5-19, 5-41  
     change point, 4-109  
     change, 4-87, 4-95, 4-99  
     engagement time, 4-93  
     holder, 5-17  
     offset values, 4-91, 4-105  
     position, 4-5  
     reference point, 4-91, 4-105, 4-111  
     selection, 4-97, 4-101  
 transfer programme, 2-3  
 turning tools, 1-9, 5-3  
     adapter, 5-19  
     compens. values, 5-13  
     cutting edge pt., 5-19  
     drills, 5-5  
     holder, 5-17  
     reference point, 5-13  
     reversible tip, 5-5-  
     select, 5-9  
 turret, 1-9  
     reference point, 4-111  
     tooling, 4-7, 4-85, 4-89, 4-95

**-V-**

vertex, 9-5, 9-13  
 vice, 4-55  
     change, 4-57  
     define, 4-77  
     delete, 4-81  
     modify, 4-79  
     rotate, 4-61  
     select, 4-59  
 video card, 1-3  
 views, 4-17

**-W-**

WOP operation concept, 2-11  
 Working Quadrant, 4-106  
 workpiece material, 4-13, 4-15  
 workpiece, 3-19, 4-5, 6-5  
     file, 2-19, 4-19  
     geometry, 9-1  
     machining, 4-127, 6-9, 8-1, 8-19  
     management, 1-15, 4-9, 4-19  
     position, 4-63, 6-5  
     sections, 2-9  
     top view, 4-17  
     zero, 4-85, 4-111, 4-115, 8-25, 9-5  
 Workplace, 1-3  
 Workshop Oriented Programming (WOP), 1-13, 7-1, 12-1  
     Alternatives Menu, 12-23  
     copy NC block, 12-33  
     delete element, 12-27, 12-31  
     dialogue, 12-17, 12-35  
     Elements Menu, 12-11, 12-33  
 exit, 12-9  
     Input Menu, 12-15  
     Main Menu, 12-9  
     reset, 12-29  
     Segment Contour Programming, 12-35  
     start, 7-29, 8-15, 12-3  
     WOP Interface, 12-5

**-Z-**

zero point, 4-111, 9-5  
     determine, 4-115, 9-17  
 Zoom Function, 1-15, 2-9, 5-21, 6-27, 9-33, 10-25, 12-3  
     choose detail, 10-29  
     invoke, 10-25  
     menu, 10-27