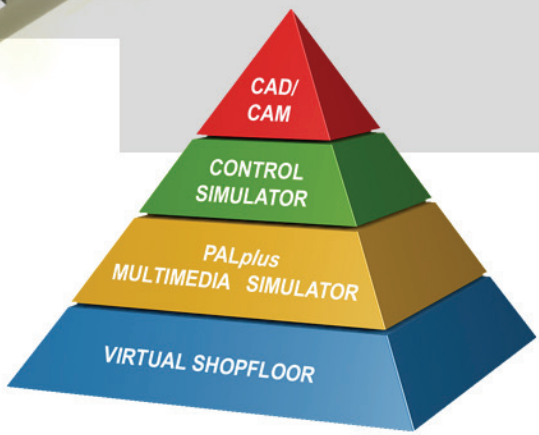
SYMplus®

5.2



Manual

CNC KELLER

Revision: February 10, 2012

Order No.: HB-E-KSXD

Publisher: CNC KELLER GmbH · Vorm Eichholz 2 · 42119 Wuppertal · Germany

Graphics and typesetting: CNC KELLER GmbH

IMPORTANT information, please read carefully

S O F T W A R E L I C E N C E A G R E E M E N T

Manufacturer of this software is CNC KELLER GmbH. The following software licence agreement shall be considered a legally binding contract between the software manufacturer and you as the licencee. By installing or using the software or parts of it you shall explicitly agree with the following licence terms. Otherwise the manufacturer is not willing to licence the software product. In this case you shall not be entitled to use, copy and install this product. Any illegal usage shall be prosecuted according to criminal and civil law.

For this reason, please read the provisions of this licence agreement carefully before deploying this product. The product contains software CNC KELLER GmbH licenses to you to be used only in compliance with the following provisions.

§ 1 Type of licence

1. In all cases, the manufacturer provides two types of licences. These are single user licences and site licences. These specify both type and scope of usage rights of the purchased product. The type that applies to the licencee is specified according to the respective order and the stipulations contained in the licence agreement. Any differences between these two types are considered in the following.

§ 2 Object of the licence

1. Object of this contract shall be the programme stored on a data carrier (CD), the programme description and the manual as well as the licence certificate along with the appropriate key number, in case of a single user licence the licence dongle as well. The mentioned objects and documents are called „software“ in the following.
2. CNC KELLER GmbH shall be the licenser and thus shall permit his customer as the licencee a non-exclusive and non-assignable licence for using the software within the scope of this agreement. All other rights concerning this software shall fully remain with CNC KELLER GmbH, hereinafter called licensor.
3. This contract shall not specify the software’s adaptation and further development, software updating, training or training services carried through by the licensor. Services like these shall be rendered based on individually concluded contracts.
4. The licensor shall leave the software for the licencee to use only on the basis of this agreement. Any contractual terms of the licencee shall not apply even if not explicitly contradicted by the licensor.
5. In case of a single user licence the licencee shall be responsible for the licence dongle, the software’s data carrier (CD) and the licence certificate along with the key number. In case the licence dongle gets lost the licence shall be considered expired. The licencee shall be obliged to inform the licensor about the loss of the licence dongle in written form. In this case the only option for the licencee shall be to sign a new licence agreement based on the terms and conditions which apply at that time.
6. In case of a site licence the licencee shall be responsible for the software’s data carrier (CD) and the licence certificate along with the key number. In case of the data carrier and / or the licence certificate plus the key number getting lost the licence shall be considered expired. The licencee shall be obliged to inform the licensor about the loss of the data carrier and / or the licence certificate plus the key number in written form. In this case the only option for the licencee shall be to sign a new licence agreement based on the terms and conditions which apply at that time.

§ 3 Copyright protection

1. The licencee shall only receive the ownership of the physical original data carrier the software is stored on. The software is copyright protected in compliance with regulations concerning the protection of computer programmes. All rights arising from copyright law shall be exclusively held by the licensor. Particularly, the copyright law protects the programme code, the documentation, the appearance, logos and other ways of depiction within the software as well as the structure and the arrangement of programme files and their names. Apart from that, the software is the licensor’s know-how which shall be protected from being accessed by unauthorised third parties.
2. The licencee shall be obliged to remain silent about any industrial secrets revealed to him while making use of the licence, also beyond the duration of the licence period. Software and documentation shall be kept as secrets by the licencee. Disclosure or revealing them to third parties shall not be permitted, as far as this is not explicitly permitted by the provisions laid down in this contract or any other written agreement concluded with the licensor.

§ 4 Ownership of rights

The licencee shall receive the usage right as laid down in this licence agreement. Purchasing rights which go beyond this software shall be excluded. The licensor shall reserve his rights particularly concerning any publishing, copying, processing and usage of this software.

§ 5 Scope of the licence

1. The single-workplace licence shall entitle the licencee to install the software on all computers at his location. Any parallel usage of this software on multiple computers at the same time is prevented by the licence dongle.

   1.1 The licensor shall grant the licencee the non-exclusive and personal right unlimited in terms of duration to install and use the software programme on as many computers as he desires. Using this programme shall comprise any permanent or temporary complete or partial copying of the programme through saving, loading, running or displaying for the purpose of executing the programme and processing of the data stored within the programme through the computer. The licencee shall acknowledge that he shall only receive the right to use the software on a computer he has access to. Any usage beyond this shall not be permitted. In particular, the data processing device the software is to be installed on shall only be located at the licencee's rooms. Furthermore, the licencee shall be obliged to have direct ownership of this data processing device. In case other persons have access to the licencee's computer the licencee shall be obliged to protect the licence dongle properly from getting into the hands of any unauthorised persons. Any transfer to third parties shall not be permitted to the licencee. The single-workplace licence is marked accordingly through the licence dongle. In case of doubt about the licence type of the software, the licencee shall be obliged to obtain information about the licence type from CNC KELLER GmbH.

2. The site licence shall entitle the licencee to use the software on all computers at a certain location. The licencee shall be obliged to inform the licensor about the location for which the respective licence agreement shall valid when signing the agreement. The granted licence shall only be valid for the location specified in the respective agreement.

   2.1 The licensor shall grant the licencee the non-exclusive and personal right unlimited in terms of duration to install and use a copy of the software programme on as many computers as he desires. Using this programme shall comprise any permanent or temporary complete or partial copying of the programme through saving, loading, running or displaying for the purpose of executing the programme and processing of the data stored within the programme through the computer. The licencee shall acknowledge that he shall only use the software on computers he owns or he has direct access to. Furthermore, the data processing devices the software is installed on shall only be located in the licencee's locations he has informed the licensor about. Any usage beyond this shall not be permitted. Any other agreements shall require the approval of the licensor in written form. In particular, the licencee shall not be allowed to carry out any further installations of the software product at different locations. In case other persons have access to the licencee's computer the licencee shall be obliged to protect the software properly from getting into the hands of any unauthorised persons, (e.g. by taking away or locking up the CD and the licence certificate, through a read-only password or the like). Any disclosure to third parties shall not be permitted to the licencee. The site licence is marked accordingly on the shipment documents. In case of doubt about the licence type of the software the licencee shall be obliged to obtain information about the licence type from CNC KELLER GmbH.

   2.2 The licencee shall be entitled to install the software on a network server. Insofar the licencee shall be obliged to purchase an extra licence for each additional location networked with the server.

3. The licencee shall not be permitted to change or translate the software or to create works derived from this software without the licensor's written approval beforehand.

4. The licencee shall not be entitled to reverse-engineer, to decompile or to disassemble the software or to try any other method to get access to the software's source code. Any right to receive the source code shall be excluded.

5. In case the licensor or the licencee cancels the usage right or if it expires due to any other reason, the licencee shall be obliged to

   5.1 hand over, in case of a single-workplace licence, the licence certificate and the licence dongle, the software which has been made available to him as well as any copies made by him as well as the documentation to the licensor immediately, yet by the time the cancellation takes effect at the latest. Any right of retention as well as a compensation for the value of the licence dongle shall not apply. In case handing over is not possible due to technical reasons, the licencee shall be obliged to delete the software, its copies and documentation from the data processing device located at his rooms. The licencee shall confirm the deletion of the licenced software, its copies and documentation to the licensor in written form immediately. The licence dongle shall be handed over to the licensor without any exceptions.

   5.2 hand over, in case of a site licence, the original data carrier and the licence certificate as well as any copies made by him as well as the documentation to the licensor immediately, yet by the time the cancellation takes effect at the latest. Any right of retention as well as a compensation for the value of the original data carrier shall not apply. In case handing over is not possible due to technical reasons the licencee shall be obliged to delete the software, its copies and documentation from the data processing device located at his rooms. The licencee shall confirm the deletion of the licenced software, its copies and documentation to the licensor in written form immediately.

§ 6 Copying

The software is protected by copyright law. The licencee shall only be allowed to make a back-up copy the purpose of which shall only be safeguarding. Such a back-up copy shall be clearly marked accordingly.

§ 7 Licence restriction

1. The software's usage right shall never be transferred to third parties. As an exception, the licencee may make a transfer based on the terms and conditions of this agreement if the licensor gives his written approval.
2. Using the software at multiple sites if no appropriate multi-site licence is signed shall be prosecuted according to criminal and civil law.
3. Generally, the licencee shall not be permitted to leave the software to third parties, in particular not to rent or lease it. In order to get an exception he shall require the explicit approval from the licensor in written form.

§ 8 Warranty

1. The licensor hereby points out that according to the current state of software technology it is not possible to fully exclude any programme errors. Thus, the object of this agreement shall only be a software which is feasible according to the terms of the programme description and the manual.
2. The licensor shall guarantee that if the contractual software is deployed as specified in the contract, it will work according to its performance description and is not defective in such a way that it cannot be used in the way laid down in the agreement to a significant degree. Insignificant discrepancies with the performance description shall not be considered defects.
3. The warranty period shall be twelve months and shall begin with the day of handing over the software. If the customer is a consumer in terms of the German Civil Code the warranty period shall be two years from the moment the software has been handed over.
4. The licencee shall be obliged to notify occurring errors immediately. Here, the licence shall point out the error and describe how it takes effect, its consequences and the circumstances under which it occurs. Warranty claims shall only be valid if the error can be reproduced or can be shown by using electronically created outputs. Error notification shall be done in written form.
5. If the defect is acknowledged by the licensor the licencee may at first demand rectification. If the acknowledged defect is not repaired within due time the licencee shall be entitled to terminate or reduce according to legal regulations.
6. Any further warranty shall not exist. In particular, the licensor shall not guarantee that the software satisfies the licencee's requirements and purposes or works flawlessly together with other programmes selected by the licencee. The licencee shall be responsible for the right choice and the consequences arising from using the software as well as the results aimed at or detained. The same applies to the written material that comes together with the software.

§ 9 Liability

1. The licensor shall be held liable only for intention and gross negligence. The licensor shall not be held liable for any damage occurring through making use of the software or the inability to use it. The licensor shall especially not be held liable for any consequential damage such as lost profit, interrupted business operations or loss of data.
2. The limitation period for claims for damage against the licensor is one year from the time of the legal begin of the limitation period.
3. Any liability of the licensor in case of a violation of the agreement through the licencee shall hereby be excluded.
4. The licensor explicitly points out that improper change of the produced NC programme may result in considerable malfunctions at the respective CNC control unit. In front of the backdrop of the area of deployment there is the possibility of considerable damage. Especially physical damage may occur. The licensor shall not be liable for damage being the result of the licencee making his own changes to the NC programmes produced by the licensor.
5. The licencee shall be obliged to have the NC programmes used by qualified personnel only and very carefully and to have the programmes checked on a regular basis. The licensor points out that during data transfer of the NC programmes, the application of the software and when running the NC programmes on the licencee's respective machine malfunctions may occur. The licensor shall not be held liable here.

§ 10 Termination

1. The licensor shall be entitled to terminate the licence agreement with immediate effect if the licencee violates a regulation laid down in this agreement.
2. The licensor shall hold the licencee liable for any damage that occurs through the licencee violating this agreement.

§ 11 Venue

This licence agreement shall be subject to the law of the Federal Republic of Germany. The sole place of jurisdiction for all disputes arising directly or indirectly from the contractual relationship, as far as legally permitted, shall be Wuppertal.

§ 12 Final provisions

1. The licencee shall not transfer individual rights from this agreement as well as this agreement as a whole to third parties. This shall not apply if the licensor has explicitly agreed to this in written form.
2. The general terms of business of CNC KELLER GmbH shall be considered additions to this licence agreement. These general terms of business may be viewed anytime online at www.cnc?keller.de and may also be downloaded. In case the contractual parties make any additional licence agreements these shall also be considered additions. If regulations of this licence agreement contradict each other and other usage agreements the regulations of this licence provisions shall apply.
3. Any side agreements have not been made. Any changes of this software licence agreement shall require the written form. The same shall apply to abrogation of the written form clause.
4. In case parts of this agreement become ineffective in parts or as a whole, the rest of the provisions shall remain untouched. The parties shall be obliged to replace the ineffective provision through one, which is closer to the economical intent.

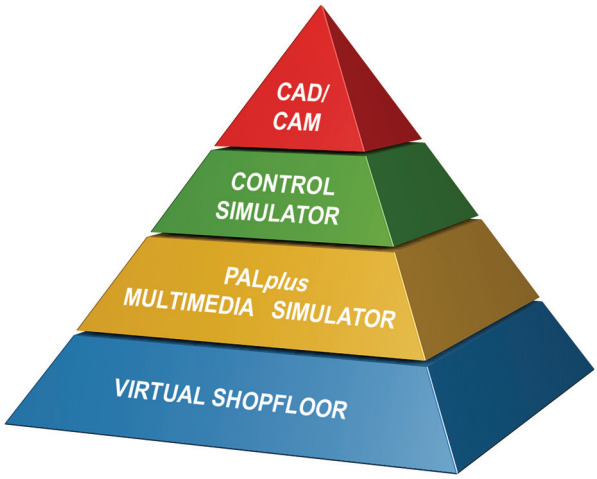
Revision: 1st of January, 2012

Foreword

Welcome to SYM*plus*.
You can set up a comprehensive training structure in CNC technology using the SYM*plus*:

- Familiarise yourself and set up CNC machines in the *Virtual Shopfloor*.
- Carry out CNC basic training with *PAL Multimedia^, G1 G2 G3* and *PAL Simulator*
- Learn the programming process for all important controls.
- Use *CAD/CAM* in the training/production interface.

This manual contains additional information about SYM*plus*. The term "additional information" is used to indicate that this manual does not contain the entire software performance (for this, more than 1,000 pages would be required), but only the information not covered in the training books and which is particularly important.



We assume that you have studied the work documents included with the software, regarding the various operating modes "from A to Z" and are capable of programming either in the CNC basic training or in the *graphical dialog* without this manual. As far as the graphical dialog is concerned, please refer to Chapter 4 of the individual work document.

We work on the basis that in this case you are familiar with approx. 90 % of the software. However, for special cases complementary information is often required and helpful to proceed with individual settings and that further information on special cases will broaden your knowledge of how to make efficient use of the SYM*plus*. Only one manual being available for the SYM*plus*, the technologies Turning and Milling are dealt with in an integrative manner:

By integrative we mean here that we cover the information across technologies in one chapter, and the technology-specific information in two further chapters.

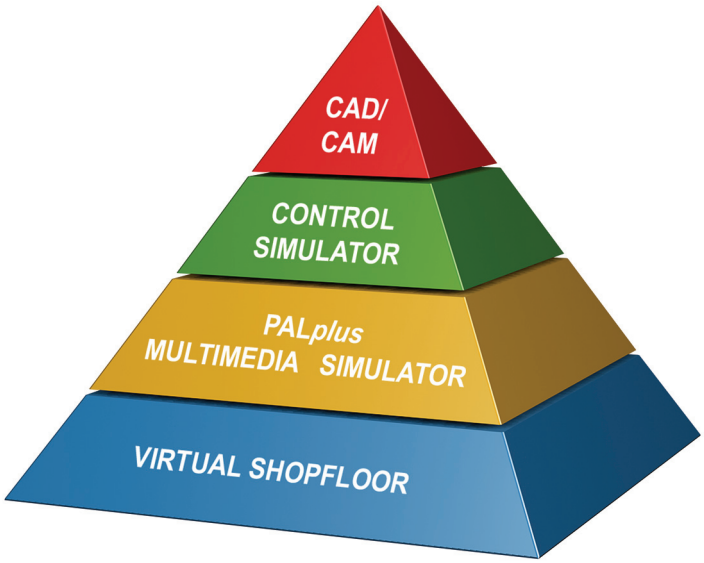
Due to the fact that each of these 3 chapters is strictly structured according to operating moves and the menus contained, it is relatively easy to find reference to your problem by looking either in chapter 'General Supplements' and - if you have not found what you wanted there - at a corresponding point in the technology chapter.

We wish you every success and pleasure with when using the SYM*plus*.

Wuppertal, Spring 2010

Table of contents

1 Didactic-methodic information



You can set up a comprehensive training structure in CNC technology using the SYM*plus*, i.e.
• Familiarising yourself and setting up machines
• CNC basic training with PAL*plus*
• Control system related specialised industrial training
and
• Graphical programming
are unified in one system.
The "Training Pyramid" on the left is to be understood as a symbolic characterisation of the scope of SYM*plus*.

1.1 General questions and our answers to CNC qualification

1. Starting with turning or milling - what is the better choice?
Answer: We recommend to start with milling.
Arguments in favour of why one should start with milling refer to both the work on the PC (theory) and work at the machine (practice).
The following arguments hold true for the PC:

1. Known co-ordinate system X/Y (instead of X/Z when turning).
2. The incremental values for determining the centre point of a circle I and J react in a ratio of 1:1 instead of 1:2 for I and K.
3. The principle of the tool path correction can be taught easier than for the turning process (see allowance with an increasing cone and reject parts with a decreasing cone, if no tool path correction is programmed).
4. Notably simpler technological interrelations (see the complex interrelations between advance/feed speed on the one hand and cutting speed/rotational speed on the other during turning).
5. "Natural movements": The tool is clamped, the tool is moving (during turning, chip removal occurs by the relative movement of workpiece and tool).

The following arguments apply to the machine:

1. The smaller mass of the moving tool provokes an initial inhibiting threshold (refer to the mostly larger mass of the workpiece and the clamping means when turning).
2. Reduced danger of collision knowing that the tool used has "no neighbouring tools" that could collide with the machine.
3. Improved visibility (see, for instance, the internal machining on turned parts).
4. It is easier to move by individual blocks (in contrast, see the development of an attachment cutter in a lathe).
5. Even if the cutting data is not optimum, no dangerous swarf/chips are removed (see the possible formation of long entangled and spiral swarf produced by lathes).

2. Making a practical start on the CNC machine or a theoretical start on the PC - what is the better choice?

Training starts with a demonstration on the CNC machine, followed by an in-depth theoretical block on the PC. The knowledge obtained is then verified on the CNC machine.

3. What is the time relationship between CNC practice and CNC theory?

The time relationship between CNC theory and CNC practice strongly depends on the number of people taking part in the course and the learning objectives strived for. Tendency: The more participants in a training group, the larger the theoretical section on the PC. In the first training course, the practical portion outweighs that of an advanced course.

4. How are maths integrated into CNC qualification?

Mathematics are marginally dealt with in working documents on SYM*plus*. Reason: Due to the fact that maths are no longer a subject in the exams, the typical applications of Pythagoras's theorem and trigonometric functions to be used when calculating in geometry are only dealt with as film demonstrations (as an aid).

If mathematics is used in the CNC basic training, however, mathematics should not be focused on "for too long and at too high a level", we must not forget that CNC technology is concerned here. Possibly, graphical programming should be used for contour determination to be able to work with G1, G2, G3 afterwards. This way is a great help particularly for complex geometries (see page 89 in the Turning manual and page 96 in the Milling manual).

5. When milling, will the level XZ with Y as a tool axis (G18) be referred to?

The XZ level is only dealt with for *PAL multimedia* in the chapter *Points on the workpiece*.

Reason: There is no XZ level with PAL, and in practice this level does not have the importance of the scale as the XY level (for this reason, many companies training apprentices have had the XZ level typical for MAHO machines changed over in the software to the XY level).

6. Starting with geometry or technology - what is the better choice?

CNC qualification with the working documents SYM*plus* starts with geometry, in other words, with the technical drawing.

Reason: Knowing that the technical drawing is always the starting point of all considerations involved in a manufacturing task, we also start with geometry in the working documents. Teaching the abstract G functions, especially the path conditions G2 and G3 are somewhat difficult to explain. For this reason, the exercises are initially limited to straight lines, quarter and semi-circles.

Also the many pre-settings, such as raw component, tools and the F, S, T and M addresses do not influence this decision, because corresponding success becomes visible after very few entries (see for instance the descriptive 3D view with mass and volume display).

1.2 Conventional and graphical programming

> "A workpiece does not reveal whether it has been programmed according to **DIN** or **graphically**."
>
> H. Forster - Head of Education and Training of Steinemann in St. Gallen

SYM*plus* includes the conventional programming according to DIN 66025 with PAL*plus* as well as the graphical programming. To allow the instructor to translate **his** method, the contents in the working documents SYM*plus* do not intermesh but are kept separate. This makes it possible to start work with SYM*plus* in every chapter. If there is time enough, the individual working document can be worked through from the beginning to the end, which also would make sense.

The following are some general comments to this subject:

1. Conventional programming will continue to be dominating for some time (refer to the majority of currently existing CNC control systems and also the PAL issue).
2. Working with the KELLER specific "graphic dialog" is much simpler than programming with the abstract NC code. The increasing importance of graphical systems in areas of CNC technology is manifested by the list of the following systems: SIEMENS ShopMill, SIEMENS ShopTurn, TRAUB IPS, FANUC FAPT, OKUMA, MAZAK, BOSCH CC200T, ...)
3. To an increasing degree, graphical programming will substitute conventional programming for the simple reason that the PC and software performance of the controls systems will continually increase, allowing work to become more and more convenient and efficient.
4. Efficiency also in qualification will gain in importance, and it is for this reason that an integration of graphical programming in further training and education is indispensable.

> "Training has to be intermeshed with production to an increasing degree and become more cost efficient, i.e. training has to become part of the chain of created value.
>
> Norbert Braun, Head of techn. education and Head of the toolmaking department, Mercedes-Benz Lenkungen GmbH, Düsseldorf

Differences between conventional and graphical programming quickly become very obvious when learning according to the working documents. At this point, we wish to compared the differences by way of the following brief review.

1.3 From the drawing to the NC program ...

1.3.1 ... by conventional programming

- In the 'PAL simulator' mode, the NC blocks are entered block by block in a guided mode and checked by simulation.
- If you prefer a "control specific" start, you can enter the NC blocks in the 'Simulator' mode in the format of your control system block by block in the guided mode and check by simulation.
- Prerequisites for this type of programming is profound knowledge of the G and M functions and the ability to translate the pre-defined parameters from the drawing into tool traverse movements in a structured manner. Depending on the drawing, also mathematical knowledge is required.

1.3.2 ... by graphical programming

- Although machining steps can be performed immediately on the raw part, when simple workpiece are concerned, first of all it is typically the workpiece geometry which is created in the 'Geometry' mode in the "Graphic Dialog" or by adopting CAD data.

  Prerequisites to do this is "just" the ability to read a drawing and to mentally translate it into geometrical elements. Special advantages: No coding, even complex contours can be entered without mathematical knowledge. Variant designs are also possible in the sense of component families.

- The designed finished part and the raw part are interlinked in the 'Workplan' mode. The machining steps are created by means of pictograph selection after the choice of both tools and material.

  Prerequisites:
  Technological know-how regarding the cutting data and optimum production step sequence

  The NC program is automatically created for any desired control system (when the attendant post-processors were installed).

  Special advantage: No coding, software "Intelligence" for residual material detection and traverse path optimisation, good "legibility" of the workplan instead of an abstract NC program.

- The following is a summary of the advantages the graphic dialog of SYM*plus* offers:

  Even complicated workpieces can be entered within minutes.

  All residual material is automatically detected leading to in part substantial production time reductions.

  The user can concentrate on the comparison of different machining strategies to be able to select the optimum process.

  For you this means:
  NC program creation is simpler, faster and more cost-effective!

2 General complementary information

This general section includes information applicable to SYM*plus* Turning and Milling.

2.1 Operation/Control

2.1.1 PC keyboard and mouse

The *plus*-Systeme were created on the basis of practical experience for practical use. This meant that also their operation had to be adapted to the shopfloor conditions.

Knowing that typically there is often too much dirt in a shopfloor to allow problem-free operation with the PC mouse, the *plus*-Systeme were initially designed for keyboard operation only. A user concept was created permitting the user to work very quickly (knowing that a constant change from mouse to keyboard and back significantly slows down the entry speed).
By-and-by, the *plus*-Systeme were used more frequently outside of the actual shopfloor. Here, users familiar with Windows frequently expect an operating concept typically for Windows.
This led us to integrate the mouse (also the mouse with a wheel is supported) in the *plus*-Systeme in order to facilitate handling or even the change-over to the KELLER operating concept:

- Selecting and changing files, work steps, NC blocks with a double click
- Moving the blue selection beam with the "mouse wheel"
- Selections options in selection windows:
  <LH or RH mouse key> = next and previous option

  Click to open the selection list and select an option

NOTE:
In the KELLER operating concept, the <Enter> key plays a role that is equally important as the corresponding key on the different controls. As from the SYM*plus* you learn that all entries have always to be concluded by pressing the <Enter> key, as a preparatory step to operating a control system.
You can also conclude entries, as usual for Windows, with the <Tab> key (also see ”Keyboard assignment of the SYMplus” on page 109).

2.1.2 Entries via the decimal number pad

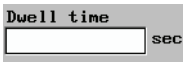
Knowing that most of the entries are numbers, we recommend to use the **decimal number pad** on your keyboard. In this manner your hand can stay on these keys, and your entries can be made much faster than by trying to ’find’ the right keys in the top row of keys on the PC keyboard. You can speed up your entries even further if you learn to make ’blind’ use of the keys. Moreover, the important keys <Enter> (to conclude an entry) as well as <+>/<-> (to select an option, contour, ...) are arranged directly adjacent.

NOTE:
For as long as no input fields are displayed, you can very easily and quickly work in all menus etc. by using the keys of the decimal number pad. For instance,
press the <1> key on the decimal number pad (or <Enter>, when the blue cursor is on the corresponding menu point), if you wish, for example, to invoke menu point <F1> ’File’ in the ’Geometry’ operating mode.

2.1.3 Input, selection and display windows

Input fields

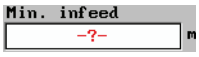
SYM*plus* offers four types of input fields to be used to enter either only numbers or also letters, depending on the situation:

- 

Empty input fields

- 

Input fields pre-assigned with a default value as a proposal.

- 

Input fields with a red question mark.
Here an entry of the user is indispensable.

- In the PAL simulator only:

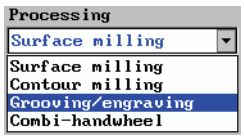

Input fields with a blue corner.
Obligatory field, this address is required

NOTE:
To be able to make an entry, select the field in question using the <Arrow keys> or the mouse. When making an entry, the existing content of the field will be replaced. You can change the existing content by double-clicking or pressing the <F9> 'Change' key. Always complete your entry by pressing the <Enter> key.

Selection fields

The selection fields of SYM*plus* are always pre-assigned with the first default option feasible. Any of the other options can be selected in the following manner:

- Press the <+> or <-> keys or use the scroll wheel of a corresponding mouse to have the individual options of the row displayed either forward or backward. An option can be directly selected by direct input (refer to the explanation given below).



A click or a double click with the mouse is the "simplest" method to open the selection list and to select the option in the blue surface.
The selection list can also be displayed using the <F9> key. The desired option is then selected with the <Arrow keys> and then taken over with the <Enter> key.

- These keys are available to navigate in the "dropped down" selection list:

"Fast scrolling" forward and backward

Jump to the first or last list entry

Letters, numbers etc.

- You can directly go to an option by entering the first few characters of the option required. When entering the first character, the marking moves to the first option that commences with this character. Should several options start with this character the entry of the second character refers to the second letter etc. until the command is clearly selected by the 'n' character.

Tip:
Try out the different possibilities and find out the easiest and fastest way for you. In general, selecting with the mouse is the best way.

Display fields

At many places in SYM*plus* you will find fields that are exclusively used to display values or options resulting from computations or presettings.

2.1.4 Dialog window sequence

In the operating modes 'Workplan' and 'Simulator' you find special dialogs consisting of a series of dialog windows. These are necessary, because several entries are required for a work step or an NC command or cycle than would fit on one dialog page. The majority of work steps and some commands of different simulator modules have such dialog window sequences. This can be seen from the small boxes in the dialog bottom.

A green hook indicates that the inputs have already been applied in the corresponding dialog with <F10>.
The blue arrows show the position of the current dialog. Grey shows that the corresponding dialogs with the data contained have not yet been taken over with <F10>.
The entry of the corresponding work step or command is only concluded when all dialogs have been applied with <F10>. It is possible to toggle between the individual dialogs using the <LH arrow> and <RH arrow> keys or by clicking into the small boxes.

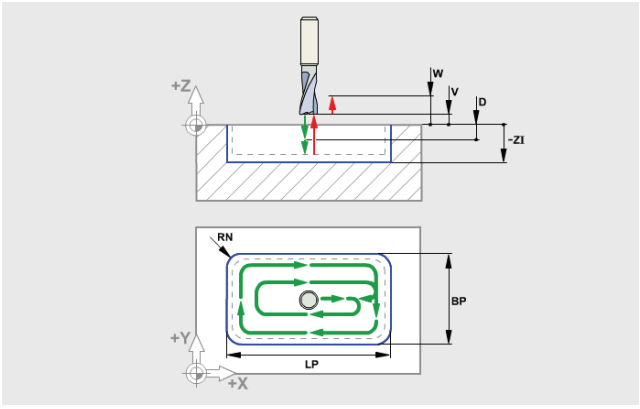
2.1.5 The symbol bar

The title bar of SYM*plus* contains a symbol bar to facilitate invoking important program functions. The appearance of this symbol bar can vary depending on the situation because only those symbols appear that are required for the current operation. Refer to the following text for explanations on the individual functions.

2.1.5.1 Help system

Help system for the NC commands

In the 'G1 G2 G3', 'PAL simulator' and 'Simulator' operating modes you can invoke a context sensitive help system using the <F12> key or by clicking on the info symbol for the individual commands.
Press the <F12> key again or click again on the symbol when you want to terminate the help system.

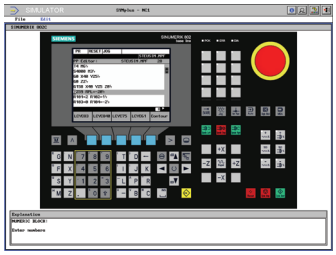


A typical help mask

Corresponding symbols above the mask indicate whether or not several help masks are available for one command. To reach the next mask, click on these symbols or use the key combinations <Ctrl>+<LH arrow key> or <RH arrow key>.

Tip:
To familiarise yourself even better with the individual commands of a control system or the PAL simulator, invoke the help system and toggle between the commands using the <+> and <-> keys. Then, the attendant masks will automatically appear and you can immediately see the function executed by the individual command.

Help system for the keyboard



In the 'Simulator' operating mode it is additionally possible to invoke a help system used to learn the respective control keyboard. If no command is edited or, if you are in the 'Free editor' (see Chapt.2.6.4.2), the keyboard information symbol is displayed.
Using <F12> or clicking on the keyboard information symbol with the mouse invokes or terminates the help system.

Multi-media 'Training module'

In the HAAS simulator (turning and milling), <F12> or clicking with the mouse on the keyboard information system invokes the multi-media 'Training module'.

Not only the control keyboard and its use are explained more extensively and detailed here but you will also get more familiar with all important geometrical and technological CNC basics. A final test indicates the learning status (incl. certificate).

Different functions are available depending on the situation:

Solution — Display of the solution for the corresponding task.

These symbols can be used to toggle between the task pages.

The menu symbol is used to return to the selection menu from the individual test sequences.

The print symbol can be used to print the test certificate.

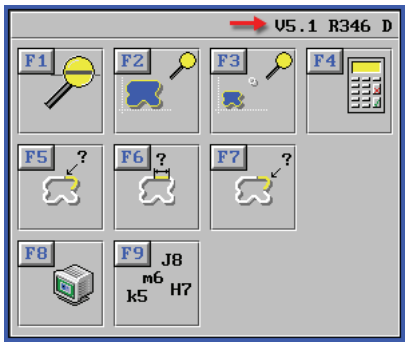
Display of the Adobe SHOCKWAVE PLAYER® version number.

For this, consider the hardware and software requirements in the installation instructions under "Installation of SYMplus" on page 1.

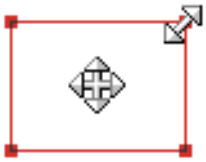
NOTE:
When the 'Training module' is switched on, the CPU of the PC is almost at full capacity. This status is normal because the Adobe SHOCKWAVE PLAYER® requires a large amount of computing capacity. When SYM*plus* is minimised to switch over to a different application, the CPU capacity required will automatically reduce to 0%.

2.1.5.2 Additional functions

You can call up a menu using the <F11> key which provides you with different additional functions or clicking on the zoom symbol. Depending on the situation, either all or certain functions can be selected. With the corresponding configuration (see Chapt.2.1.5.4) you can also invoke the following functions directly using the respective key combination <Alt>+<1>...<9>.
The version number of the *plus* software used on your computer is displayed in the top right. Press the <F11> key again or click again on the symbol when you want to terminate the additional functions.



<F1> Zoom: Enlarges a settable section



The desired section is set via a red frame with four pulling points. Click on the position you want to have displayed enlarged. The frame is automatically positioned over the position clicked on. Now change the frame size using the scroll wheel of the mouse or click on the pulling points to pull the frame to the desired position. The entire frame can be displayed by clicking and pulling inside the frame. Then double click inside the frame, the set section will be displayed enlarged.

<F2> Overall: Changes to the total view of the workpiece

<F3> Work area: Displays the complete machining compartment according to the 'Machine description' (see Chapt.2.10.4.1).

<F4> Calculator: Invokes a "calculator". The calculated value is taken over into the active input field.

<F5> Points: Invokes the point determination. The coordinates of the points can directly be taken over into corresponding active input fields.

<F6> Dimensions: Invokes the measuring function. The spacing and angle between any selected points can be determined. In the 2D simulation, this function can be used to measure the "manufactured workpiece".

<F7> Elements: Invokes the geometry information. After selecting an element, all information is displayed (start/end point, length, radius for arcs, etc.). It can also be used in the 2D simulation to check the simulation result.

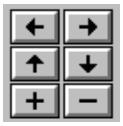
<F8> Print: The screen print-out function serves to print images/masks (for example, simulation, 3D or tool masks). The menu of the respective operating mode is used to print out programs, set-up sheets or other text information. This function must only be used for printing evaluations in the 'PAL simulator' and 'Teacher' operating modes because the evaluations are displayed in a separate dialog. The operating mode menu cannot be reached from this dialog.

<F9> Adjusting dimensions: Allows the entry of adjusting dimensions. The corresponding limit values are displayed and the tolerance field centre can directly be taken over into corresponding active input fields.

3D additional functions

The 3D view requires special functions of the magnifying glass. These functions may be required to rotate <F1>, position <F2> and change <F3> the size of the displayed workpiece in 3D.

If you feel that the mouse is not precise enough for this process, you can also set the current view by means of the displayed navigation keys, in accordance with the respective selected function.



In addition you can use the <F6> 'Screen print-out' function.

Tip:
The workpiece can also quickly be moved directly in the 3D view without the 3D additional functions.
Rotate by keeping the LH mouse key pressed and moving the mouse. Displace by keeping both (or the central) mouse key(s) pressed and moving the mouse. Enlarge by keeping the RH mouse key pressed and moving the mouse or using the mouse wheel.

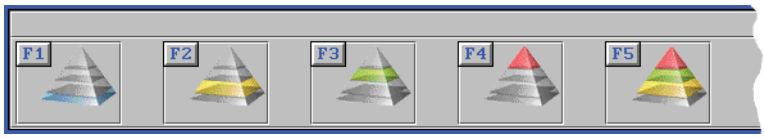
2.1.5.3 Configuration selection

This symbol is only displayed if SYM*plus* is respectively configured. It is used to invoke the selection dialog containing the configuration steps at any time. For this purpose, the software is automatically restarted. For calling up, you can also use the key combination <Ctrl>+<F12>.

Tip:
You can also select one of the possible configurations without this symbol when starting SYM*plus* .

2.1.5.4 Configuration adaptation

As a standard, up to five training levels are offered when starting SYM*plus* (depending on the access right, see "Changing access rights" on page 17).



Possibilities through configuration

SYMplus can be completely adapted to the requirements of trainers/teachers and trainees/students.
The 'Teacher' operating mode (see "Operating modes" from page 18), for example, is not provided in the factory-set installation with restricted access rights (for trainees/students).
In the configuration, you can also determine whether or not the start assistant should appear in the operating modes, the tools are allowed to be changed, the 'Transfer' operating mode should be displayed and much more.
If, for example, the 'PAL simulator' is to be automatically started only without configuration selection, this can be configured.

Changing configurations

ATTENTION:
The SYM*plus* configurations are factory-set to conform with the working documents and the various user groups. When SYM*plus* is to be adapted to your own working environment, you can adapt the configuration to your requirements. For this purpose, change the files described below respectively. These files are provided with comments to support you when carrying out changes.

However, these files are sensitive to incorrect entries. They might prevent SYM*plus* from starting correctly. For this reason, we recommend to only adapt these files with caution and after a back-up has been made.

During installation, you have decided either for 'Full access rights' or 'Restricted access rights'. The following files of the system data are used by SYM*plus* for the respective configuration:

- Full access rights = **`emod.ini`** (e.g. teachers, trainers)
- Restricted access rights = **`mod.ini`** (e.g. students, trainees)

In these files you are allowed to set:

- the configuration levels to be displayed.
- the operating modes to be offered on each configuration level and the order and position.
- the options to be inaccessible to the user in the individual operating modes.
- whether or not the start assistant should be displayed in the individual operating modes.
- whether or not the key combination <Alt>+<1>...<6> to invoke the additional functions is available.

In these files you are **not** permitted to set:

- several identical operating modes in one configuration level
- more than nine configuration levels
- 'Machine' operating mode together with other operating modes in one configuration level

You cannot change the configuration level names displayed.

Changing access rights

If you want to subsequently change access rights for the users defined during installation, open the `cpmain.ini` file in the user data of the respective users. Write behind the line '`eMod=`' '`0`' for 'Restricted access rights' or '`1`' for 'Full access rights'.

2.1.5.5 Operating mode selection

This is used to invoke the operating mode selection dialog displaying the operating modes available for selection in the currently selected configuration level. For calling up, you can also use the key combination <Ctrl>+<F10>.
This is the most important symbol because it is not possible to work with this software without knowing how to toggle between the operating modes.

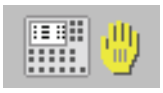
Then select the desired operating mode or exit the dialog by pressing the key combination <Ctrl>+<F10> again or clicking on the symbol.

Tip:
Once you become an experienced user, the desired operating mode can also be directly invoked with the key combination <Ctrl>+<Fx> without invoking the operating mode selection dialog. <Fx> complies with the F keys displayed in the operating mode selection dialog. The available operating modes can vary depending on the configuration level and type of installation.

2.1.5.6 Operating modes

In the ’**Shopfloor**’ operating mode you can interactively familiarise yourself with the CNC turning and milling machines and their environment in a virtual 3D shopfloor. You will experience a new dimension of modern CNC qualification! *

In the ’**Machine**’ operating mode, you can explore the essential components and the interrelatedness of the functions of a typical CNC milling machine and/or CNC lathe.



In the ’**Operation**’ operating mode you can learn all important operating steps on the real machine 1:1 with the 3D machines as well as with the SINUMERIK control systems without stress and without presenting any danger.*

In the ’**PAL multi-media**’ operating mode you can familiarise yourself with all important geometrical and technological basics for CNC programming. Furthermore, you will be introduced multi-medially into new PAL codes (special commands and cycles). A final test indicates the learning status (incl. certificate) *

In the ’**G1 G2 G3**’ operating mode you can use the geometrical basics of G0, G1, G2, G3, G90 and G91 for any contour. Then these geometry programs can be imported into the ’PAL simulator’ operating mode and "animated" by adding technology blocks. In this manner, you will be introduced step-by-step into NC programming according to DIN 66025.

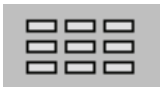
In the ’**PAL simulator**’ operating mode you can write NC programs according to the new PAL coding in a "guided" editor with an integrated help system and simulate same for checking. The integrated exam module is intended for your individual preparation for the examination. The automatic evaluation shows you where you stand.

In the ’**Simulator**’ operating mode, it is possible to write and simulate NC programs for various control system formats in a "guided" editor with an integrated, control system specific help system. A keyboard help system is additionally provided for each control system in which the functions of the individual control keys are explained. A simulator of the SINUMERIK 802S/C control system is provided in the standard version, further simulators are optionally available.

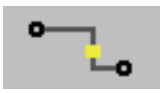
In the ’**Geometry**’ operating mode, the construction of the workpiece geometry is created in the graphical dialog via pictographs or existing CAD data. For this purpose, DXF and IGES files can be used. For this refer to the subject ”’CAD input’” on page 30.

In the ’**Workplan**’ operating mode, the graphical programming guides you quickly and easily from the drawing to the NC program without ever having written a G and M function.

In the ’**Teacher**’ operating mode, you can create NC exercises and NC examinations yourself for your training lessons (requiring the entry of a password) and proceed with an automatic evaluation by floppy disks of the students.



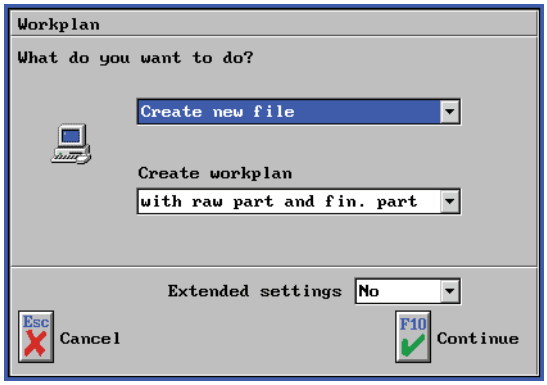
In the ’**Setup**’ operating mode, you can geometrically record tools, file cutting data and equip turrets or magazines. The folders for the various file types of SYM*plus* are also managed here; settings can be made in the system configuration.



In the ’**Transfer**’ operating mode, you can transfer NC programs between PC and control system. You can also look at or adapt NC programs and setting sheets in a simple text editor, simulation, however, is not possible here.

* For this, also take the hardware and software requirements in the installation instructions under ”Installation of SYMplus” on page 1 into account.

The start assistant in the operating modes



In all "main operating modes", i.e. in which the main work is to be carried out, you are welcomed by a start assistant and quickly guided to the actual entry. It carries out the main work for you particularly when creating a new file because it combines the presetting for the respective operating mode with your entries and subsequently provides the new file for processing.

When using the start assistant and want to enter more detailed settings, set the switch for *Extended settings* in the start assistant to *Yes*. In the following dialog, further settings can be carried out.

You can also terminate the start assistant and carry out required settings via the respective menu points. If you want to skip this little assistant in general, please read chapter ”Configuration adaptation” on page 16. The start assistant does not appear in the "Secondary operating modes" ’Set-up’, ’Teacher’ and ’Transfer’.

2.1.5.7 Window frame call-up

SYM*plus* has a fixed operating surface of 1024 x 768 pixels. If you use a higher screen resolution on your PC, SYM*plus* will appear in a correspondingly smaller window.

SYM*plus* is only displayed as full image and becomes visible as a window symbol in the title bar when the screen resolution is set to 1024 x 768 pixels. The window frame of the application and the Windows task bar are mapped out with this screen resolution.

To have them displayed again, click on the window symbol in the title bar or use the key combinations <Alt>+<Pos1>.

This ensures that the window frame is displayed with its usual symbols. To return to the full image display, click again on the window symbol, on the maximise symbol now displayed in the window frame or use the
key combinations <Alt>+<Pos1>.

2.1.5.8 Software exit

Click on this symbol or use the key combination <Alt>+<F4> to exit the software. When the window frame of the application is visible, you can also click on the closing symbol of this frame.

2.2 Printing

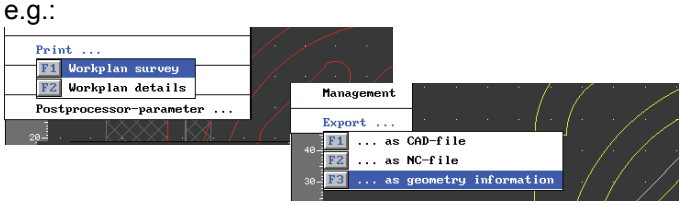
When you use the printing function in SYM*plus*, the Windows print dialog opens for you to select the desired printer.

2.2.1 Print call-ups

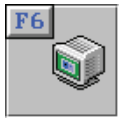
SYM*plus* offers the possibility to print files from the 'additional functions' dialog either via menu points or also via the print symbol.

Each of these two possibilities produces different results:

- The individual menu points in the operating modes permit printing out or editing a file of detailed information on a corresponding subject, e.g.:



- When invoking the 'Additional functions' dialog, you can print the current screen contents using <F6>. If the current screen content does not show any printable data or if you have to print out via the individual menu, the printer symbol is not displayed or is locked.



2.2.2 Print contents in the operating modes

Please refer to the following list for detailed information on what you can print in each case:

Operating mode: 'G1 G2 G3'

NC program:
Open NC program / <F1> 'File' / <F6> 'Print'

Simulation 'PAL simulator' operating mode / 'Simulator' / 'Workplan'

Simulated image 2D:
2D simulation / <F8> Stop simulation /
<F11> 'Additional functions' / <F8> 'Print'

"Production photo" 3D:
2 D simulation / <F8> Stop simulation / <F3> '3D view' /
<F11> 'Additional functions' / <F6> 'Print'

Simulated image 3D:
3D simulation / <F8> Stop simulation /
<F11> 'Additional functions' / <F6> 'Print'

Operating mode: 'PAL simulator'

NC program:
Open NC program / <F1> 'File' / <F6> 'Print'

Correction sheet of an NC exercise:
<F3> 'Exercise/Examination' / <F2> 'Solution ...' <F1> 'View at' / Select solution / <F11> 'Additional functions' / <F8> 'Print'

'Simulator' operating mode

NC program:
Open NC program / <F1> 'File' / <F6> 'Print'

‘Geometry’ operating mode

Workpiece as a 2D graphic:
Open geometry / <F11> ’Additional functions’ / <F8> ’Print’
Workpiece as a 3D graphic:
Open geometry / <F2> ’Edit’ / <F8> ’3D view’ / <F11> ’Additional functions’ / <F6> ’Print’
CAD data as a table:
Open geometry / <F1> ’File’ / <F6> ’Export’ / <F1> ’... as CAD file’ /
Select contours and subsequently set the ’Output on the printer’ option
Geometry data as a table:
Open geometry / <F1> ’File’ / <F6> ’Export’ / <F2> ’... as NC file’ /
Select contour and subsequently set the ’Output on the printer’ option
Geometry data as a table:
Open geometry / <F1> ’File’ / <F6> ’Export’ / <F3> ’... as geometry information’

’Workplan’ operating mode

Workplan review:
Open workplan / <F1> ’File’ / <F6> ’Print’ / <F1> ’Workplan overview’
Workplan in detail:
Open workplan / <F1> ’File’ / <F6> ’Print’ / <F2> ’Detailed workplan’

’Teacher’ operating mode

NC exercise (with empty gaps):
Open exercise or examination / <F1> ’File’ / <F6> ’Print ...’ / <F1> ’as task’
NC sample solution (with the gaps filled in):
Open exercise or examination / <F1> ’File’ / <F6> ’Print ...’ / <F2> ’as sample solution’
NC program (as program without gaps):
Open exercise or examination / <F1> ’File’ / <F6> ’Print ...’ / <F3> ’as NC program’
Evaluation as NC program (with the gaps filled in by the student):
<F2> ’Evaluation’ / <F2> ’Print ...’ / <F1> ’as NC program’ / Mark files with <F5> ’Add’ / <F10> ’OK’
Evaluation as correction (with the points achieved by the student):
<F2> ’Evaluation’ / <F2> ’Print ...’ / <F2> ’as correction’ / Mark files with <F5> ’Add’ / <F10> ’OK’

’Set-up’ operating mode

Individual tool:
<F1> ’Tools’ / <F4> ’Print’ / Mark tool / <F3> ’Print tool’
Tool list:
<F1> ’Tools’ / <F4> ’Print’ / <F4> ’Print tool list’
Magazine or turret:
<F2> ’Magazine’ or ’Turret’ / <F4> ’Print’ / Select ’Magazine’ or ’Turret’ / <F10> ’OK’ or double-click

Operating mode: ’Transfer’

NC program:
Open NC program / <F1> ’File’ / <F5> ’Print’
Set-up sheet:
Open set-up sheet / <F1> ’File’ / <F5> ’Print’

2.3 'Shopfloor', 'Machine' and 'Operation' operating modes

2.3.1 Keyboard assignment

In the virtual shopfloor and in the Machine familiarisation / set-up menu, you can move around the machine in various ways. The keys to be used for this purpose are specified under 'Keyboard assignment', once you have clicked on the top right ? symbol.

2.3.1.1 'Shopfloor' operating mode

Forwards / backwards	<Arrow key up> or <Arrow key down>
Turning to the left / to the right	<LH arrow key> or <RH arrow key>
Fast forwards / fast backwards	<Ctrl>+<Arrow key up> or <Arrow key down>
Sideways to the left / to the right	<Ctrl>+<LH arrow key> or <RH arrow key>
Tilting forwards / backwards	<Scroll up> or <Scroll down>
Up / down	<Pos1> or <End>

2.3.1.2 'Machine' / 'Operation' operating modes

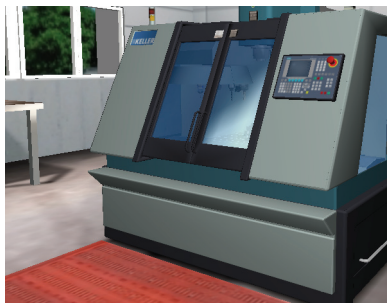
Forwards / backwards	<Arrow key up> or <Arrow key down>
Turning to the left / to the right	<LH arrow key> or <RH arrow key>
Sideways to the left / to the right	<Ctrl>+<LH arrow key> or <RH arrow key>
Tilting forwards / backwards	<Scroll up> or <Scroll down>
Up / down	<Pos1> or <End>

2.3.2 Graphic settings

If the cooperation of graphic card and drivers, or their settings respectively, is not optimal, smooth surfaces with stripes or semitransparent objects might appear in the virtual shopfloor as illustrated in the example of the milling machine.

To optimise the display, click on the ? symbol at the top right and then on 'Graphic settings'.

Test the various options one after the other and observe the changes in the display. The machine should be look as illustrated in the picture on the right.



Tip:
The 'Software' option is only intended as a makeshift for PCs without sufficient graphical capacity. Due to the fact that this option only allows a slow and jerky display of the shopfloor
; for this reason, we recommend to use a state-of-the-art
3D graphics card. For this, consider the hardware and software requirements in the installation instructions under "System requirements" on page 1.

2.3.3 Further functions

Depending on the situation, further functions are available via additional symbols:

These symbols can be used to navigate within the help pages.

The menu symbol is used to return to the selection menu from the individual sequences of ’Set-up machine’.

If you have caused a collision on the virtual machine, invoke the € symbol to call up the collision examples.

This symbol is used to start animations in the training modules.

The Exit symbol is used to terminate training modules.

This symbol appears, when pointing with the mouse indicator to specific machine components or objects within the *Virtual shopfloor* or in *Machine familiarisation*. Invoke the corresponding training module using the mouse.

A text containing instructions on the task to be executed appears, when you point on the information symbol in *Machine set-up* with the mouse.

2.3.4 CPU capacity utilisation

When the ’Machine’ operating mode is switched on, the CPU of the PC is almost at full capacity. This status is normal because the Adobe SHOCKWAVE PLAYER® requires a large amount of computing capacity. When SYM*plus* is minimised to switch over to a different application, the CPU capacity required will automatically reduce to 0%.

2.4 Operating mode: ’G1 G2 G3

In the following please find information for the ’G1 G2 G3’ operating mode which apply to Turning and Milling.

2.4.1 ’File’ menu

2.4.1.1 ’Settings’

The start point for the first NC block can be adapted under <F1> ’File’ / <F2> ’Settings’. The start point is preassigned by the presettings for this operating mode.

2.4.2 ’Edit’ menu

Depending on the operating mode configuration, you can create, change and delete NC blocks at any position using <F1> ’Create’, <F2> ’Change’ and <F3> ’Delete’ or, you can execute this functions in the last NC block of the geometry program only.

Changing the geometry program at any position offers, for example, the possibility to learn the effects of G90/G91 on the following blocks. On the other hand, it is not necessary to delete all blocks up to an incorrectly entered block to be able to correct this block. However, changes in the middle of a program can cause consequential errors, for example, circle end point errors. Although these errors will be displayed, beginners are often stressed or insecure, especially when more than one error has occurred.

For this reason, you have to decide about the degree of freedom suitable for you and the students. You can adapt the editor to your requirements. For this purpose, change the respective configuration file as described under . Go to the ’editModExt=’ line and enter ’0’ for "changes only at the end" or ’1’ for "Changes in each block" behind the equals sign. For this reason, you have to decide about the degree of freedom suitable for you and the students. You can adapt the editor to your requirements. For this purpose, change the

2.5 Operating mode: ’PAL simulator

The PAL simulator is an individual operating mode to cope with training requirements. However, its behaviour is widely identical with that of the control system simulators (see Chapt.2.6).

2.5.1 Zero point table

In the PAL simulator from version 4.5 it is possible to work with zero point tables. Other than in most other simulators, the G53 'Machine coordinate system' is pre-programmed as start-up status for the PAL simulator. The unmachined part reference point is also fixed in the machine coordinate system. For milling, the point is X-260/Y-190/Z-200, for turning X0/Z205. The corresponding offset values are also preset for the G54 memory. The consequence is that ...

- ... ... programs are automatically simulated correctly, when the student programs G54 at the program start.
- ... ... programs are simulated offset by these offset values in relation to the unmachined part (thus in the air or in the chuck), if G54 is not programmed.

2.5.2 Exercise / Examination’ menu

In this operating mode, an NC exercise and examination module is contained under <F3> ’Exercise/Examination’ intended to help you to evaluate the knowledge level of the trainees with regard to DIN 66025/PAL and controls.

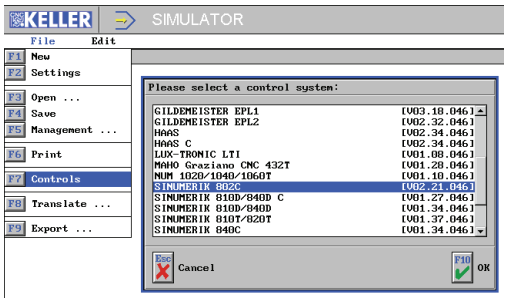
In the ’Teacher’ operating mode you can use NC exercises and NC examinations as classwork. In the NC exercises, each trainee can individually have a look at an evaluation, with NC examinations the evaluation is exclusively made by the instructor. Access to the NC examinations is only possible by a corresponding validation password which can be defined by the instructor.

2.6 ’Simulator’ operating mode

In the following please find information about the ’Simulator’ operating mode, which applies to Turning and Milling.
For this operating mode you need a "guided", control-specific editor including simulation as an additional module. We offer this editor for many control systems. The standard version includes a simulator for the SINUMERIK 802S/C.

2.6.1 Selection of a simulator module

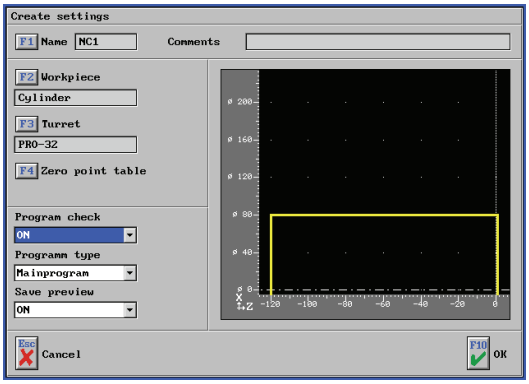
The ’Simulator’ operating mode allows you to simulate different control types when the corresponding simulator modules are installed. Prior to creating a program it might be necessary to select a different simulator module.



Under the <F1> 'File' menu it is possible to invoke a selection list of available simulator modules via <F7> 'Control system'. Mark the type you wish to work with and select by a double click or the <F10> key.

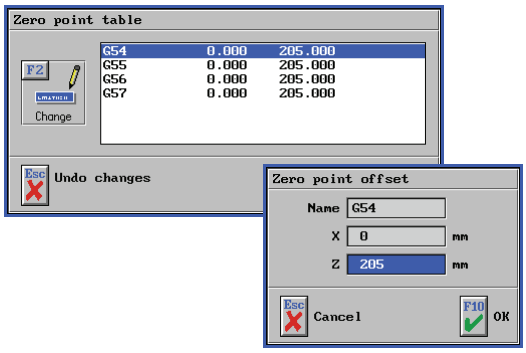
2.6.2 Simulator settings

When a program is already open, the setting dialog for this program appears after invoking <F1> 'File' / <F2> 'Settings'. Only when the start assistant is deactivated by the respective configuration (see Chapt.2.1.5.4), this dialog is directly accessible when creating a new program using <F1> 'File' / <F1> 'New'.



You can assign a name to the program in this dialog window and perform further settings referring to the editor mode and the simulation.
The 'Program check' button can be used to determine whether or not the collision control is active during simulation or carry out technological checks such as, for example, the rotational direction control.

2.6.2.1 Zero point table



In the majority of simulators and in the PAL simulator from version 4.5 it is possible to work with zero point tables. The table is invoked in the setting dialog using <F4> 'Zero point table'. You can edit the offset values of the individual table entries after invoking <F2> 'Change'.
Observe that for turning the Z offset value is the only value that can be changed.

Depending on the control system/simulator, either G53 (machine coordinate system) or G54 (workpiece coordinate system) are preset as switch-on status.
The unmachined part reference point (<F2> Unmachined part in the setting dialog) always complies with the zero point of the preset coordinate system (G53 or G54).

How to use the zero point table:

- If G53 is the switch-on status, an immediate result in the simulation can be seen with values not equal to 0 in the G54 memory and use of G54 in the program. Then it is possible to simulate the reality so that the unmachined part can be nearly positioned freely in the compartment (at X/Y/Z instead of 0/0/0) and taking this position into consideration as an offset for the entries in the G54 memory.

  Remark: This basis offset of the unmachined part in the machine coordinate system can also be preset in a presetting file. This is useful, for example, if you wish to move to the tool changing point with G53. This setting, however, has the consequence, that previous programs might no longer be simulated correctly.
  Please contact out hotline, if you want to carry out such a basis offset.

- If G54 is the switch-on status, entries in the G54 memory do not affect the simulation. However, these values are taken into account when switching over, e.g. to G55 in the program and values different to those in G54 are saved there. Then offset is carried out through the difference between G54 and G55.

Zero point tables are saved simultaneously with each program, same as the unmachined part and the magazine assignment. However, this does not function with programs created with a version older than 4.5. To be able to use a zero point table with previous programs, create a new program, change to the "free mode" of the editor (<F7> 'Editor', see 2.6.4.2), open the old program (<F4> 'Open', see 2.6.6) and copy all blocks of the old program into the newly created program.

2.6.2.2 Raw part description

Some controls permit a dedicated raw part description within the program (HEIDENHAIN, MAHO, ...). In this case, you should ignore the 'Raw part' field here. If you enter a raw part despite of this, *this* raw part and the raw part description are combined to one raw part for the simulation graphics. For control systems that do not provide any raw part description in the program, however, it is practical for later simulation, to enter a raw part here.

From version 3.5, important simulation data such as unmachined part dimension and tools used is saved together with the NC program in a description file with an identical name. From version V4.5, the zero point table is additionally saved. This allows that these NC program are simulated without using the settings.
These description files have the name affix "_sdt", e.g. program name = SIN840D.MPF / description file = SIN840D.MPF._sdt. If an NC program is copied, displaced or deleted within SYM*plus* , the same automatically happens to the corresponding description file. If accessing the NC programs using a different software than SYM*plus* (e.g. Windows Explorer, back-up software, etc.), ensure that these pairs of files are not separated.

2.6.2.3 Tools in the simulation

The default magazine or turret equipment which you set here, form the basis for simulation. If the tool is defined within the program (e.g. for HEIDENHAIN with TOOL DEF ...), please ensure that this definition coincides with the "real" tools of the magazine or turret. Tool definitions in the NC program are not used for simulation.

2.6.3 'NC converter'

From version 4.5 it is possible to use the menu point <F1> 'File' / <F8> 'Translation' / <F1> 'NC output' to translate a program to other control system formats via a post-processor.
The procedure corresponds to that of menu point <F3> 'NC output' in the 'Workplan' operating mode (see Chapt.2.8.2).
Menu point <F1> 'File' / <F8> 'Translate' / <F2> 'Post-processor ...' can be used to create a new post-processor parameter or to change existing parameters. This procedure also corresponds to the behaviour in the 'Workplan' operating mode (see Chapt.2.8.1.3).

NOTE:
During transfer, always G0, G1,G2 and G3 blocks are generated, i. e. cycles etc. are resolved into individual blocks.

2.6.3.1 'Service data' creation in a service case

Should you encounter a problem when creating an NC program that does not suggest an application error but rather a software error, please inform us accordingly. Proceed as described under "'Service data' creation in a service case" on page 32. However, take into consideration, that the 'Export ...' menu is invoked in the "'Simulator' operating mode" using <F8>.

2.6.4 Editing programs

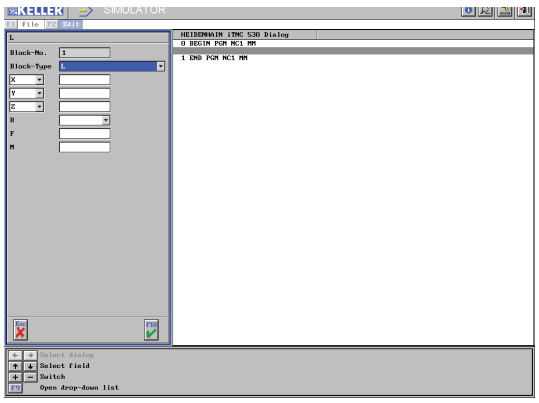
For editing, a 'Guided mode' and a 'Free mode' are available.
In the 'Guided mode' you can enter program blocks with the aid of a dialog window, similar to the PAL*plus* editor, which you as a SYM*plus* user are familiar with from the 'PAL simulator' operating mode.
As such, the 'Guided mode' accepts nothing else but correctly written programs from the syntax point of view. Only this mode offers the possibility to simulate the program.
You should only work in the 'Free mode' if you are very skilled in programming the controls. No simulation is available here.

NOTE:
Please note that a syntax check is made when changing from the 'Free' to the 'Guided' mode (with <F5> 'Guided mode') which might entail reformatting a program (insertion or deletion of free spaces, changed order of addresses, ...).

2.6.4.1 Entry of NC blocks in the guided mode

F2 Process

F1 Create

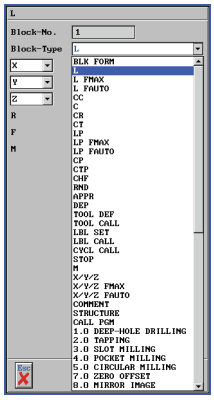


Execute the menu point 'Edit' in order to enter the editing mode.
You can create or insert a new block with <F1> or (more comfortable, but without image) with <Enter> (behind the possibly existing first block).
The blue cursor is now in the 'Block type' field in the dialog window.

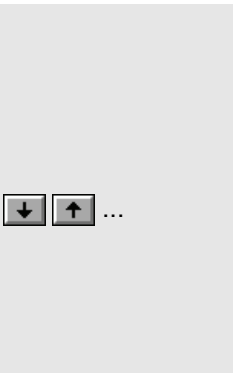
Direct entry,
mouse click or

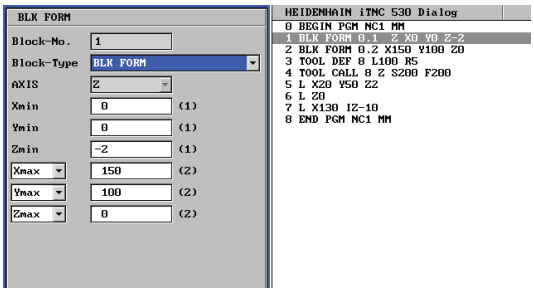
F9 Display selection list

↓ ↑ ...



This selection field contains all path commands, cycles, etc. of the simulator.





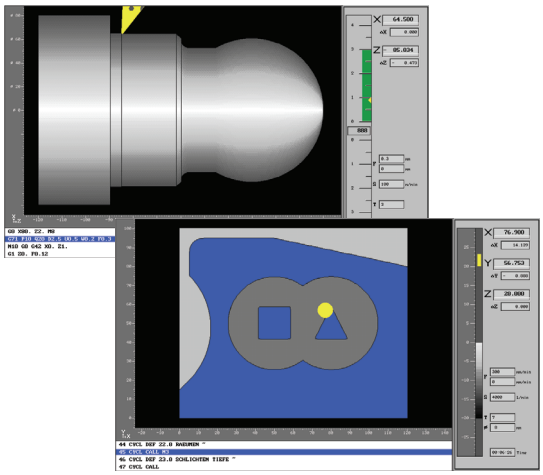
Depending on the controls, it can happen that several program lines are created via one input window (example Heidenhain BLK FORM 0.1 and 0.2).
When "leafing" in the program, the cursor can only be positioned on the first line each of such a block.
<Arrow down> causes the system to jump to the next "self-contained" block.
The function <Insert block after>functions accordingly.

2.6.4.2 Enter NC blocks in the free mode

The "free" editor is a simple ASCII editor ensuring that only those characters can be entered which are admitted for reading into NC controls.
In this editor you can write NC blocks as in a text processing software. You have, of course, to ensure the correct spelling (upper case, lower case, space, etc.).

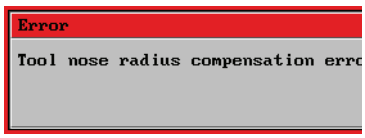
You can change your program as you wish, i.e. insert new blocks. All blocks will be automatically assigned new numbers when invoking the "guided" editors.
The "free" editor only providing limited functions for changing an NC program, you can, of course, use an ASCII editor of your choice. To have the "externally" changed NC program checked for correctness and simulated, you only have to open it anew and invoke the simulation in the guided editor.

2.6.5 Deviation from the original controls system



The simulated traverse paths of a cycle are created, independent of the controls, with KELLER own algorithms, for which reason they generally do not correspond to the actual traverse paths on the machine.
The result obviously geometrically corresponds exactly to the workpiece on the machine.

With the tool radius correction switched on, only contour elements are permitted, which are actually touched by the tool on its traverse path.
Only the NC block following next is taken into account during the block advance.



For example, internal radii smaller than or equal to the tool radius, will generate an error message.
In other cases, contour infringement can occur during simulation.
The radius correction can only be compensated for on a straight line in the machining level.
The functionality of the machine is, of course, not affected by these limitations.

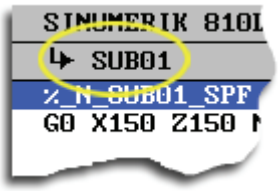
NOTE:
A support of the entire command block of the controls is not guaranteed.
If a program is opened (from external) which contains unsupported commands, the message 'Faulty NC program' appears when selecting the "Guided editor", and the corresponding program line is marked. You can then have the line changed accordingly in the "Free Editor" and the changes checked by switching over with <F7> 'Editor'. The software will not switch over to the guided editor before all lines have been corrected accordingly.

2.6.6 Creating / changing subroutines

Subroutines can be created and changed during the entry of a main program.

When the cursor is positioned on a line containing a subroutine call-up (e.g: SUB01), you can invoke the subroutine for changing using <F4> 'Open'. This will open an existing
(or new) subroutine for editing - in parallel to the main program currently active.

Then the respective subroutine number is displayed in a separate title bar, as displayed here. **Up to 10 subroutines** can be opened simultaneously. The subroutine opened last can always be edited. Apply changes by pressing the <OK> key. Then the program is saved. If you have only viewed a subroutine, you can close it using the <ESC> key.



2.7 'Geometry' operating mode

In the following please find information about the 'Geometry' operating mode which applies to Turning and Milling.

2.7.1 'File' menu

2.7.1.1 'Zero point offset'

The <F1> 'File' /<F2> 'Settings' / <F1> 'Zero offset' menu point is used when a drawing is dimensioned from several reference points and awkward re-calculations can be avoided by corresponding zero offsets.

2.7.1.2 'Contour data export'

The geometry data of the selected contour can be put out in different ways and various formats (also as CAD files) under <F1> 'File' / <F6> 'Export ...' This type of output refers to the provision of geometry data for manual further use, for instance when writing an NC program on the machine.
Two examples:

a) If for instance you have a drawing with a complicated or under-dimensioned geometry, it can be entered using the "Graphical dialog". Then have the geometry data output under <F1> 'File' / <F6> 'Export ...' / <F3> '... as geometry information'. Now you can easily read off all required start and end points as well as circle centre points of the elements and use same, for example, in a manually written NC program.

b) You have written an NC program and all you need for instance are the blocks for a complicated finishing contour. For this purpose, you can have output the geometry data of the selected contour as finished DIN blocks via <F1> 'File' / <F6> 'Export ...' / <F2> '... as NC file'. These can then be inserted in an existing program for example with an editor of your choice and adapted in accordance with the control dialect.

2.7.2 ’CAD input’

External CAD data often needs particular attention. For this reason, you will find the most important information for a successful interplay of your CAD system with SYM*plus* in the following.

Preparing CAD data for the import

The internal mathematic model of SYM*plus* operates with an accuracy of ten digits behind the decimal point. For this reason it is of particular importance to rely on CAD data which is as precise as possible. Therefore, please note the following requirements on external CAD data:

- Make sure that you use the maximum output precision of your CAD system for the creation of CAD data for SYM*plus*.
- A high output precision is of little use when the design lacks precision. Only use geometrically exact CAD data.
- Ensure that the design elements at the transition points to the next element do not intersect but touch precisely at their end points.
- Correct your design by removing superfluous elements. Here, please observe in particular elements present twice and superimposed.
- When you have designed grooves, you need the corresponding centre point lines for treatment in SYM*plus*.
- To ensure that the CAD data import is even quicker and simpler, they should not contain any dimensioning elements.
- Avoid approaching curves by lining.
- When exporting your CAD data, use the AutoCAD 12 format or an older one, if possible.
- Only ASCII formats are read, binary files cannot be imported.

Preparation of CAD data following the import

After the import, all elements are only created two-dimensional in the X/Y coordinate system, all Z information contained in the CAD data is not transferred to SYM*plus*. The depth determination of the contours when milling has to be performed manually following the import.
Although the position of the individual elements and contours in X/Y is defined after the import by values from the CAD file, they are regarded as undefined in SYM*plus*. A zero point has to be defined first, and the elements and contours are then moved accordingly.

2.7.3 Excess precision in drawings / limits of computation precision

Internally, the mathematical background of SYMplus computes with a 10 digit precision. When converting a workplan in an NC program, the values are rounded to 3 digits. Only a 3 digit precision is also available for the contour construction in the ’G1/G2/G3’ operating mode (in SYM*plus*) as well as for writing an NC program in the ’PAL simulator’ and the ’Simulator’.

From the mathematical point of view, circular arcs are always somewhat imprecise (except in special cases, where simple quarter circles are involved). This imprecision could result in the vector model forming the basis of the simulation and crash monitoring, no longer being capable of coping with ’escalating’ rounding errors at some point in the NC program and signalling a (putative) crash.

You can typically trace this supposed crash back to a faulty 3D image. To be able to simulate such a program up to the end, change over the simulation to ’Line graphic’ in the ’PAL simulator’ or ’Simulator’ operating modes in the <F2> ’Edit’, <F9> ’Simulation’, <F3> ’Presettings’ dialog independent of the magazine/turret. All paths are then simulated without tool (and, as a consequence, without slaving the workpiece template, and without crash monitoring) as a line graphic.

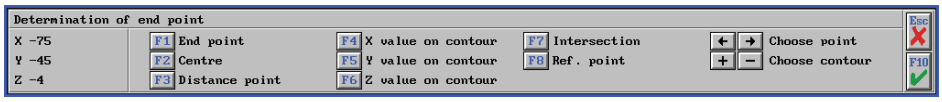
In the ’Work plan’ operating mode, this problem occurs relatively seldom because of the higher computation precision this mode permits - e.g. only when using imprecise CAD data or poorly or over-dimensioned drawings:
To be able to rule out problems generated by rounding errors, you should, if at all possible, always observe the following when entering a "random" geometry:

- Where dimensions with three digits after the decimal point are contained in a drawing, it is better to use more precise CAD data (if available), knowing that these drawing dimensions are frequently rounded and can, thereby, lead to computation imprecision.
  For this refer to the chapter "’CAD input’" on page 30.
- If more precise data is not available, then look for ’genuine’ design dimensions in the drawing (refer to the following example). This holds particularly true for the design of tangential transitions between contour elements. In addition, the 'Determination of end point' (<F11> / <F5> ’Points’) offers the possibility to take over "precise" dimensions from an already existing design.

Example:
In a drawing, a straight line was designed under 20° of X0/Y0 on X100/Y36.397 (and all these dimensions appear in the drawing). Here, for the manual input you should use the design dimensions of the angle and the "straight" end point in X (instead of end point in X and Y).

2.7.4 Design aid by determination of points

SYM*plus* offers a high degree of support with the design of geometries by the high performance end point determination.
As soon as workpiece is open, the determination of end point can be invoked at any time via <F11> / <F5> ’Points’.



The individual functions are described in the following:

- Header — Here the selected function type is shown.
- ’End point’ — Only end points of elements can be selected.
- ’Centre’ — Centre points of linear paths and circles or circular arcs of the active contour can be selected.
- ’Distance point’ — After having entered a value, it is possible to select all points of the active contour located at this distance at a right angle to end points.
- ’... value on contour’ — Following the entry of a value, all points of the active contour can be selected which are located at the entered position of the corresponding straight line.
- ’Intersection’ — All intersections of the active contour can be selected where this contour intersects with other contours.
- ’Reference point’ — A reference point can only be selected when the active contour is not of the ’Random’ type but has been designed using a geometry macro, such as ’rectangle’, ’circle’, ’bore hole’ or ’text’.
- ’All points’ — The points of all functions listed here can be selected without entering a value.
- ’Point’ — The keys <RH arrow> and <LH arrow> are used to move the indication point on the selected contour.
- ’Contour’ — The desired contour is selected using the<+> and <-> keys.

2.8 'Workplan' operating mode

In the following please find information about the 'Workplan' operating mode which applies to Turning and Milling.

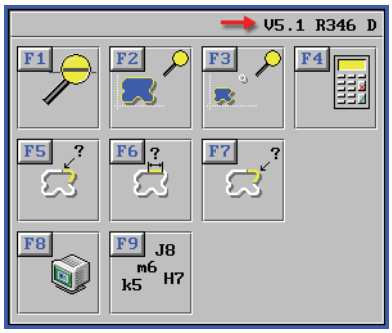
2.8.1 'File' menu

2.8.1.1 Machining status as 'Save geometry'

You can save any desired machining status of a workpiece under <F1> 'File' / <F7> 'Export' / <F1> 'Save geometry' as geometry. This is of advantage when you, for example, premachine workpieces for heat treatment, and then want to use this workpiece as an unmachined part of any shape.
In the 'Geometry' mode, you can, of course, change, print out, put out via the CAD interface etc. the saved geometries.

2.8.1.2 'Service data' creation in a service case

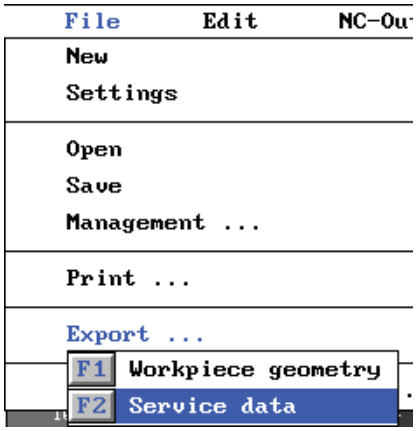
Should you encounter a problem when creating a workplan that does not suggest an application error but rather a software error, please inform us accordingly. Proceed as follows:



Invoke the additional functions and write down the SYM*plus* version number displayed there.

Workplan

F1 File
F3 Open
...
F1 File
F7 Export
F2 Service data

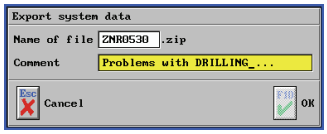


If necessary, change to the 'Workplan' operating mode and open the corresponding workplan (unless already done) in which the error occurred.

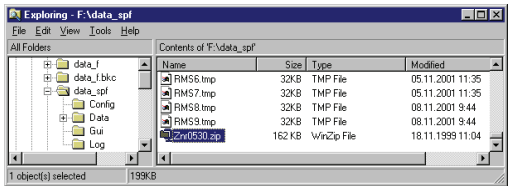
Then invoke the function <F7> 'Export' / <F2> 'Service data'
in the <F1> 'File' menu.

...

F10 OK



You can briefly comment on your problem in the comment line.

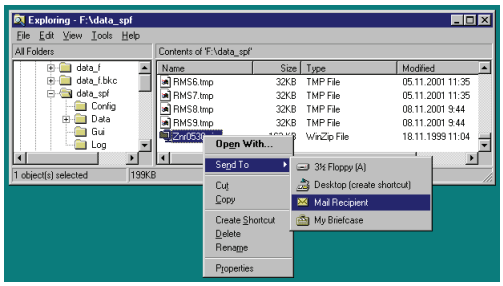


Start

...

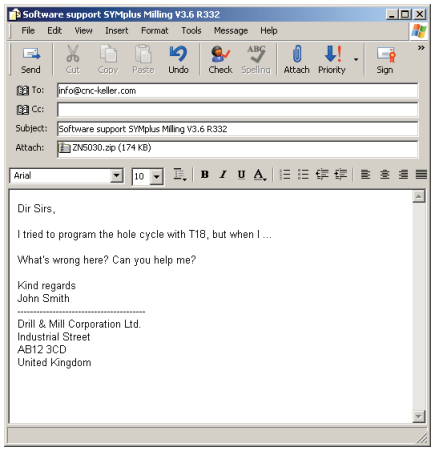
A file with the **ZIP** ending is created and filed in the user data directory.
On the operating system level you can mark this file e.g. in the **Windows Explorer** .....

...



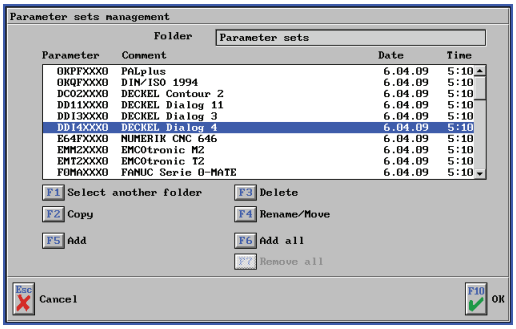
... and transmit it to CNC KELLER GmbH by e-mail using the "RH mouse menu:
info@cnc-keller.com
(Alternative: CD or 3.5" disk by post)

...

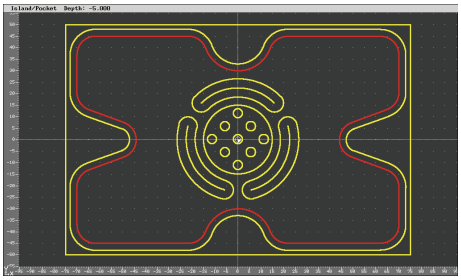


In subject, enter the key word "Software Service", the exact software designation and the version number you have noted before (example: 'SYM*plus* Milling V4.5 R340'), and describe the malfunction and the situation in which it occurs as precisely as possible.

In this manner you contribute to us being able to react quickly and hopefully find a solution.



When the problem is connected to an NC program, generated via a post-processor, please also send the program file and specify the post-processor designation ('Parameter block') used to generate the program (Example: 'DDI4XXX0 DECKEL Dialog 4 06.08.07 4:05').



When a mere geometry problem is concerned, please mail the corresponding workpiece file with the WS1 ending and fax us the attendant drawing, if necessary.
Fax No.:0202 4040-99

2.8.1.3 Postprocessor parameter adaptation

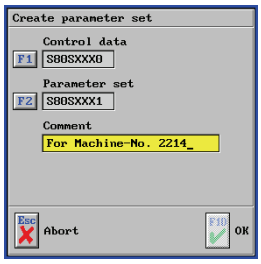
The NC output format of the post-processors is influenced with the post-processor parameters. When your controls need special characters or formats, these can be set as described in the following.

NOTE:
Creation and adaptation of parameter sets and post-processors is more or less complicated, depending on the type of controls or machine. If the required specialist knowledge is not available in your company, you can have the necessary work performed by CNC KELLER GmbH as a customer service.

For all following menu points on the post-processor parameters subject refer to the 'Workplan' operating mode under <F1> 'File' / <F8> 'Post-processor ..." or the 'Simulator' operating mode (see Chapt.2.6.2) under <F1> 'File' / <F8> 'Translate ...' / <F2> 'Post-processor ...'.

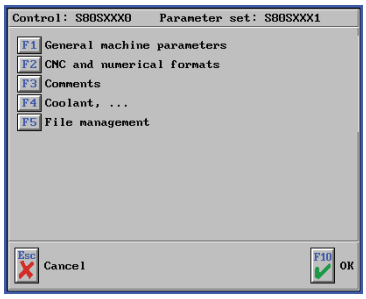
Create new parameter set

A new parameter set is created with the <F1> 'New' function. For this purpose, the underlying control family has to be selected first in the following dialog window with <F1> 'Post-processor'. <F2> 'Post-processor parameter set' is then used to define a name for the new parameter set. In the commentary line a further text can be entered, which is displayed when opening and managing the parameter sets.



Dialog window 'Create parameter set'

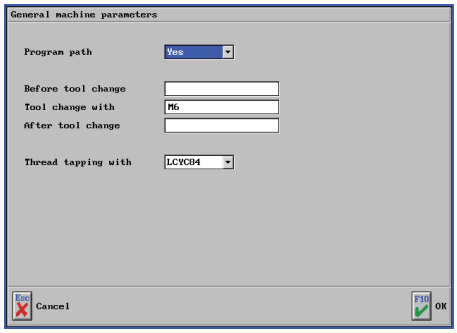
Once the 'Create parameter set' was closed, the details of the new parameter set can be defined. Fields with a grey background cannot be changed. They can only be changed by CNC KELLER GmbH directly in the post-processor. Upon request, we will create a customised solution for your control system.



Dialog window 'Parameter settings'

General machine parameters

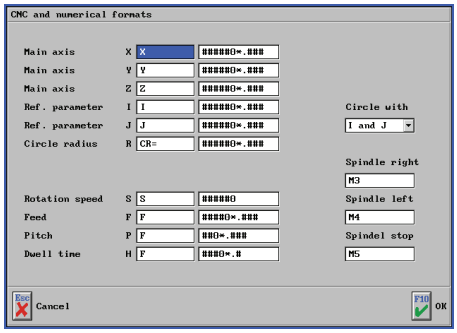
Here a post-processor specific tool change point is set for some post-processors. With these post-processors, the change point, the co-ordinates of which are entered during 'Clamping', is used as an intermediate point before and after moving over the tool change point. In this case, the coordinates of the post-processor represents the actual tool change point. For turning, the value for a speed limitation can be set to maximum speed.



Dialog window ’General machine parameters’

CNC and numerical formats

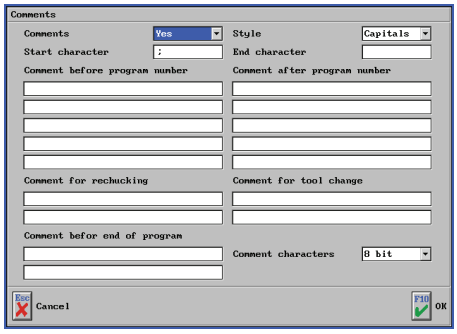
Here the addresses (letters) and the number formats for the individual commands can be set. Some post-processors also allow the output of circles with radius information (e.g. `R`) instead with `I` and `J` or `I` and `K`.



Dialog window ’CNC and numerical formats’

Comments

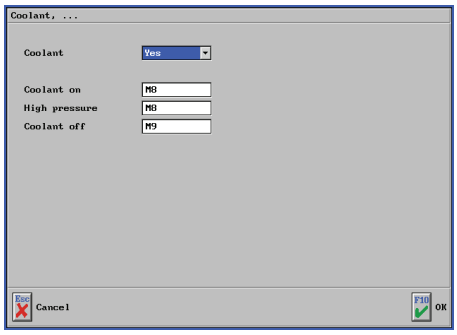
At this point you define the additional comments, which appear within the NC program.



Dialog window ’Comments’

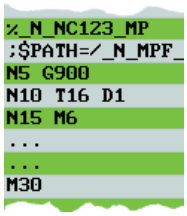
Coolant

Here the M functions for the activation and deactivation of the coolant supply are defined. If the machine does not provide a specific setting for high-pressure coolant, set the same M function for high pressure as for the coolant.

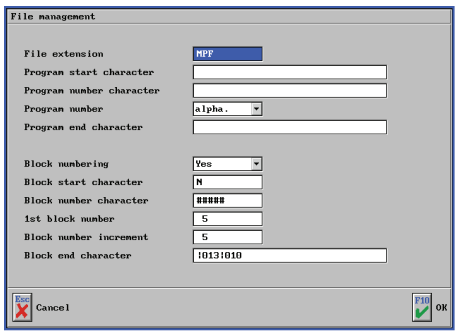


Dialog window ’Coolants’

File management



The figure below shows an example for the settings in the file management for NC programs. In the example, an NC program with the name NC123 is to be put out:
The set values would create an NC program with the ending `.mpf`, combined `nc123.mpf`.
This NC program would then have this structure.



Dialog window 'File management'

Editing

An existing parameter set can be selected and changed with <F2> 'Change'. The way of proceeding is similar to that described for 'Create parameter set'.

2.8.2 'NC output' menu

Upon the selection of a post-processor parameter block, you can have the system generate an NC program for your controls from the current workplan.
From version 3.5, important simulation data such as unmachined part dimension and tools used is saved together with the NC program in a description file with an identical name. This allows to immediately simulate these NC programs in the 'Simulator' operating mode, if a corresponding control system simulator is installed, without using the settings. These files have the name affix "_sdt", e.g. program name = SIN840D.MPF / description file = SIN840D.MPF._sdt. If an NC program is copied, displaced or deleted within SYM*plus* , the same automatically happens to the corresponding description file. If accessing the NC programs using a different software than SYM*plus* (e.g. Windows Explorer, back-up software, etc.), ensure that these pairs of files are not separated.

2.8.2.1 'Set-up sheet'

If you need a set-up sheet as a planning basis or as an aid to make your machine ready, then invoke this function. The saved set-up sheets can be opened in the 'Transfer' operating mode in the 'free editor'.

2.9 'Teacher' operating mode

This operating mode is password-protected, and its access should normally be reserved to the instructor.
The main difference between an exercise and an examination is to be seen in that a detailed evaluation of an examination is only performed by the examiner and that errors cannot be corrected any more. An average of the skills of the entire class can be derived automatically from the individual results.

2.9.1 Password, 'Teacher' operating mode

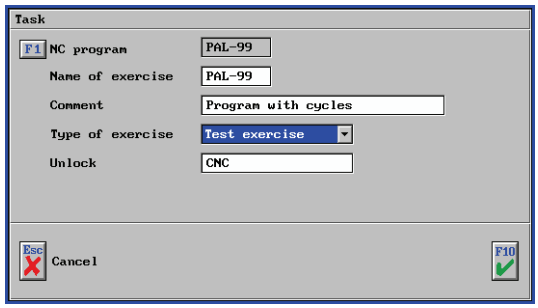
The 'Teacher' operating mode can only be accessed after having entered a corresponding password. The objective is to prevent unauthorised or unintentional access on part of the user. **'123'** is factory-set as a password. You can change the password within the operating mode.

2.9.2 'File' menu

Creating a new exercise or examination

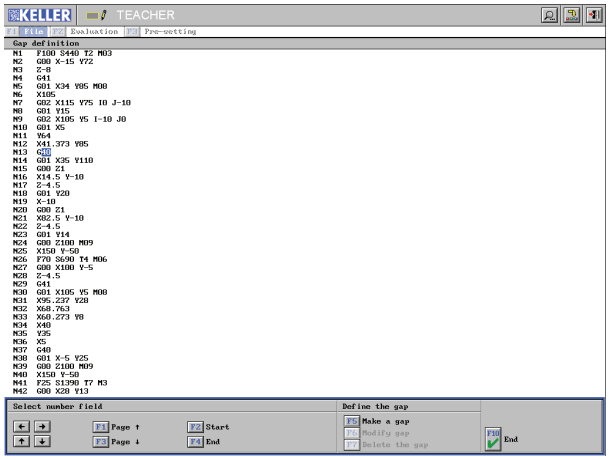
To create an exercise or examination, please proceed as follows:

1. In the 'PAL simulator' operating mode, create a corresponding NC program for which you wish to create a task.
2. Invoke the 'Teacher' operating mode and enter the password.
3. You create a new task with <F1> 'File' / <F1> 'Create new'.
4. Use <F1> 'NC Program' to select the NC program created by you.
5. Assign a name and a comment for this task.
6. Select if the task is meant as an exercise or an examination. Following this, for an examination a password has to be assigned for application. This prevents unauthorised and premature access to the examination contents,
7. Terminate this dialog with <F10>.

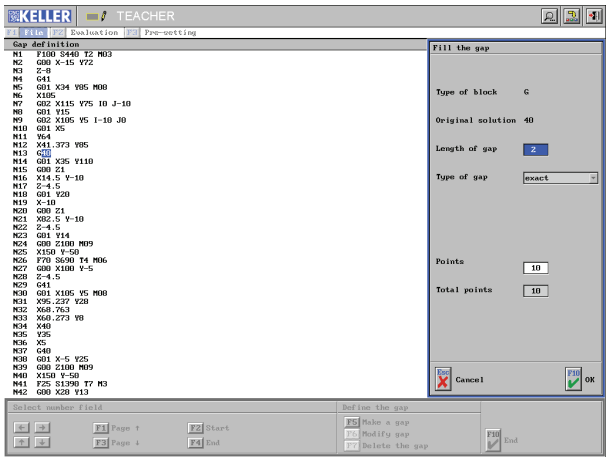


Create tasks

1. Use the <arrow keys> to navigate the blue mark to the function for which you wish to create a gap.



2. Press key <F5> 'Make a gap'.
3. Define the 'Length of the gap' and the number of 'Points'.
4. The dialog is terminated with <F10> 'OK'. Following this, further gaps can be defined.

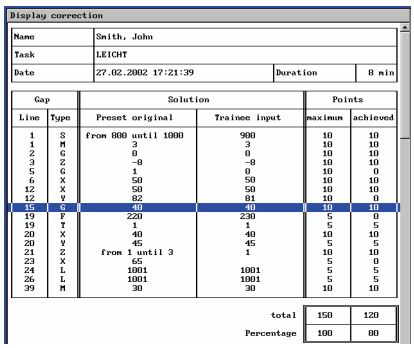


The following types of gaps can be created:

- 'exact' With G and M functions, obviously only exact values are admitted.
- 'Range' Useful for F or S values to define a range.
- 'Tolerance' You can define a permissible tolerance range.

2.9.3 'Evaluation' menu

Exercise evaluation



The detailed evaluation of an exercise is made automatically after having terminated the inputs with <F10> individually for each trainee examined. This result can be saved afterwards. In addition to the evaluation, the trainee examined is offered additional partial text information.

Examination evaluation

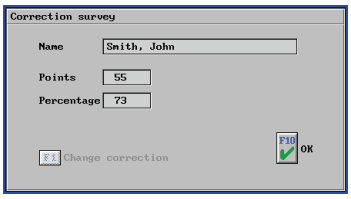
The detailed examination of an exercise is exclusively performed by the examiner who requires the floppy disks of the trainees examined, on which the corresponding examination contents have been saved.

1. Invoke the 'Teacher' operating mode. There, you have to select <F2> 'Evaluation' / <F4> 'Read-in floppy disks'.

2. After having selected a suitable folder for the solutions (evaluations), you are requested to insert the first trainee floppy disk. If a corresponding floppy disk is in the drive, confirm this dialog with <F10>.

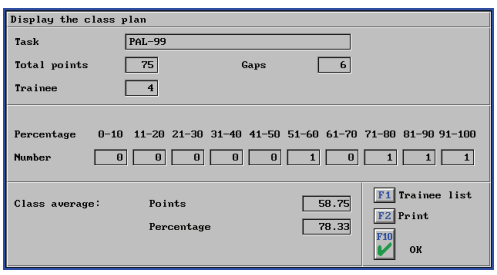
NOTE:
The floppy disks must not be write protected because each solution is marked directly upon being read in. Double read-in and, as a consequence, multiple evaluation is thereby excluded.

3. Having read-in the solution, a brief evaluation is displayed.

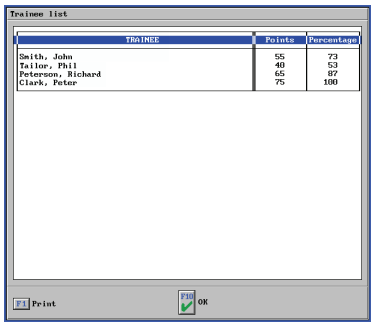


4. Following this, you are requested to insert further floppy disks with solutions, if available.

5. Once all floppy disks have been read in, a class review ...



...and a review of each individual trainee tested appears.



2.10 'Set-up' operating mode

In the following please find information for the 'Set-up' operating mode which apply to Turning and Milling.

2.10.1 'Tools' menu

Here you can create any number of tools, such as in a tool distribution.

2.10.2 'Magazine' or 'Turret' menu

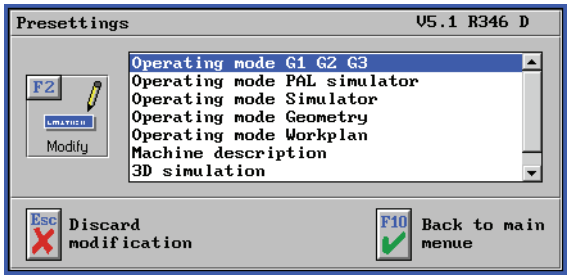
Here any number of magazine or turret equipment matched to the individual machine can be entered.

2.10.3 'Material' menu

The materials entered in this list are used for technology assignment of the individual tools. If you then set a certain material when creating a work plan, the systems automatically offers you the corresponding technologies for the tools in the individual work steps.

2.10.4 'System configuration' menu

2.10.4.1 'Pre-settings'



Presettings can be made for the individual operating modes, the machine data to be used and printing.

'Operating modes'

The presettings for the individual operating modes determine the settings used to create a new file. These default values can be respectively adapted in the corresponding operating mode under <F1> 'File' / <F2> 'Settings'. Separate presettings can be carried out for each available simulator modul under 'Operating mode simulator'.

'Machine description'

In the 'Machine description' you specify the key data, e.g. dimension of the machining compartment, to be used by SYM*plus*.

'Simulation'

- 'Simulation' option — You decide here whether or not actual values are to be displayed during 3D simulation. The 3D simulation is displayed larger when no actual values are displayed.
- 'Machine' option — Use this option to set the machine type to be used mainly in the 3D simulation.
- 'Simulation type' option — Decide whether or not an inquiry for 2D or 3D simulation appears prior to each simulation. You can also determine one of the two types here, the two simulation types, however, offer different information. Thus, the 2D simulation, for example, shows a graphical block preview in the individual block and the 3D simulation checks the machining compartment for collision, etc.

'Print'

Due to the fact that printer types are different, SYM*plus* allows you to make certain pre-settings for your print-out.

- Option 'Output' — Here you make the general decision if the information is to be put out on a printer on in a file.
- Option 'Print file' — If you wish to put out the information in a file, you can enter a corresponding file name (without file extension) here.
- Option 'Font size' — Here you enter the font size. This has an influence on the number of characters in a line that will still fit in the print area of the printer. If some or all lines of the print out exceed the right-hand paper edge as a result of the set font size, change the font to an appropriate smaller size.
- Option 'Lines per sheet' — Here you enter the number of text lines which your printer is able to print out on one page (depending on the font size).
- Option 'Title line' — Each print-out is assigned a title line. Here you can define if you want to have the date and a time printed out, too.
- Option 'Set-up sheet' — Here you determine, if when printing a set-up sheet only the data of the tools used in the current work plan or all tools loaded in the corresponding magazine or turret are to be put out.

NOTE:
If you re-route the print output into a file, the file is automatically filed by the operating system. It might happen that the file is not filed in the folder of the corresponding SYM*plus* software. In this case, please use the search function of your operating system to find the corresponding file.

2.10.4.2 'Storage location for files'

Data exchange and data safety are important issues for the majority of users. For this reason the following details how to arrange folders in SYM*plus* within the user data (internal) and outside of the user data (external, for instance on a server or a removable drive). Furthermore you learn which files of the user data directory contain the important data of your work environment. This allows you to see at a glance which files should be subject to a regular backup.

2.10.4.3 Using USB memory sticks

We are not able to deliver the product pre-configured because Windows operating systems dynamically allocate drive letters for removable drives, e.g. USB sticks. For this reason, please find an instruction for creating paths for removable drives in the following

1. If all PCs with Microsoft Windows® 2000 / XP used for SYM*plus* have identical hardware constellations and the drives allocated by the server are also identical, the same drive letter should automatically appear on each PC for the connected USB stick. In this case, you can directly continue with point "Changing the folder paths in the CNCPLUS.DAT file" on page 42.

2. If all PCs with Microsoft Windows® 2000 / XP used for SYM*plus* have different hardware constellations and/or different drive letters allocated by the server, continue with point "Setting the drive letter in Microsoft Windows® 2000 / XP" on page 41.

3. When using PCs with Microsoft Windows® 98SE or Me for SYM*plus*, continue with point "Setting the drive letter in Microsoft Windows® 98SE / Me" on page 41.

Setting the drive letter in Microsoft Windows® 98SE / Me

Prior to using USB sticks under Microsoft Windows® 98SE / Me install the respective USB drivers first. For this purpose, please contact the manufacturer of the respective USB stick.

Subsequently continue with point "Changing the folder paths in the CNCPLUS.DAT file" on page 42. Enter the respective drive letter for the removable drive individually for each PC.

Setting the drive letter in Microsoft Windows® 2000 / XP

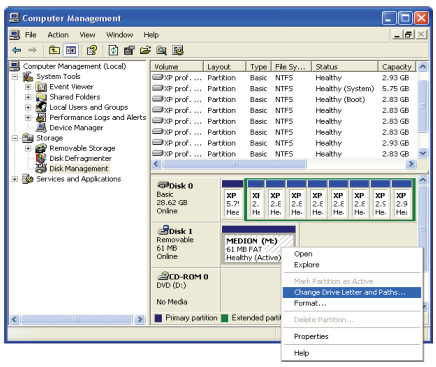
The following **example** shows how to proceed to manually allocate a fixed drive letter to a removable drive in **Microsoft Windows® XP**:

1. For this purpose, log into the PC with administrator rights.

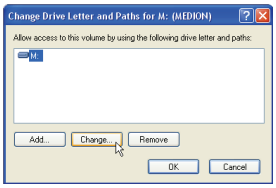
2. Now connect the USB stick to the PC.

3. Click on *Start* / *System controls* / *Management* / *Computer management*.

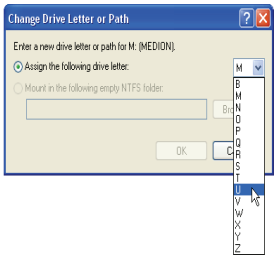
4. Click on *Data carrier managment* and then use the RH mouse key to mark the drive corresponding to the USB stick to open the context menu. Here select the *Change drive letters and paths* option.



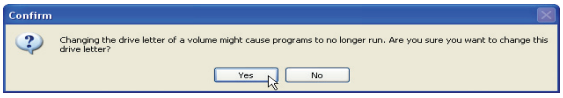
5. Select the drive letter and then click on *Change*.



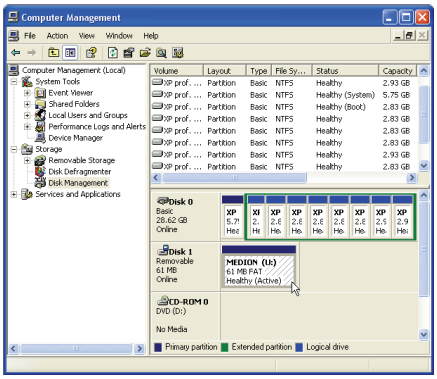
6. Select the new desired drive letter and close the dialog.



7. Confirm the following message by clicking on *Yes*.



8. In this example, the drive letter was successfully changed from `G:` to `U:`.



Now SYM*plus* must be allowed to access the new drive letter. For this reason, continue with 'Changing the folder paths in the CNCPLUS.DAT file'.

Changing the folder paths in the CNCPLUS.DAT file

Prior to accessing the files saved on a USB stick, respectively adapt the `cncplus.dat` file. These are saved within your user data in the `data` folder.

You can easily create the USB path specifications within SYM*plus* by means of the folder administration. For this, please also refer to "Folder management" on page 43. If you want to adapt the `cncplus.dat` file directly using a text editor, adhere to the following instructions.

1. Open the `cncplus.dat` file with an ASCII editor (e.g. with the `edit cncplus.dat` command in the input request). Now several key words appear in square brackets, the so-called sections. Below you can see the corresponding entries for the different storage locations. These entries are always structured according to the following scheme: `'Path = Synonym'`.

2. Now enter the path information and the desired designation beneath the *[Workplans]* section which will then be displayed in SYM*plus* according to the following model:

```
@Drive:\Folder\...=Any designation with max. 36 characters
```

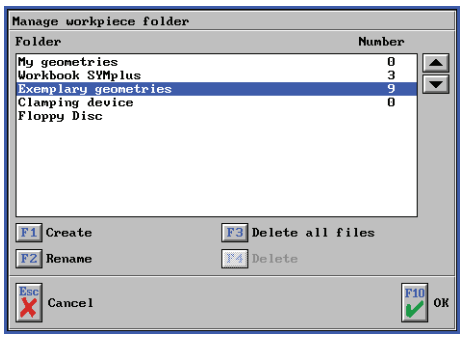
According to the previous example 'Setting the drive letter in Microsoft Windows® 2000 / XP', the new entry in the cncplus.dat should be displayed as follows:

```
[Workplans]
USER.AP1=Own work plans
DEMO.AP1=Example work plans
@A:=Disk
@U:=USB stick
```

3. Once this file is stored, you can immediately use the newly entered folder within SYM*plus*.

Folder management

To be able to conveniently create or delete folders for saving your user data, the 'Set-up' operating mode includes the <F2> 'Storage location for files' menu point under <F4> 'System configuration'.

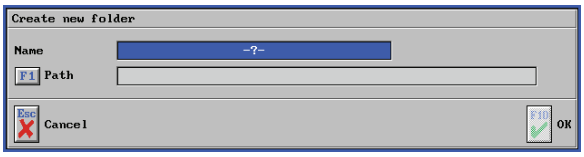


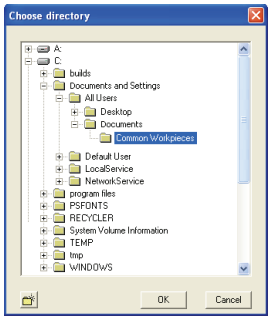
Here you can create or delete new folders for the individual data types of the individual operating modes.

ATTENTION:
The <F3> 'Delete all files' function only deletes the file types corresponding to the selection, e.g. workplans. However, when using the <F3> 'Delete all files' function for NC programs, ALL files contained in the respective folder or drive are always deleted. For this reason, we recommend to only directly connect drives, e.g. on USB sticks, when no further data exists on these drives.
For data storage in SYM*plus* only use sub-folders and not the root directory of hard disks!

You can select the order used to display folders when opening or saving a file. Mark the respective folder and displace it to the desired location using the two arrow symbols.

You can then add a further folder for instance using the <F1> 'Create' function. If only one designation is entered for a new folder, it is automatically created within the user data (also refer to "Installation Folder" on page 4 in the installation instructions).





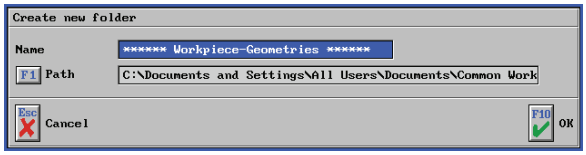
<F1> 'Path' can be used to select an existing folder or to create a new folder in a targeted manner.

Select an existing folder
or
click on the  symbol to create a new folder.

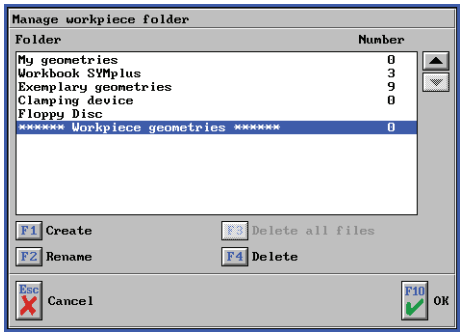
Once the folder has been created, select it first and subsequently take over the dialog.

NOTE:
It is, of course, possible to distribute the folders on any number of computers of your network or on various drives within your PC.
Important is, however, that the corresponding folder can be accessed via a path name supplemented by a drive letter (`Drive:\Folder name`).
UNC path information, such as `\\Server name\Validation name` is **not** permissible!

Finally enter a suitable designation for the selected folder.



NOTE:
Please note that the name of a folder is limited to a maximum length of 36 characters. Otherwise the name may not be correctly displayed in some dialogs.



Now the selected folder is available in the selection list of the corresponding "Open" and "Save" dialogs. If desired, the folder can be sorted according to a different order.

Data backup

If you wish to proceed with a data backup of your user and configuration data, all you have to do is only backup the folder containing your user data and the two files named below from the system data folder. If, however, you only backup or transfer to a different PC the configuration data for example, you only require the files listed below:
User data:

- `gke.pre` Geometry settings
- `material.dat` Material data base
- `ncserver.ini` Machine settings
- `vorein.ini` System settings
- `cpmain.ini` Start options
- `*.ini` all further setting files
- `wkzdb` Section 1 of the tool data base
- `wkzdb.ldb` Section 2 of the tool data base

System data:

- `mod.ini` Settings for the restricted access rights
- `emod.ini` Settings for full access rights

File management

The dialogs for opening and saving files in SYM*plus* differ completely from those known from other Windows programs. At the beginning, this might be somewhat confusing, but in practice you will quickly learn to appreciate it:

- Only those files and folders are displayed that are of significance for the individual situation.
- Exponential designations can be assigned to the individual folders. As a result, you do not have to remember complicated path names. In this manner it is very easy to assign the files to e.g. individual projects.
- When a different folder is selected, the number of files of the desired file type stored in the corresponding folders is displayed behind the folder name.
- SYM*plus* "remembers" exactly for each operating mode in which folder work was performed last, proposing this automatically again the next time opening or saving is performed.

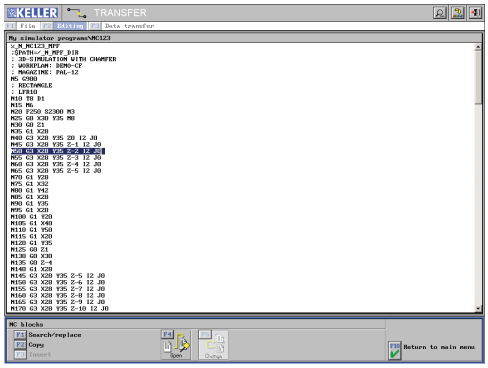
NOTE:
There is a corresponding file management dialog in each operating mode. In this dialog, you can perform the usual file operations, such as for instance 'Copy', 'Rename' or 'Delete' files.
The 'Management' menu point is included in **every** operating mode under the first main menu point <F1> ...

2.11 'Transfer' operating mode

In the following please find information about the 'Transfer' operating mode which applies to Turning and Milling.

2.11.1 'Edit' menu

SYM*plus* is equipped with an ASCII text editor. It can be used to change and create NC programs and set-up sheets, etc. Due to the fact that no support can be invoked during the entry, it is named **free editor**.



The ASCII format of this editor ensures that the characters entered by you will be "read and understood" by the CNC controls.

Its behaviour is identical with that of well-known standard editors, text can be marked, copy and paste by means of the Windows key combinations <Ctrl><C> and <Ctrl><V> is (as an alternative to the function keys) possible.

In addition to the displayed functions, there are several function keys to facilitate your working with this editor:

- <Enter> Insertion of a new line at the cursor position and termination of the input in a line
- <Del> Deleting the character at the cursor position
- <Pos1> Positioning the cursor at the beginning of the current line
- <End> Positioning the cursor at the end of the current line
- <Ctrl><Pos1> Positioning the cursor at the beginning of the first line in the file
- <Ctrl><End> Positioning the cursor at the end of the last line in the file
- <Scroll up> Scrolling upwards page by page
- <Scroll down> Scrolling downwards page by page
- <Ctrl><Scroll up> Positioning the cursor at the beginning of the first line on the current page
- <Ctrl><Scroll down> Positioning the cursor at the end of the last line on the current page

2.11.2 'Data transmission' menu

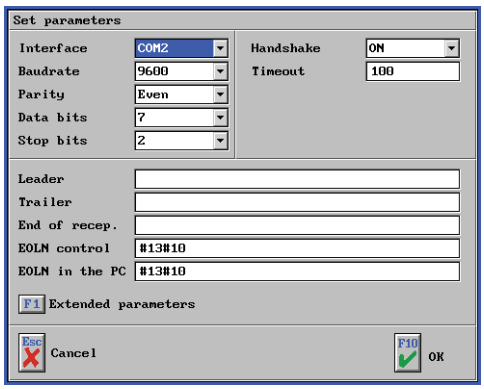
2.11.2.1 Setting the transmission parameters

NOTE:
Perfect data transmission indispensably requires the correct setting of the transmission parameters. As an example, a sample file is supplied for the transmission parameters. However, the parameters have to be individually set to the corresponding control system and saved as a new parameter set.

Opening parameter blocks

Parameter sets already saved can be opened and changed with <F3> 'Data transfer' / <F3> 'Parameters ...'. From the list of parameter blocks you can now select a parameter block and open it with <F10> using the direction keys.

Transmission parameters



Serial interface

For data transmission, one of the installed serial interfaces (mostly COM 1 or COM 2) of the PC can be selected. Please note that the plug for the cable connection is also connected to the corresponding interface.

Baud rate

The transmission speed can be set in increments between 110 Baud (bits per second) and 19,200 Baud. Transmission can only function when the Baud rate, the number of data and stop bits and the parity of the transmitter and receiver are identical. For longer cable connections (longer than 20 m) a slower transmission speed is to be recommended, otherwise transmission faults can occur.

Parity

Parity settings can be changed between 'Even', 'Odd' and 'None'. Typically, a transmission is made in the ISO code with an even parity.

Data bits

Transmission with 7 data bits takes place on almost all CNC control systems. The number of data bits can be set between 5 and 8.

Stop bits

Depending on the requirements of the controls, a selection of between 1, 1½ or 2 stop bits can be made.

Handshake

The **hardware** handshake is used in the **ON** position. In addition to the data lines, also the lines validating the transmission from the transmission program are evaluated. In this manner, the transmitting and receiving devices can wait for each other. When the handshake is **OFF**, data can be transmitted without being sure that the data sent is actually accepted by the receiving device. In other words, there is the danger of a data loss. Therefore, without handshake the Baud rate selected should not be too high. The maximum transmission speed depends on the speed of the receiving device.

Timeout

At the end of the transmission of a program to the PC, the program end is automatically recognised when the transmission parameters are set accordingly. Should an automatic recognition of the transmission end not be possible, the transfer module waits for several seconds until the transmission is ended. The length of the waiting time is set with the Timeout (Unit: Seconds).

Leader

Prior to starting the actual CNC program, some control systems wait for certain characters indicating that the program start will be correctly recognised. These ASCII signs can be entered here. The indication of the ASCII characters is either made in decimal (e.g. #13) or hexadecimal (e.g. $7E). Several characters in succession are separated by a comma.

Trailer

As described under 'Leader' ASCII characters for the trailer can be entered here.

EOLN control system

Here the ASCII characters for end of line can be entered in the order in which they are to be transmitted to the controls. CR, LF are typical for an end of line. The entry of the ASCII characters is then: #13,#10.

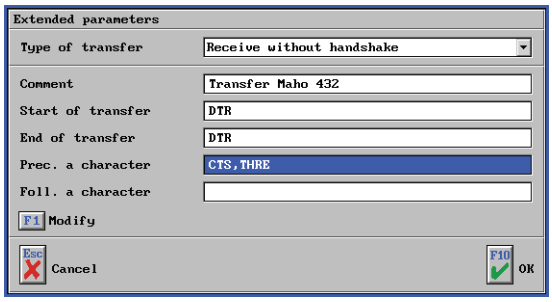
EOLN PC

Here the ASCII characters for end of line can be entered in the order in which they are to be saved in a file on the PC. CR, LF are typical for an end of line. The entry of the ASCII characters is then: #13,#10.

Extended transmission parameters

Extended parameters

Further sequences can be defined for setting control signals when transmitting and receiving using the function <F1> 'Extended parameters'. The change of these settings requires specialist knowledge. For this purpose, contact CNC KELLER GmbH. Telephone: +49(0)202-40 40-0.



Should you wish to change the extended transmission parameters, the following is a brief explanation of the meaning of a few extended parameters by way of examples.

RTS

This line is set to **HIGH** level indicating that the PC is ready to receive data.

-RTS

This line is set to **LOW** level, the PC thereby indicating that it is no longer ready to receive a character because characters received still have to be processed.

DTR

This line is set to **HIGH** level indicating that the PC is connected.
A typical sequence for **Transmitting** with Handshake looks as follows:
Before a character: CTS, THRE
The CTS signal is interrogated for as long as it is set by the controls. Subsequently the THRE signal is interrogated until it is set. The serial module is then empty and can be written on.

A typical sequence for **Receiving** with Handshake looks as follows:

Start of transmission:	RTS & DTR	(set RTS and DTR)
End of transmission:	-RTS & -DTR	(delete RTS and DTR)
Before a character:	RTS	
After a character:	-RTS	

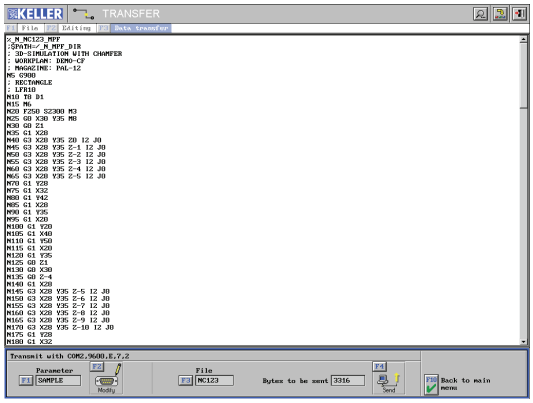
To receive an individual character, first the transmission of character is validated by the controls with RTS. Then the RTS signal is deleted with -RTS which entails a brief transmission stop to allow the transmission module sufficient time to process the character received.

2.11.2.2 Transmission of NC programs

NOTE:
To perform a data transmission, make sure that a corresponding cable is available between the PC and the machine controls and that the parameters for data transmission are identically set on the PC and on the controls. Should these pre-requisites not yet be fulfilled follow the instructions given in chapter '2.11.2.1 Setting the transmission parameters'.
For the new installation of the data transfer, you can also contact the CNC KELLER GmbH service.
For the new installation of the data transfer, you can also contact the CNC KELLER GmbH service

Transmission of data

To transmit a program, invoke <F3> 'Data transmission' / <F1> 'Send' first.



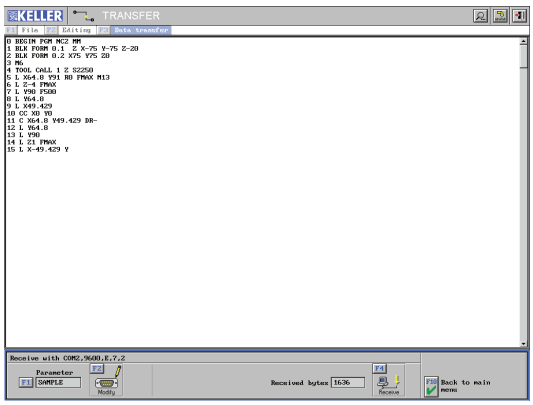
Then use <F1> to select the parameters for data transmission. You can view at and, if necessary, adapt these parameters using <F2>. Function <F3> is used to set the program to be transmitted. This program is also displayed in the top section of the window and can thus be checked again prior to starting transmission.

Now bring the CNC controls at the machine in a receiving status. The transmission of the NC program is started with <F4> followed by <F10>. At the beginning, the display 'Bytes to transfer' shows the file size. During transmission the bytes still to be transmitted are constantly updated.

2.11.2.3 Receiving NC programs

Receiving data

To receive a program, invoke <F3> 'Data transmission' / <F2> 'Receive' first.



Then use <F1> to select the parameters for data transmission. You can view at and, if necessary, adapt these parameters using <F2>.

The transmission of the NC program is started with <F4> followed by <F10>. Now the PC waits for data from the controls. Start the NC program transmission to the controls now. The number of bytes received is displayed under 'Bytes received'. During transmission this display is constantly updated and the bytes received are displayed in the window.

The end of transmission is automatically recognised by the PC, provided that the setting of the transmission parameters is correct. Otherwise, the PC waits for several seconds before it signals Timeout. In this case, the receipt has to be ended with <F10>. Now define a name under which the NC program received is saved. Once it has been saved, you can view and change the program using <F1> 'File' / <F2> 'Open' and <F2> 'Edit'.

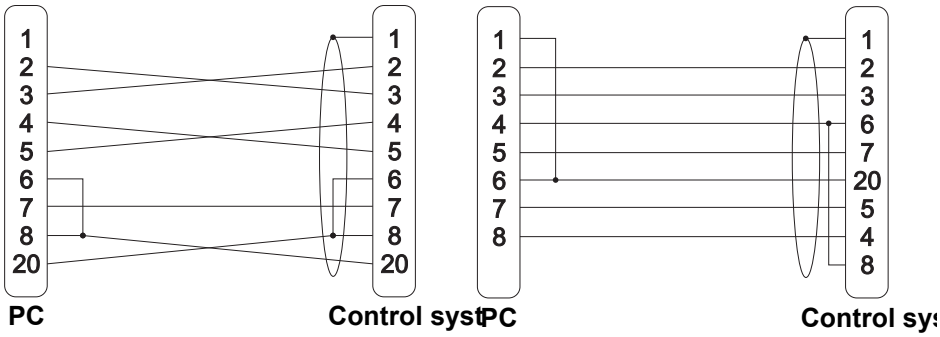
2.11.2.4 Meaning of the signals

DSR	The controls can use this signal to indicate that it is connected.
CTS	This line is used by the controls to signal its readiness to receive data.
DTR	The PC can use this signal to indicate that it is connected.
RTS	The PC uses this line to signal its readiness to receive data.
THRE	The send register of the serial module is empty and can be written on.
TEMT	When this signal is set, the serial interface has sent a character.

2.11.2.5 Transmission cable

Perfect data transmission between the computer and the controls calls for a transmission cable which meets all requirements of the V24 interface of the controls and the computer. Unfortunately, the various controls require different cables, for which reason no generalised cable assignment can be suggested. The cable illustrated in the following picture, however, meets the standard V24 interface and can be used for most of the modern control systems.

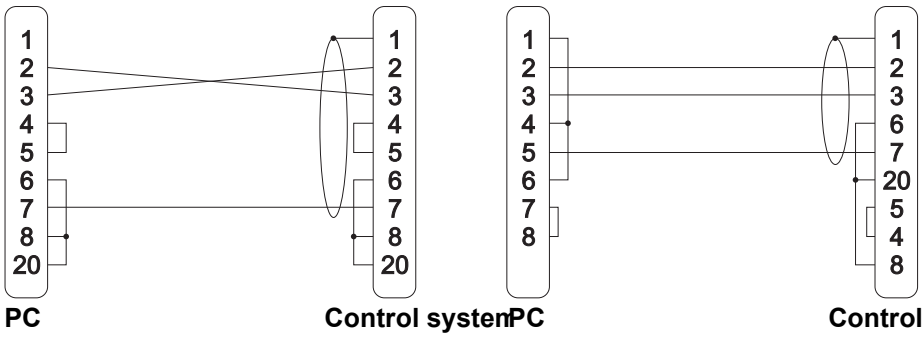
This seven strand connection also uses handshake conductors which are addressed by the majority controls and by the transfer module.



V24 data transmission cable for 25 and 9 pole connectors

If the handshake conductors are not to be used, these can be bridged, or the handshake log is deactivated in the software. With a 3 strand connection without handshake conductors, however, this implies the risk of data loss at high Baud rates.

NOTE:
The following is the connector assignment for data transmissions without hardware handshake.
If the handshake conductors are not to be used, these can be bridged, or the handshake log is deactivated in the software. With a 3 strand connection without handshake conductors, however, this implies the risk of data loss at high Baud rates.

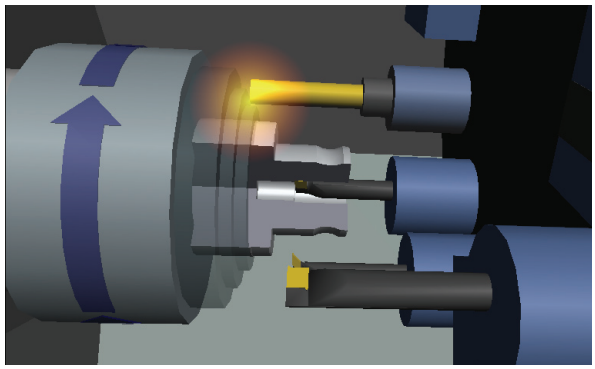


V24 data transmission cable for 25 and 9 pole connectors **without** hardware handshake

2.12 Simulation

After invoking the simulation with <F9> 'Simulation', you can select between a 2D simulation and a 3D simulation. Use <F3> 'Presettings' to influence the behaviour of the two simulation types.

2.12.1 Differences in the collision detection between 2D / 3D simulation

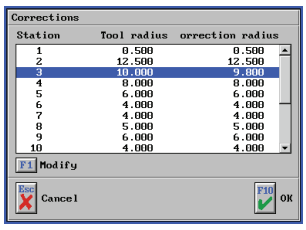


In opposite to the 2D standard simulation, the 3D simulation is "familiar" with the machining compartment, the workpiece and its clamping means and the tools. For this reason, all movements, regardless of traverse path or turret swivelling, etc., are checked for possible collisions - for even more safety. In the event of a collision, the simulation is stopped and the point of collision is lit in addition to the respective message. In this example, the drill on the adjacent station collides with the turning chuck during internal machining.

2.12.2 2D simulation

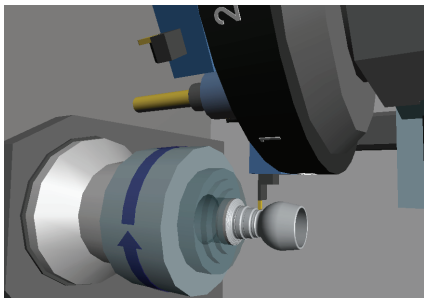
For milling, you can select between 'Tools', 'Precision correction' and 'Line graphic' and for turning between 'Tools' and 'Line graphic'.

- The 'Tools' setting stands for an erasable graphic with the tools created in the 'Setting' operating mode and for the possibility to check programming by way of the 3D image.
- The 'Precision correction' setting offers the possibility to simulate the effects of the interaction of a real tool radius and the tool radius set in the control system. In this way, it is possible to display the effects of deviations in the radius correction values in relation to the real tool radius to the programmed workpiece contour.

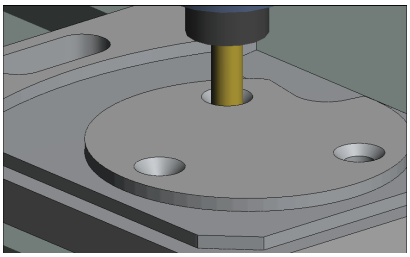


- The 'Line graphic' setting is of particular advantage if you wish to work time-savingly, knowing that the simple simulation is built up much quicker. Moreover, the 'Line graphic' can be a help if simulation with the 'Tools' setting is not possible as a result of numerical problems. During milling, these lines are displayed as a 3D space grid so that you can also see the traverse paths in the tool axis.

2.12.3 3D simulation

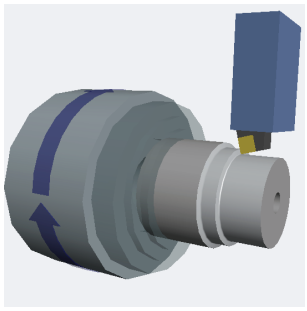


The 3D simulation with material removal allows a realistic preview of later machine production. The complete machining compartment including clamping means and neighbouring tools are also checked for collision prevention here.



2.12.3.1 View possibilities

The 3D simulation offers various perspectives to the machining process and different detail grades (= number of displayed components) to always ensure the best view possible. Due to the fact, that real situations are to be simulated, this also means that the default values for simulation must comply with the reality. The supplied standard machines have a fixed machining compartment, fixed maximum traverse paths and a fixed number of stations on the turret. If the devault values for simulation deviate from the standard machine, it is only possible to display a 3D simulation with workpiece/clamping means and tool.

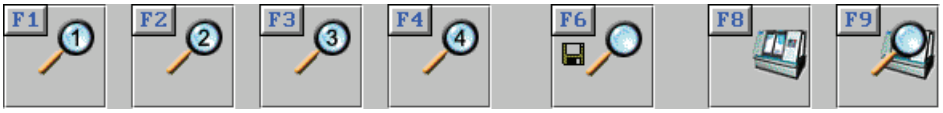


This, for example, is the case, if a tool was selected with more than 12 tool positions (in this example T14), because the 3D standard lathe has only a 12-position turret. In this case, the simulation automatically changes to this view. If traverse paths are exceeded, this will not be carried out automatically, because the software is not able to differentiate between an intentional and unintentional deviation. For this reason, we recommend to switch over this view, e.g. in the case of special machines with long traverse paths, using <F9> Simulation / <F3> 'Presettings' to the '... Tool/workpiece only' option.

2.12.3.2 Saved views

In the 3D simulation, various views to be freely defined can be invoked under <F6> 'Views'.

The views can be freely assigned by means of the <F6> 'Save' function. The currently set view and detail level is saved respectively.
The <F8> 'Machine' and <F9> 'Machining compartment' functions are factory-set according to their designation, but can also be used for your own settings, if necessary.



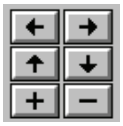
Tip:
The view saved under 'Machine' is always automatically invoked when the 3D simulation is started. Proceed as follows to see your favourite view directly after invoking the 3D simulation:
Set the desired view, invoke <F6> 'Save', switch over to 'Machine' in the selection field and apply by pressing <F10>.

2.12.3.3 Any views

You can invoke functions for setting the view using the <F11> key or clicking on the zoom symbol.

These functions can be used to rotate <F 1 >, position <F 2 > and change <F 3 > the size of the displayed workpiece in 3D.

If you feel that the mouse is not precise enough for this process, you can also set the current view by means of the displayed navigation keys, in accordance with the respective selected function.



In addition you can use the <F6> 'Screen print-out' function.

Tip:
The desired view can also quickly be set directly without invoking the 3D additional functions.
Double click on the object to be viewed in detail in the 3D simulation first. This will centre the view of the object.
Then proceed as follows:
Rotate by keeping the LH mouse key pressed and moving the mouse. Displace by keeping both (or the central) mouse key(s) pressed and moving the mouse.
Enlarge by keeping the RH mouse key pressed and moving the mouse or using the mouse wheel.

2.12.3.4 View in detail

The <F3> 'Details' function can be used to consecutively switch various detail grades from the overall machine over workpiece/tool to tools during turning. In this way, it is possible to map out components obstructing the sight in the desired view.

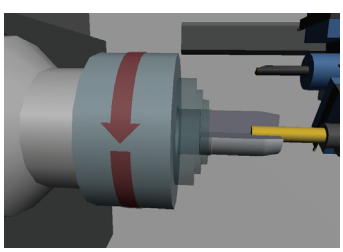
2.12.4 3D turning simulation

2.12.4.1 Internal view during turning

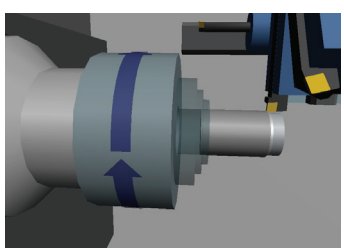
The <F5> 'Internal' key can be used to switch over to a semi thickness cutting of the workpiece at any time. This switch-over is carried out automatically for an internal machining (compare the images under "2.12.4.2 Display of the rotational direction").

NOTE:
Once a driven tool has been used, an internal view is not possible for the remaining simulation, even for internal machining.

2.12.4.2 Display of the rotational direction



The physical rotational speed of the spindle is indicated by means of an arrow on the chuck.
The arrow colours indicates whether M03 or M04 were programmed.



2.12.4.3 Machine types

For the 3D simulation in turning, an inclined bed machine with a main spindle is available in addition to the tool/workpiece simulation.

2.12.4.4 Workpiece clamping

In the ’Workplan’ and ’PAL simulator’ operating modes, the clamping depth of the workpiece is determined via the <F4> ’Clamping’ function in the respective setting dialog (<F1> ’File’ / <F2> ’Settings’). In the ’Simulator’ operating mode, the clamping depth is preset to 15mm during program creation. This default value can be changed in the ’Vorein.ini’ file with an ASCII editor. If necessary, change the value of the ’3DSim_clamping_depth’ entry. This is located below the ’[SteuSim]’ section. If NC programs are created in the ’Workplan’ operating mode, the respective clamping depth set there is used.

2.12.5 3D Milling simulation

2.12.5.1 Machine types

For 3D simulation in milling, machines in knee-type, cross slide table and travel stand design are available in addition to the tool/workpiece simulation. You can invoke the desired machine type via <F3> ’Presettings’ in the simulation dialog.

2.12.5.2 Workpiece clamping

’Workplan’ operating mode

In the 3D simulation, the workpiece zero point is aligned in relation to the machine table centre. The workpiece is positioned above the machine table according to the clamping situation set under <F1> ’File’ / <F2> ’Settings’ / <F4> ’Clamping’.

’PAL simulator’ and ’Simulator’ operating modes

In the 3D simulation, the workpiece zero point is aligned in relation to the machine table centre. The workpiece bottom edge is positioned 10 mm above the machine table. This default value can be changed in the `'vorein.ini'` file with an ASCII editor. If necessary, change the value of the `'AbstandWstTisch'` entry. This is located in the `'[sim3d]'` section.

3 Supplementary information on turning

This chapter describes supplementary information applicable to SYM*plus* used in Turning technology.

3.1 Operating mode: ’G1 G2 G3’

This subject was also dealt with in chapter ’General complementary information’ under the point ”Operating mode: ’G1 G2 G3” on page 23.

3.2 Operating mode: ’PAL simulator’

For this also refer to point ”Operating mode: ’PAL simulator” on page 24 in chapter ’General complementary information’.
The behaviour when changing the clamping position corresponds to that of the control simulators. The name of the presetting file of the PAL simulator is `0kqdwx.ini` (see 3.3.1).

3.3 ’Simulator’ operating mode

For this also refer to point ”’Simulator’ operating mode” on page 24 in chapter ’General complementary information’.

3.3.1 Changing the clamping position of workpieces

For a more descriptive education, the ’Simulator’ operating mode offers the possibility to change the clamping position of workpieces within the 2D and 3D simulation via a special M function. These function only serves for this internal purpose in the simulator. It should have no function on the control system and thus lead to an error message. For safety reasons, we recommend to delete this M function prior to starting the program on the machine.

This freely determinable, free M function (e.g. `M999)` can be used to change the clamping position of workpieces in the simulation. The position set for the first workpiece edge is taken. These clamping settings are carried out for the ’Simulator’ operating mode in the ’System configuration’ (also see Section 2.10.4.1 on page 39).

Here are some example block containing a zero point offset in Z to allow facing, e.g. after changing the clamping position:

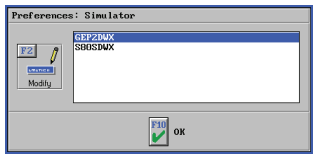
```
...
N320 G0 X42.5
N321 G0 X150 Z150 M9
( CLAMPING      )
N322 M999
N323 G59 Z-1
N324 M00
(DRILLING)
N325 G97 F0.1 S2000 T0808 M3
...
```

Determination of M function for changing the clamping position

’Simulator’ operating mode
The simulators are factory-set to a corresponding free M function (`M99`,`M900` or `M999`) suitable for the respective control system. If this setting must be changed, use a text editor to change the respective presetting file of the corresponding control simulator.

Proceed as follows:

1. Open the dialog containing the presettings of the control simulators in the 'Set-up' operating mode using <F4> 'System configuration', <F1> 'Presettings' and <F2> 'Change'. Write down the name of the respective presetting file, in this example `GEP2DWX` for the GILDEMEISTER EPL2 simulator (`S80SDWX` corresponds to the SINUMERIK 802 C simulator supplied as a standard).



2. Open the `gep2dwx.ini` file with an ASCII editor (e.g. with the `edit gep2dwx.ini` command in the input request). The file is in the user data directory (also see Section 3.2 on page 4 in the installation manual).

3. Now several key words appear in square brackets, the so-called sections. Below the `[INTERPRETER]` section you will find the `CHUCKING_NR=` entry. You can change the value positioned behind this respectively. Restart the software to ensure that the change is applied.

'Workplan' operating mode
Set the respective postprocessor so that NC programs created in the 'Workplan' operating mode are automatically provided with a corresponding M function.

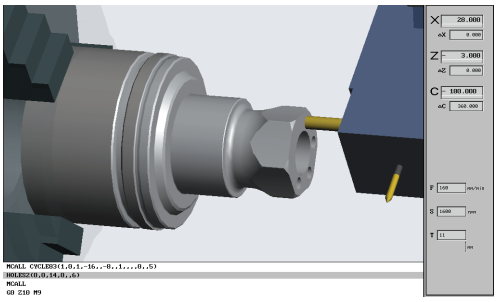
Proceed as follows:

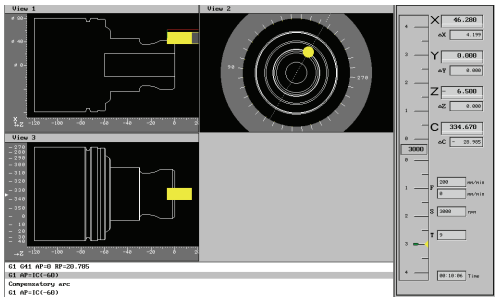
1. Open the dialog containing the pre-settings of the postprocessors in the 'Workplan' operating mode using <F1> 'File', <F8> 'Postprocessor' and <F2> 'Change'.

2. Select the corresponding postprocessor parameter block via <F1> and then invoke the <F4> 'Coolants, ...' option. Here you can enter the desired M function (e.g. `M888`) for changing the clamping position.

3.3.2 Simulation of C and Y axis (option)

When you have purchased the respective simulator, it is possible to simulate the programmed C and Y axis commands in a 3D or 2D simulation in a special 3-window view (figure on the right).





Independent of each other various zooms can be set in all three windows.
For this, invoke the zoom function as described under "Additional functions" on page 15.
The red zoom frame can be positioned in the various views using the mouse.

Tip:
In the C and Y axis 2D simulation, it is also possible to move the red zoom frame to the next view by means of the <TAB> key.

3.4 ‘Geometry’ operating mode

This subject was also dealt with in chapter ’General complementary information’ under the point "‘Geometry’ operating mode" on page 29.

3.5 ’Workplan’ operating mode

Here you obtain important basic information and tips of how to create and change a work plan.

3.5.1 ’File’ menu

The finished part designed in the ’Geometry’ operating mode has to be set first under <F1> ’File’ / <F1> ’New’ and then the unmachined part entered (if the start assistant is not used).

NOTE:
When you change the geometry of a finished or unmachined part already selected in the work plan in the ’Geometry’ operating mode at a later point in time, the work plan does not ’realise’ this automatically. To do so, you have to import the changed geometry manually into the work plan via the <F1> ’File’ / <F2> ’Settings’ function. Subsequently you have to have the work plan computed anew (see below) and check the work steps, if necessary, for possible changes.

The entered material and the entered turret result from the pre-setting in the ’Setting’ operating mode. It is, of course, possible to proceed with any desired setting.
Clamping is subdivided into five areas:

Workpiece position

Turn workpiece	This option indicates whether the workpiece designed in the ’Geometry’ operating mode as an ’Overall contour’, is to be machined on the right-hand or left-hand side.
Spacing from chuck	Here you enter the spacing between the contact surface of workpiece and the chuck face side (for instance for drilling through).

Zero point position

Reference	This option indicates the position of the workpiece zero point (face side or contact side).
Offset Z	Enter here where the zero point of the finished part is to be in relation to the unmachined component. Note the prefixes, depending on where you have set the zero point reference.

Finished part position

Reference point Z	This option indicates the position of the reference point of the finished part is to be positioned in relation to the zero point position.

Protection zones

Reference	Here you decide the point from which you wish to describe the protection zones (they are used later for the traverse path computation. Normally you will describe the protection zones with the ’Chuck’ option (here the chuck face side is meant).

External value Z — Suppose you have set the protection zone reference to 'Chuck' with the clamping jaws having a length of 30 mm, and you wish to approach the clamping jaws up to a maximum of 5 mm, then enter the value 35 mm.

Diameter X — With internal machining extending in the protection zone area, you set the corresponding internal diameter here. Without internal machining, you can also enter a value of 0 mm.

Internal value Z — When you have entered a value of > 0 in the 'Diameter X' field, you can set the corresponding protection area here.

Adjustments/Settings

Change point X — Here enter the desired tool change point for the X axis.

Change point Z — Here enter the desired tool change point for the Z axis.

NOTE:
The tool change point is moved to with the measured tool point. Knowing that at the beginning of a work plan a tool is not yet changed in, the tool change point is moved on the basis of the tool holder reference point.

Chuck thickness — For checking purposes, the chuck thickness entered in the pre-settings of the 'Setting' operating mode is displayed. This value cannot be modified here. If you require a different chuck thickness for a work plan, you have to set this first of all in the 'Setting' operating mode under <F4> 'System' / <F2> 'Presettings' / <F3> 'Clamping' and create a new work plan later.

Maximum speed — For checking purposes, the rotational speed limitation entered in the pre-settings of the 'Setting' operating mode is displayed. This value cannot be modified here. If you require a different speed limitation for a work plan, you have to set this first of all in the 'Setting' operating mode under <F4> 'System' / <F2> 'Presettings' / <F1> 'Machine' and create a new work plan later.

The work step editor described in the following text is usually invoked via the <F2> 'Edit' menu point. Only when creating a new workplan does the system directly change to the work step editor for efficiency reasons when applying the setting dialog using <F10>. If, however, you return to the 'Workplan' menu later, the work step editor is no longer automatically activated when changing values in the setting dialog of the workplan and applying the window.

3.5.2 'Edit' menu

In the following, please find all important information on how to perform the work steps.

3.5.2.1 Work step editor

The work step editor contains functions allowing the work step creation, editing and management of a work plan.

Functions and their meaning:

<F1> 'Create' This function is used to create new work steps.

<F2> 'Change' This function is used to invoke the work step which the blue cursor is on for changing.

<F3> 'Delete' This function is used to delete work steps. Either just the work step which the blue cursor is on is deleted, or all selected work steps (see 'Select').

<F4> 'Return' When the blue cursor is located in the top 'Work steps' window, these steps can be shifted into the bottom 'Filing' window. There you can, for instance, keep work steps ready to be used at a later point in time. Now either just the work step which the blue cursor is on or all selected work steps can be shifted (see 'Select').

or

<F4> 'Execute' When the blue cursor is in the bottom 'Filing' window, (**the <F4> symbol changes correspondingly**), these work steps can be shifted to the top 'Work step' window. When doing so, they are always appended to the end of the work plan. Now either just the work step which the blue cursor is on or all selected work steps can be shifted (see 'Select').

<F5> 'Select' This function can be used to select several work steps for a following action (see above).

<F6> 'Compute' If one or more work steps are marked with an asterisk resulting from changes to tools or technological parameters etc., you have to have the work plan re-computed with this function. This is not performed automatically to give you the opportunity to perform further changes, for instance on other work steps. You can also deactivate the corresponding work steps for computation and simulation and have them performed again.

ATTENTION:
Following this, check the work plan (for instance by means of simulation), because existing traverse paths can change depending on the situation!

<F7> 'Information' Activate this view if you want to have the traverse paths and red chip removal surfaces displayed when moving the blue cursor over via the work steps. In this manner you obtain a very good view of the production (machining) sections of the workpiece
(also refer to <F8> '3D view' and <F9> 'Simulation').

NOTE:
Please note that the work steps always show the traverse paths and chip removal surfaces at the time of their creation.
When you have a look at the preview of a work step under '**Filing**' it must not necessarily match any longer the work plan/workpiece modified in the meantime. The traverse paths and chip removal surfaces will be computed anew the next time you perform such a work step!

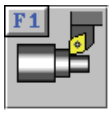
<F8> '3D view' If the 'Information' function is active, the manufacturing status of the workpiece is displayed up to the selected work step. When the Information display is deactivated, the total status of the workpiece is always displayed.

In addition to the mass and volume display, you can also view and cut all sides of the workpiece by changing its position.

<F9> 'Simulation' If the <F7> 'Information' function is active, only the **selected work step** is simulated. When the Information display is deactivated, the **entire workplan** is always simulated.

3.5.2.2 Work steps

Rough cutting

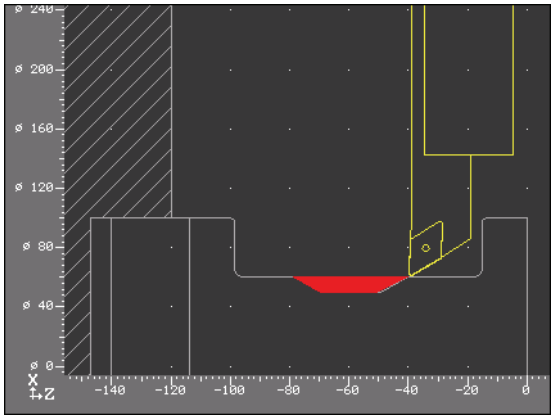


Once the work step is invoked, the software selects the first tool - starting with station 1 of the current turret -, for which 'Roughing' was set to 'YES' in the 'Setting' operating mode under Operating Modes.

The selection of the tool type is decisive for the chip removal strategy used in the work step. If you use a "standard" 80° rotary cutting tool, you can select between various chip removal directions for facing, longitudinal turning and turning with parallel contours.
If, however, you employ a customised parting-chisel for this purpose supplied by a manufacturer, in this work step you can automatically use high-performance parting-tool strategies for all contours which can be produced with the corresponding cutting edge geometry.

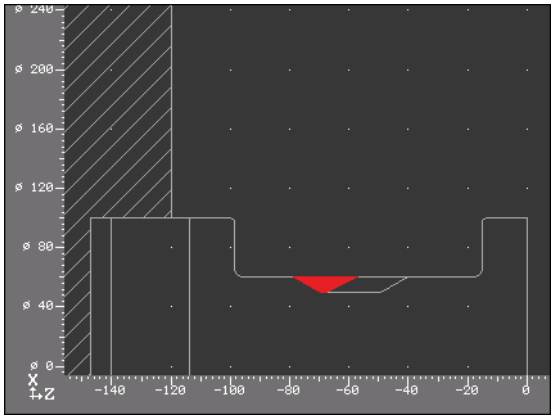
Once the tool was selected, the possible chip removal area is displayed in red. Take note of the following particularity:

With SYMplus the internal mathematical consideration of the cutting edges of turning tools does not terminate at the end of the cutting tips. On the contrary, mathematically the tool cutting edges undergo "unlimited" extension to assist the automatic computation of cut-in angles. This is used for the possible "chip removal surfaces" - the latter always being displayed in red in SYM*plus*.
However, this "intelligence" requires your attention to a special feature which is explained by way of the following example:



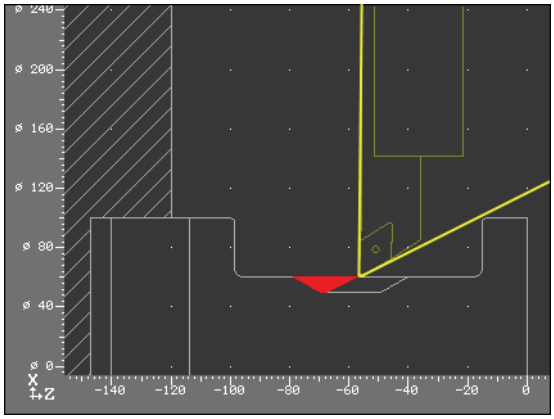
SET

This residual material (red surface) is to be roughened with the 55° turning tool shown here. Although this tool could move to the shown start position without problem, the red surface is displayed much smaller when the work (machining) step is created.

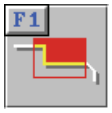


ACT

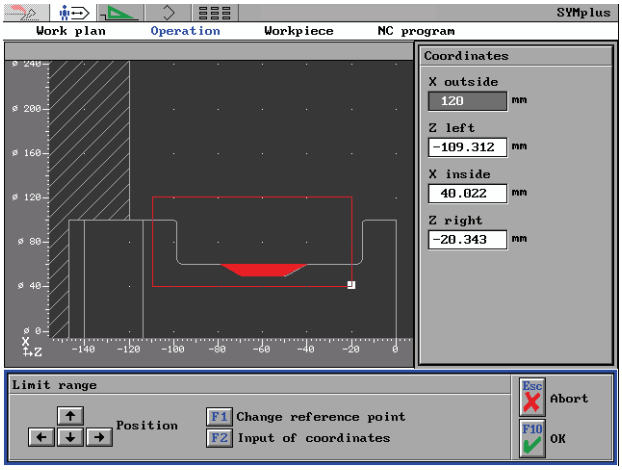
This is caused by the "unlimited" tool cutting edges. SYM*plus* "regards" the tool cutting edge as a large "wedge" (as shown in the following picture). To avoid reject and collision, this "wedge" must not intersect with the workpiece contours!



In general, this automatic computation is of great help. But in these and similar cases, you can 'tell' the SYM*plus* the workpiece areas that should remain unconsidered in the corresponding work step.



For this purpose, use the 'Limitation' function. You can call up this function with the symbol at the side in the work steps for roughing, finishing and cutting-in machining.



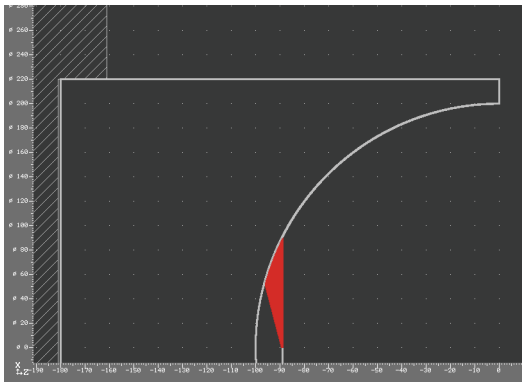
In this case, ensure first of all with <F1> 'Change reference point' that the reference point (white square) changes to the bottom right corner of the limitation frame. Then you can reduce the frame using the <arrow keys> such that the right-hand shoulder is outside of the frame. Now this section of the workpiece will be ignored and the red surface and the tool paths can be computed without problems.

NOTE:
Simply imagine that this limitation frame confines the 'field of vision' of the SYM*plus*. You determine what SYM*plus* should or is allowed to "see". However, consider that this limitation of the 'field of vision' cannot make the actual material of the workpiece disappear. If, for instance, the tool is so wide that it would hit the workpiece edge with its holder, the software signals a collision, and the work step must be changed accordingly.

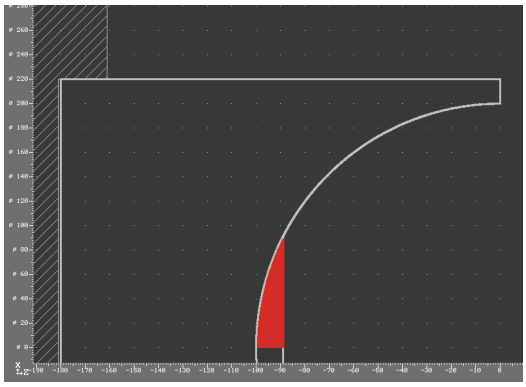
NOTE:
You are certainly aware of the problems involved in producing a long workpiece which is not optimally clamped and with high cutting values, in a cost-efficient manner. Here SYMplus can also be an aid.
You can divide the overall volume into several work steps using the <F1> 'Limitation' function. Due to the fact that you use own technology data for each work step, they can always be adapted to the selected area. In this manner, you can use smaller approach and advance values further away from the chuck than directly in front of the chuck.

Roughing up to the centre line

Machining up to the centre line with X0 is a special case of the "unlimited" tool cutting edges. An example for this with a spherical internal contour:



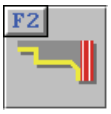
For such a geometry, the software "usually" offers a respectively restricted roughing section on the ball "bottom".



When setting the 'Section calculation' option to 'Extended (up to X0)' in the second dialog of the 'Roughing' work step, it is possible to machine the workpiece up to the centre line.

Why is this option not preset in general?
Due to the fact that unexpected results may occur depending on the respective workpiece geometry and because the safety can only be ensured by a "strict" setting to "Normal".

Facing with the 'Roughing' work step

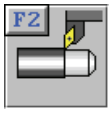


Once the <F1> 'Roughing' work step has been selected, a further dialog appears offering the various machining possibilities. The unmachined part section on the RH side of the finished part is automatically selected when invoking the <F2> 'Facing' work step.

If the existing unmachined part material is larger on the facing surface than the proposed section or should the facing work step also consider other sections, you can set the section to be machined respectively via the <F1> 'Section' function.

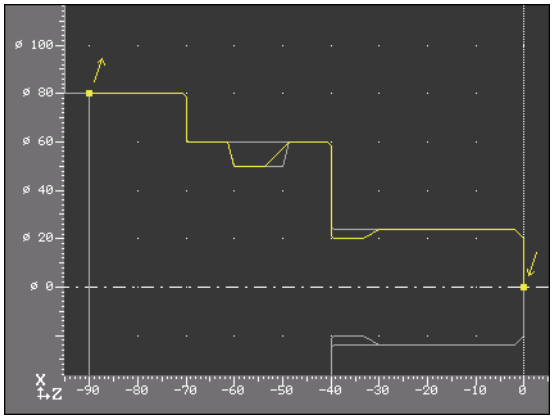
The <F2> 'Approach/Retract' function is used to determine the approach and retract lengths as well as the retract angle for the 'Facing' work step.

Finishing



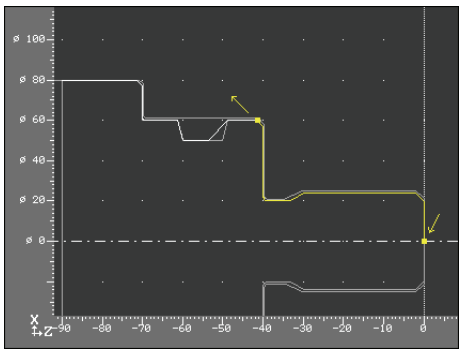
If the work step 'finishing' and a corresponding tool are invoked, the system always offers the full finishing path possible.
Depending on the setting and cutting tip angle, the tool either does not move at all (0° cut-in angle) or moves into the material with the set cut-in angle.
The following examples are intended to show you the different possibilities offered by the 'Finishing' work step.

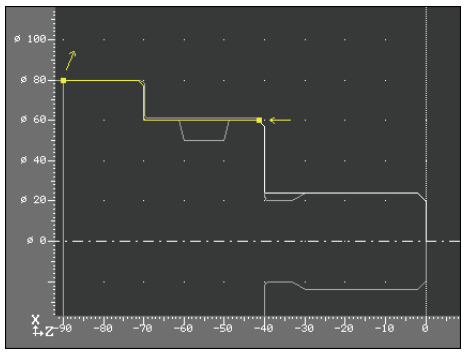
Example 1:
A workpiece with a cut-in is to be finished with a 35° turning tool **prior to** cutting in.
The system automatically proposes the entire finishing path possible, however, this does not rule out the recess. From the technological point of view, finishing the recess with this tool, however, does not make sense.
In this case, a cut-in angle of 0° is of no help knowing that then also the undercut would not be finished.



Finishing, then cutting-in

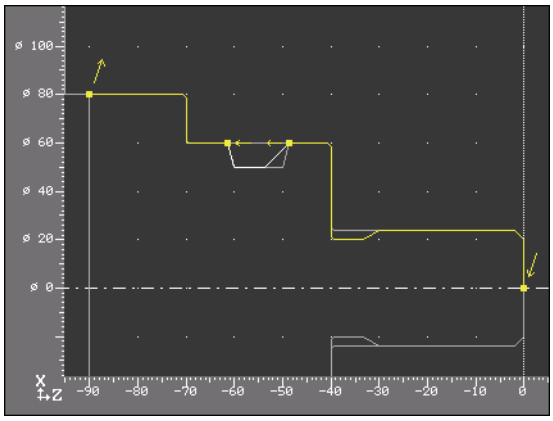
Solution:
Create **two** 'finishing' work steps'.
For the first you use a cut-in angle suitable for the undercut, then you finish the contour, for instance up to the end of the bevel for a 60 mm diameter.
In the second work step, you set the cut-in angle to 0° and finish to obtain a 80 mm diameter from the bevel end, starting tangentially, up to the end of the workpiece.





Example 2:
A workpiece with a recess is to be finished **after** cutting in using a 35° turning tool without the tool moving into the recess or into the reducing contour.

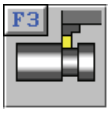
Solution:
Within the 'finishing' work step, you create two finishing paths for this contour. In this case, the first finishing path stops with a tangential retract angle (<F1> 'Tangential' in the 'Change retract path' dialog) at the beginning of the recess, and the second finishing path starts, also tangentially, at the end of the recess.



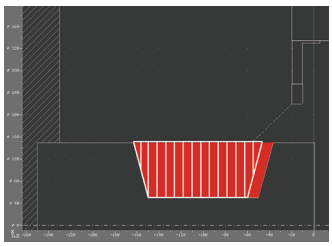
Cutting-in, then finishing

Now SYM*plus* ignores the recess and moves over it. You can, of course, create any number of these individual traverse paths, in order to "bridge" all reducing contours in this manner.
The path between the end of the retract path and the beginning of the approach path is travelled at high traverse speed to save time, especially with correspondingly large recesses or reducing contours. When this path is to be traversed at feed speed, the end of the retract path must be exactly on the start point of the next approach path.

Cutting-in

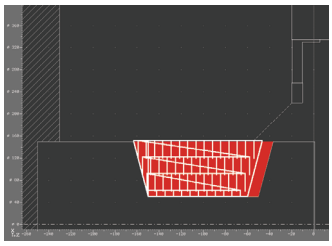


Whereas generally the corresponding cut-ins are designed in the 'Geometry' operating mode, you can also do without designing cut-ins in the 'Geometry' operating mode and insert these cut-ins here at a later point in time using the <F2> 'Cut-in' and <F4> 'Random' functions. You have to bear in mind, however, that in this case no outer bevels or radii can be created. In general, as already detailed, the cut-ins are designed in the 'Geometry' operating mode.



The usual cut-in process can result in unfavourable machining conditions when large cut-in depths are to be machined.

Cut-in to full depth



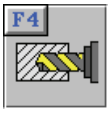
For this reason, it is possible to select the 'Depth proportion' option in the second dialog of 'Cut-in' work step. A cut proportion is defined here by entering a maximum cutting depth.

Stepped cutting-in

NOTE:
With the cut-in work step the cut-in width must always exceed the width of the parting-tool. If, for instance, you wish to produce greasing grooves having the same width as the cutting tip, you have to use the supplied 'CUT-IN' machining macro.

'Parting off'
The <F3> 'Parting off' function is a special feature. Generally, the parting off area need not be designed in the 'Geometry' operating mode. You should always directly use this function for this purpose. You can use the 'Z value' to define the location of parting off; the software then automatically defines the diameter to start with. Finally you can enter whether or not a bevel or rounding is to be processed on the workpiece side. The possibility to enter a transition diameter to define the limit from which the system is to change over to a constant rotational speed, is very helpful. In this manner you prevent the workpiece from turning at an excessive speed, and, as a consequence, from being ejected uncontrolled when parted off.

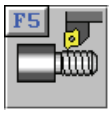
Drilling



You can only select drilling strategies which you have applied for the corresponding tool in the 'Setting' operating mode under the 'Machining' option with YES.
For chip breaking and deep hole drilling you can define, whether or not the tool is to cut in to the hole or retract at a certain advance speed or at rapid traverse.

NOTE:
For threading, please use the 'Thread' work step.

Thread



After invoking the <F5> 'Thread' function, you can select whether you wish to cut a thread using <F1> 'Turning' or with <F2> 'Drilling'.
Thread cutting
Depending on the cutting edge geometry, you can produce all common thread forms in this operating step. The type of tool automatically defines whether an internal or external thread is to be cut. You decide if a right-hand or left-hand thread is to be produced. Even multiple threads can be produced.

We recommend that you always select the <F1> 'Thread path' function for the contour section on which the thread is to be cut. If you wish to produce a taper thread you have to change the permitted machining type from 'longitudinal' to 'taper' for the corresponding tool in the 'Setting' operating mode. This is the pre-requisite for the selection of slanted contour sections, too, provided that these can the produced at all with the tool concerned.

Following this, you can influence the moving-in and moving-out length as well as the approach and retract lengths and angles. The use of the zoom function with <F11> 'Additional function' / <F1> 'Zoom' can be very helpful for the assessment of the correct moving-in and -out length, especially when threading against a workpiece shoulder (for example with undercut).

Finally, in addition to the chip removal technologies, you only have to set whether and how you want to realise a 'degression' of the approach depth and decide about 'Approach direction' to be used for approaching between flanks.

'Degression'

'At the start point':
The cuts are distributed from the beginning such that a constant chip cross section is produced from the first to the last cut.

'At the end point':
Linear cut proportion until the cutting depth before the last, the last cutting depth is divided into four cuts in the ratio 1/2, 1/4, 1/8, 1/8.

'None':
Linear cut proportion concerning the approach depth

'Approach direction'

'Centre' 'LH/RH' 'LH' 'RH'

NOTE:
We have developed powerful machining macros for special productions of very large saw blades or round threads (for instance for sheaves). Knowing that these large thread geometries cannot be produced by the shape of a cutting tip, we have included special machining strategies into these machining macros.

They permit to roughen and finish these threads using a standard turning tool. Please contact us, we would be pleased to advise you.

Tapping
The cored hole need not be designed in the 'Geometry' operating mode, but can also produced directly in the work plan via the 'Drilling' work step. Please note that the software monitors whether the hole is larger than or equal to the cored hole diameter of the screw tap, this being the pre-requisite for tapping.

Handwheel/Teach-In

Even the "most intelligent" software cannot know all your production requirements. This means that if you have to traverse a very special path with a specific tool, it can be simply entered with this work step.

Each traverse path is displayed for you in the form of a 'hull' around the surface which is 'moved over' by the tool following this path. In this manner, you can check quite easily if the traverse paths have been correctly arranged. Later modification of these traverse paths is, of course, possible.

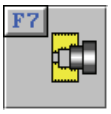
Traverse paths travelled at feed speed are displayed green, those travelled at rapid traverse with a red frame.

In the 'HANDWHEEL' work step it is possible to graphically change the tool position interactively in some input fields under 'Position' (marked with) and under 'Manual':

- For this switch on the "keyboard handwheel" first. <Shift>+<F4>
- Handwheel increment increase <Shift>+<F5>
  Handwheel increment reduction <Shift>+<F6>
  Here, observe the pointer position
  underneath the actual value display: 

- Use the handwheel to move in the positive direction <Alt>+<Scroll down>
  Use the handwheel to move in the negative direction <Alt>+<Scroll up>

Clamping



This work step can be used at any time if you wish to change the clamping situation (e.g. changing the clamping position), the protection zone or the tool change point.
When the workpiece can be produced in one clamping position and without changing the protection zones or the tool change point, you do not need this work step. All required clamping data has already been entered when creating the work plan.

Macro

You know better than anybody else how different the individual workpieces and machining possibilities are.
SYM*plus* includes the corresponding work steps for all standard machining processes when turning.
For an extended functionality we also deliver machining macros, such as 'CUT-IN'. This macro is used when a cut-in has to be produced with a cut-in width that equals the tool width.
Even if you have to produce a special part or look for a lower cost way of production, this will be no problem. In most cases we are able to create special work steps in the form of customised machining macros.

3.5.3 'NC program' menu

This subject was also dealt with in chapter 'General complementary information' under the point "'NC output' menu" on page 36.

3.6 'Teacher' operating mode

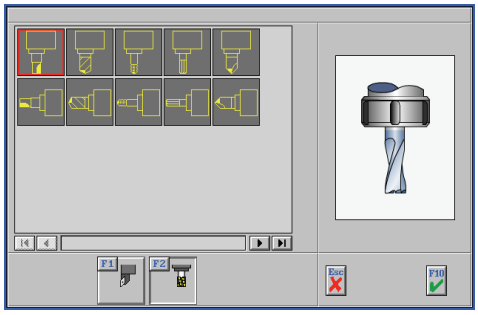
This subject was also dealt with in chapter 'General complementary information' under the point "'Teacher' operating mode" on page 36.

3.7 'Set-up' operating mode

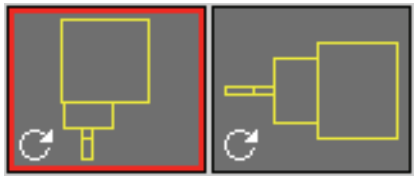
For this also refer to the further information in chapter 'General complementary information' under point "'Set-up' operating mode" on page 39.

3.7.1 Driven tools for C and Y axis (option)

When you have purchased the respective simulator, it is possible to also create driven tools in the 'Tools' menu in addition to the turning tools.



After the installation of a respective simulator, two new buttons appear with 'Create new' tools. It is possible to toggle between turning tools and driven tools using <F1> 'Turning' and <F2> 'Milling'.



To ensure that driven tools in a turret can be easily distinguished from the cutting-in tools, they are marked with a rotation symbol.

3.8 'Transfer' operating mode

This subject was also dealt with in chapter 'General complementary information' under the point "'Transfer' operating mode" on page 45.

4 Supplementary information on milling

This chapter describes supplementary information applicable to SYM*plus* used in Milling technology.

4.1 Operating mode: ’G1 G2 G3’

This subject was also dealt with in chapter ’General complementary information’ under the point ”Operating mode: ’G1 G2 G3” on page 23.

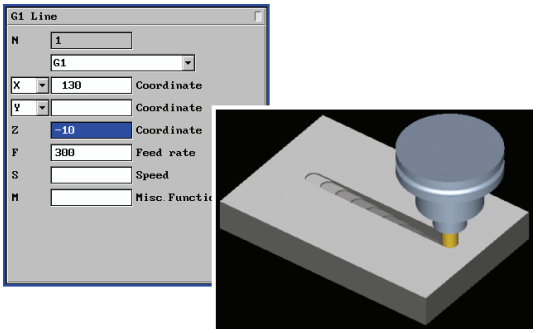
4.2 Operating mode: ’PAL simulator’

This subject was also dealt with in chapter ’General complementary information’ under the point ”Operating mode: ’PAL simulator” on page 24.

4.3 ’Simulator’ operating mode

For this also refer to point ”’Simulator’ operating mode” on page 24 in chapter ’General complementary information’.

4.3.1 3D traverse movements in milling simulators



It can traverse in three axes. In the simulation graphics this is achieved by a "stepped movement".

NOTE:
Please observe, however, that the simulated internal workpiece model becomes more complex with each step, causing the simulation to slow down. Therefore, only make use of this 3D option in very special cases.

4.4 ‘Geometry’ operating mode

This subject was also dealt with in chapter ’General complementary information’ under the point ”‘Geometry’ operating mode” on page 29.

4.5 ’Workplan’ operating mode

Here you obtain important basic information and tips of how to create and change a work plan.

4.5.1 ’File’ menu

The finished part designed in the ’Geometry’ operating mode has to be set first under <F1> ’File’ / <F1> ’New’ and then the unmachined part entered (if the start assistant is not used).

NOTE:
When you change the geometry of a finished or unmachined part already selected in the work plan in the ’Geometry’ operating mode at a later point in time, the work plan does not ’realise’ this automatically. To do so, you have to import the changed geometry manually into the work plan via the <F1> ’File’ / <F2> ’Settings’ function. Subsequently you have to have the work plan computed anew (see below) and check the work steps, if necessary, for possible changes.

The entered material and the entered magazine result from the pre-setting in the ’Setting’ operating mode. It is, of course, possible to proceed with any desired setting.

If desired, clamping means can be used. The clamping means directly influence the traverse path computation in the sense of "protection zones" and are also taken into account for the anti-collision check. The clamping means have to be created individually and cannot be copied. Should you wish to use a clamping means several times, you have to design it in the ’Geometry’ operating mode (same as complex clamping situations) and save it as a random clamping means.

In the ’Clamping’ menu, also the important settings for the tool changing tool are entered. When doing so, please bear in mind that the tool change point is chosen such that the longest tool can be changed in into the magazine.

In addition to the tool change point, the retract level is of significance, for instance if you use clamping means located above the workpiece surface (e.g. clamping claws). Here you have to enter the retract level on which rapid traverse is reliably possible during positioning.

The work step editor described in the following text is usually invoked via the <F2> ’Edit’ menu point. Only when creating a new workplan does the system directly change to the work step editor for efficiency reasons when applying the setting dialog using <F10>. If, however, you return to the ’Workplan’ menu later, the work step editor is no longer automatically activated when changing values in the setting dialog of the workplan and applying the window.

4.5.2 ’Edit’ menu

In the following, please find all important information on how to perform the work steps.

4.5.2.1 Work step editor

The work step editor contains functions allowing the creation and management of a work plan.

Functions and their meaning:

<F1> ’Create’	This function is used to create new work steps.
<F2> ’Change’	This function is used to invoke the work step which the blue cursor is on for changing.
<F3> ’Delete’	This function is used to delete work steps. Either just the work step which the blue cursor is on is deleted, or all selected work steps (see ’Select’).

<F4> 'Return'

When the blue cursor is located in the top 'Work steps' window, these steps can be shifted into the bottom 'Filing' window. There you can, for instance, keep work steps ready to be used at a later point in time. Now either just the work step which the blue cursor is on or all selected work steps can be shifted (see 'Select').

or

<F4> 'Execute'

When the blue cursor is in the bottom 'Filing' window, (the <F4> symbol changes correspondingly), these work steps can be shifted to the top 'Work step' window. When doing so, they are always appended to the end of the work plan. Now either just the work step which the blue cursor is on or all selected work steps can be shifted (see 'Select').

<F5> 'Select'

This function can be used to select several work steps for a following action (see above).

<F6> 'Compute'

If one or more work steps are marked with an asterisk resulting from changes to tools or technological parameters etc., you have to have the work plan re-computed with this function. This is not performed automatically to give you the opportunity to perform further changes, for instance on other work steps. You can also deactivate the corresponding work steps for computation and simulation and have them performed again.

ATTENTION:
Following this, check the work plan (for instance by means of simulation), because existing traverse paths can change depending on the situation!

<F7> 'Information'

Activate this view if you want to have the traverse paths and red chip removal surfaces displayed when moving the blue cursor over via the work steps. In this manner you obtain a very good view of the production (machining) sections of the workpiece
(also refer to <F8> '3D view' and <F9> 'Simulation').

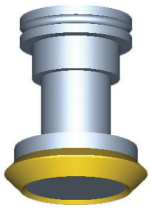
NOTE:
Please note that the work steps always show the traverse paths and chip removal surfaces at the time of their creation.
When you have a look at the preview of a work step under '**Filing**' it must not necessarily match any longer the work plan/workpiece modified in the meantime. The traverse paths and chip removal surfaces will be computed anew the next time you perform such a work step!

<F8> '3D view'

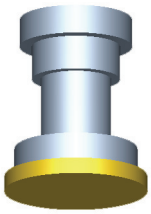
If the 'Information' function is active, the manufacturing status of the workpiece is displayed up to the selected work step. When the Information display is deactivated, the total status of the workpiece is always displayed.

In addition to the mass and volume display, you can also view and cut all sides of the workpiece by changing its position. If you need an asymmetric section through the workpiece, perform the <F2> 'Cuts' several times, because every time only a rectangular section window is offered.

An important point is the <F4> 'Tool' function :



If you poll the <F4> 'Precision view' function here, the tool is displayed the way you defined it in the 'Setting' operating mode.



When you invoke the <F3> 'Tool' function, the tool is displayed in the way the software sees it internally for the computation of the travelling path and the workpiece model. These are only cylindrical bodies, in other words, in the 3D view and on the internal workpiece model, rounded or slanted edges on the tool only create rectangular workpiece edges.

<F9> 'Simulation' If the <F7> 'Information' function is active, only the **selected work step** is simulated. When the Information display is deactivated, the **entire workplan** is always simulated.

4.5.2.2 Work steps

Surface

Having invoked the work step, the software selects the first workpiece - starting with station 1 of the current magazine - for which 'Surface milling' was set to 'YES' in the 'Setting' operating mode under machining types.

It is always the highest surface to be rough-cut that is displayed red. To obtain a review of the possible application strategies of this tool, it is practical to find out at this point with the <+> key whether this tool is suitable or if you should select a different tool from the magazine. Please bear in mind that when selecting the internal milling you select a tool for which in the tool properties 'Cut-in' is 'YES' in the 'Setting' operating mode (otherwise an error message is generated).

'Multiple' choice
Once the first dialog window is applied, the geometry data belonging to the red surface are displayed. If you wish, you can add further machining surfaces for this work step with <F1> 'Multiple'. Here it is the order of addition which determines the later processing order in the sense of a milling strategy. By doing so, you are assisted by the <F4> function 'Add all of a kind' when selecting machining surfaces having the same start and end depth, corresponding to the surface with the red frame.

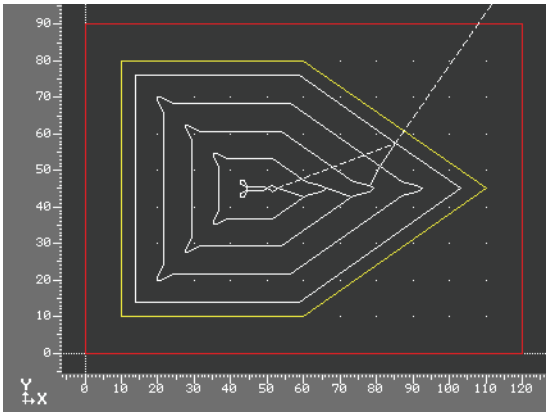
'Type of chip removal'
In the third dialog window you can select the type of chip removal, 'contour-parallel' usually being the optimum (in terms of time and technology).

In some cases also the 'Hatching' setting could be useful whereas the 'Plunge milling' only leads to good results in very special cases.

Traverse path 'Setting'
During internal milling, typically the traverse path setting 'Internal -> External' is correct whereas during external milling the 'External -> Internal' setting usually is the better choice. Reason: You define the point from which the tool is to start chip removal.

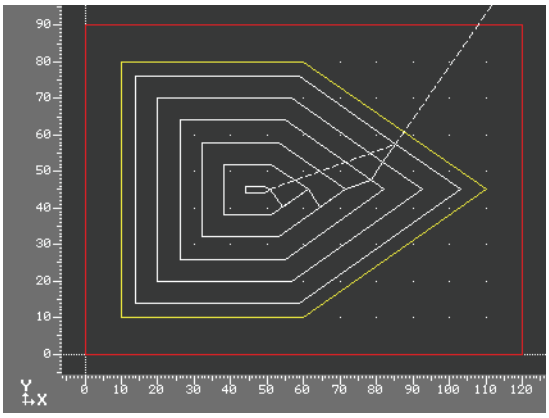
'Contour spacing'
The 'Contour spacing' button gives you the possibility to notably reduce the production times depending on the workpiece and tool.

If the 'Contour spacing' is set to 'Constant', the selected 'Cut-in width' is kept unchanged from contour to contour. To ensure that no residual material will remain for cut-in widths of more than 50 %, special compensation movements are created in the corners. This strategy might require a smaller number of milling contours and also less time compared to the adapted contour spacing:



Milling cutter: Ø 8mm
Cut-in width: 95%
Contour spacing: **Constant**
Milling contours: 5
Time: **8:02 min**

If the 'Contour spacing' is set to 'Adapted', the selected 'cut-in width' is adapted from contour to contour such that with cut-in widths of more than 50 % no residual material will remain. This means that not all contours have the set cut-in width.



Milling cutter: Ø 8mm
Cut-in width: 95%
Contour spacing: **Adapted**
Milling contours: 6
Time: **8:59 min**

Chip removal 'direction'
With the 'Direction' switch you can define if the tool is to run only in 'Synchronous, counter-run' or in 'Synchronous and counter-run'. The system automatically attempts to use this default direction for all traverse paths of the work step. However, this is not always possible and depends on the selected type of chip removal and the individual workpiece geometry.

Start and end points
In the fourth dialog window, please remember that you can influence the start and end point automatically determined in Z by the software for the selected machining surfaces.

An example for this:
You wish to produce an opening and have the tool move down by 2 mm deeper than the contour was designed (to compensate for wear or cutter radii). Simply set the 'End point in Z' field to 'Manual' and enter the desired 'Final depth'.

Cutting-in strategies
The fifth dialog window is used to determine whether cutting into the workpiece is to be carried out vertical, helical or along a ramp:

Cutting-in strategy 'Vertical' in the 'Surface' work step

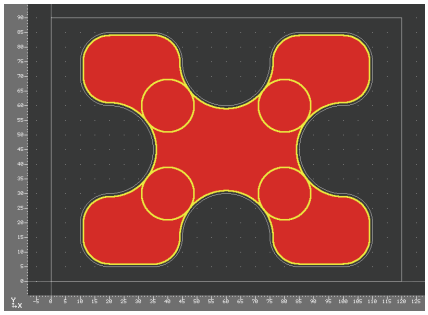
Cutting-in strategy 'Vertical'
The tool always cuts in vertically up to the respective approach depth. For complete cutting in only tools having the corresponding cutting geometry can be used, otherwise collision messages will appear.

Cutting-in strategy 'Helix' in the 'Surface' work step

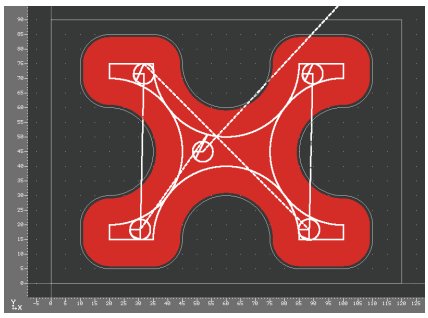
Cutting-in strategy 'Helix'
The tool always cuts into the material helically up to the respective approach depth. If an approach point is outside the material, cutting in is automatically directly carried out vertically. It is only possible to use tools for which the corresponding features have been set in the 'Set-up' operating mode.

NOTE:
If selecting a tool for the work step which should preferably cut-in helically, it is possible, depending on the set allowance and/or the helix diameter, that none or only individual possible (red) surfaces to be machined are displayed.

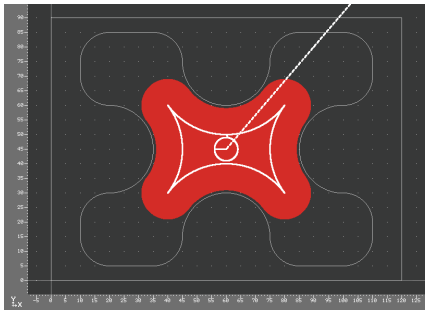
An example for this:



Due to the existing contour allowance of 1mm and taking the tool diameter of 18mm into account, 5 individual surfaces are already generated during calculation of the possible surfaces to be machined.



With a set helix diameter (= forming hole diameter) of 24mm cut-in points can be calculated in all 5 surfaces because the helix diameter fits into these surfaces.



With a set helix diameter of 26mm it is only possible to calculate a cut-in point for the central surface because the helix diameter no longer fits into the other surfaces.

Solution:
In this case, reduce either the allowance, e.g. to 0.5mm or the helix diameter to 25mm.

'Cutting-in angle' and 'Pitch'
The cut-in angle is a maximum admissible value corresponding to the respective tool. For this, also refer to the corresponding specifications of the tool manufacturer.
The pitch is automatically calculated from the respective helix diameter and the max. cut-in angle, but it can also be directly preset.

If the approach dimension for the work step is selected smaller than the pitch, the tool path on the helix is automatically reduced for the NC output.

'Reference point'
The reference point complies with the geometrical helix centre. The real start point is always automatically on the centre line of the tool.
If only one surface is selected for machining, the reference point can also be determined manually.

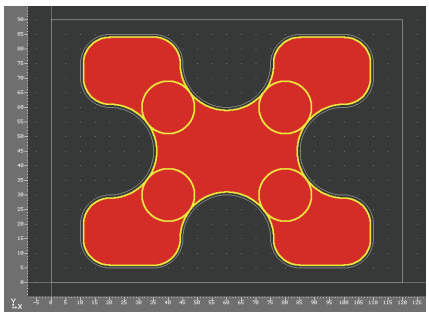
NOTE:
A surface to be machined can "collapse" into several partial surfaces although only one surface was selected for machining (see pictures concerning 'Helix'). In this case, the "Manual" option cannot be selected because several surfaces to be machined exist. Then, if necessary, use the 'Vertical' cut-in strategy with the aid of a pilot hole. As an alternative, you can also machine each partial surface in a separate work step.

Cutting-in strategy 'Ramp' in the 'Surface' work step

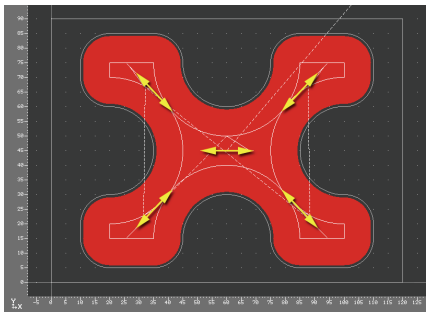
Cutting-in strategy 'Ramp'
The tool always cuts into the material along a ramp up to the respective approach depth. If an approach point is outside the material, cutting in is automatically directly carried out vertically. It is only possible to use tools for which the corresponding features have been set in the 'Set-up' operating mode.

NOTE:
When selecting a tool for the work step which should preferably cut-in along a ramp, it is possible, depending on the set allowance and/or the helix diameter that none or only individual (red) surfaces to be machined are displayed.

An example for this:

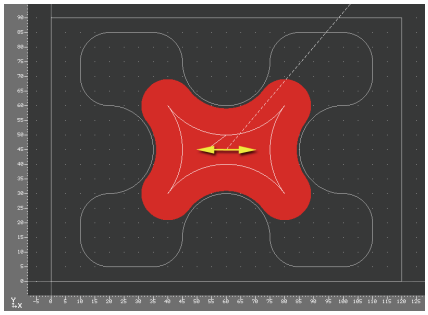


Due to the existing contour allowance of 1mm and taking the tool diameter of 18mm into account, 5 individual surfaces are already generated during calculation of the possible surfaces to be machined.



With a set ramp length (= groove length) of 36mm cut-in points can be calculated in all 5 surfaces because the ramp length fits into these surfaces. The ramp is always centralised in relation to the cut-in point.

In this example, you can see that the angle position of the ramp (the milling cutter centre lines are displayed in yellow here) was automatically adapted in the four external surfaces.



With a set ramp length of 38mm a cut-in point can only be calculated for the central surface because the ramp length no longer fits into the other surfaces with no ascertainable angle.

Solution:
In this case, reduce either the allowance, e.g. to 0.5mm or the ramp length to 36mm.

Automatic ramp angle adaptation
The software tries to automatically find an angle position within the surfaces to be machined under which the ramp can be executed. This is an approximation technique used to consecutively check various angle positions. As a standard, the process starts at 0° and executes max. 16 checks.

If it is necessary to always cut-in under a specific angle position for production reasons, the calculation process can be accelerated by changing the preset value for the start angle:

1. Open the `vorein.ini` file in the folder of the user data via an editor.
2. Enter a new line containing `RampW=X` (`X` = Start angle, standard is 0°) under `[Werken]`.

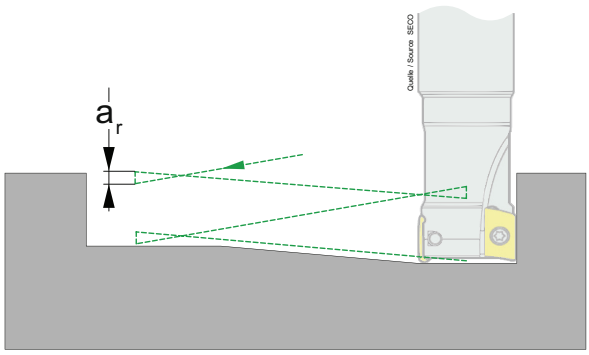
When a higher accuracy is required for the above described approximation technique, the preset value for the number of angle position checks can be respectively changed (Attention: with each increase in the iteration steps by one point, the max. number of checks doubles. For this reason, the required calculation time will increase extensively!):

1. Open the `vorein.ini` file in the folder of the user data via an editor.
2. Enter a new line containing `RampNmax=X` (`X` = Number of iteration steps, standard is 4) under `[Werken]`.

'Cutting-in angle' and 'Pitch'
The cut-in angle is a maximum admissible value corresponding to the respective tool. For this, also refer to the corresponding specifications of the tool manufacturer.
The pitch is automatically calculated from the respective ramp length and the max. cut-in angle, but it can also be directly preset.

If the approach dimension for the work step is selected smaller than the pitch, the tool path on the ramp is automatically reduced for the NC output.

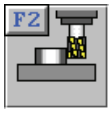
'Return value'
The tool is lifted by a value a_r each time the cut-in direction is changed (see drawing).
This value is previously specified by the tool manufacturer. This prevents that excessive material is left over "in the core" with the respective tools which could then result in a tool break.



'Reference point'
The reference point complies with the geometrical ramp start point. The real start point is always automatically in the centre the ramp (see drawing for 'Return value'). If only one surface is selected for machining, the reference point can also be determined manually.

NOTE:
A surface to be machined can "collapse" into several partial surfaces although only one surface was selected for machining (see pictures concerning 'Ramp'). In this case, the "Manual" option cannot be selected because several surfaces to be machined exist. Then, if necessary, use the 'Vertical' cut-in strategy with the aid of a pilot hole. As an alternative, you can also machine each partial surface in a separate work step.

Contour



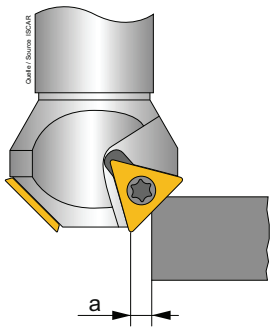
Use this work step for finishing work, for bevelling (with centre drills or bevel cutters) and for partial contouring work. Here work steps are meant requiring, for instance, the cutting of one edge of a cast workpiece to the desired dimension in several steps. For this purpose, you create several contour work steps on the corresponding contour, bearing in mind that the allowances have to constantly decrease, and the traverse paths kept constant.

The contour work step offers, via the <F1> 'Finished part' symbol, the possibility to machine contours already existing in the workplan (from the 'Geometry' operating mode).

The <F2> 'Partial contour' symbol can be used to quickly create additional contours - partial contours - along which traverse should take place. This type of traverse path setting via a geometry entry directly in the workplan is used for machining work directly on the unmachined part as well as for complex traverse paths for which it is not possible to create a useful construction in the 'Geometry' operating mode. This is valid, for example, for crossing traverse paths on a housing sealing surface.

Bevelling with the 'Contour' work step

When an NC spot drill or bevelling cutter were selected as the current tool, <F2> 'Values' in the second dialog window can be used to enter the respective specifications concerning the bevel geometry.



When entering the bevel width or the bevel depth dimension and the distance to be adhered to 'Cross cutter←→Contour' (see figure), the software automatically calculates the respective traverse paths.

Take into account that the tool is also traversed deeper in the Z direction corresponding to the set distance. This can result in collisions with subjacent surfaces/contours.

NOTE:
In the software, only the cross cutter of the tool is always displayed. Select the approach arc or approach path (depending on the bevel size) respectively large to prevent a collision with the actual blade during cutting in at the approach point.

Traverse paths along finished part contours

In the second dialog window, the values are to be changed in the 'Radius correction' window from the '**Control**' setting to the '**CAM**' setting. If you employ an older control version which commits compensation errors with circular arc/circular arc/circular arc designs. In this case, the NC program for the equidistant is created with the 'CAM' setting. Remember that also the circular arc and the contour allowance can vary in this window.

We wish to point out that in addition to entire contours also any number of partial paths can be created on each contour.

'Multiple' choice
The strategy using Multiple choice, i. e. several contours are finished in one work step, is efficient, however, requires somewhat more concentration (Instead, you can create each traverse path individually in the form of **one** work step).

We strongly recommend that you always select the desired contours first with <F1> 'Multiple' (if only one contour is to be machined, you can also select this contour directly by pressing the <plus/minus> keys). With <F2> 'Values' you define certain framework conditions for the contour presented in red. With multiple choice you select one contour after the other and enter the values. As a last step, you define the individual traverse paths on the contours for all selected contours using <F3> 'Traverse path'. The contour selection order and the order when creating the traverse paths defines the later production order.

'Traverse path'
You use this option to define where and how the individual traverse paths have to start and end. For this purpose, you can determine the approach and retract points and define whether or not the type of approach is to be 'circular' or 'linear'.
A special feature is the linear approach and retract, knowing that you can use this option to define any desired cut-in and retract point. In this manner, you can, for instance, plunge into a drilled hole and then approach the individual contour linearly.

Traverse paths along any partial contours

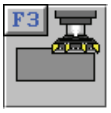
In the second dialog window, <F1> 'Geometry' can be used to invoke the geometry editor. Then you can decide whether a contour already existing is to be imported from a workpiece file or from the finished part of the current workplan or directly created new in the work step.

Please observe that only one partial contour can be used for each work step. When a further partial contour is selected, the first partial contour is deleted including all settings from the first work step.

<F3> 'Approach/Retract' is used to determine whether the traverse path should start or end 'Circular', 'Linear' or 'Direct', i.e. corresponding to the set 'Traverse side'. The start or end point of a contour is always automatically the corresponding approach and retract point of the traverse path. If the approach/retract point is to be located at a different point of the contour, change this respectively.

'Traverse side' and **'Direction'** or **'Start point'**
The 'Traverse side' option is used to determine the side of the contour on which the traverse is to be carried out depending on the 'Direction' setting. When the 'Centre' option is set in 'Traverse side', the 'Start point' can be set to the start or end point of the contour.

Facing



In the FACING work step, the tool always only moves to the centre point in the desired direction over the left-hand and right-hand workpiece tool boundary.
If a longer approach and overshoot path is desired for reason of surface properties, the suitable tool paths can be generated by means of the 'Macro' operating step and selecting the corresponding point 'SURFACE MILLING'.

Drilling

While generally the corresponding holes are designed in the 'Geometry' operating mode, you can do without designing the holes in the 'Geometry' operating mode and insert these holes later using the <F2> 'Individual' to <F5> 'Pattern' functions. Typically, as already mentioned, the holes are designed in the 'Geometry' operating mode and then automatically selected by choosing the <F1> 'Finished part' function.

The holes and the tool diameter are completely independent of each other, i.e. you can use a milling cutter with a diameter of 6 to sink into a hole with a diameter of 5, etc. However, no holes are proposed which could result in a collision. For example, if you want to enlarge a hole by means of a boring bar which has not been pre-drilled large enough, you cannot select this hole. Here, the chamfer Ø of the individual tool plays a role.

'Order'
If several holes or drilling patterns were selected, you can use the 'Order/Sequence' switch to decide if the holes are to be machined in the selected order or automatically following the principle of the shortest possible spacing. With complex drilling patterns, either one or the other method can prove advantageous.

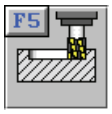
'NC output'
If you set the 'NC output' field to 'With cycle', boring cycles in accordance with your controls are put out in the NC program.

If you have to solve special production tasks, for instance deep hole drilling with several tools, start by switching the 'NC output' field to 'Without cycle'. This activates the two symbols <F1> 'Approach' and <F2> 'Pre-drilling':

'Approach'
This function offers you the possibility, for instance when deep drilling with a follow-up tool, to plunge into or retract from partially drilled holes produced by the first tool at an increased advance speed.

'Pre-drilling'
Under very difficult chip removal conditions, this function offers you the possibility to move along a path to be defined by you at the start and at the end of the individually selected holes at a reduced advance speed, the 'Pre-drilling advance'.

Groove



When grooves are finished, the diameter of the tool used must always be smaller than the groove width. However, you can also use a tool the diameter of which equals the groove width.
In the second dialog, you can define how the tool is to approach in the groove using the <F2> 'Approach' option:

'Equal'
This function is used to have the tool always cut in either at the 'start' or at the 'end' of the groove, depending on your pre-selection. To this effect, the tool is moved out of the groove in the tool axis and moved to the corresponding approach point.
'Alternating'
This function is used to approach the tool always at the start and end of the groove.
'Special'
This option is intended for disk-type cutters and T groove cutters. This function is used to have the tool always cut in either at the 'start' or at the 'end' of the groove, depending on your pre-selection. To this effect, the tool is returned along the groove for instance to the start of the groove (with T grooves this is usually outside of the workpiece), without changing its current machining depth. Only then is the approach performed.

Handwheel/Teach-In

Even the "most intelligent" software cannot know all your production requirements. If you use a special tool and have to travel along a very specific path, or if you are not happy with the travelling paths automatically created, you have the possibility to enter this path with this work step. Milling over a sealing surface of a cast component, for instance, is such a case.

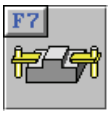
Every traverse path is displayed as a groove. In this manner, you can check quite easily if the traverse paths have been correctly arranged. Later modification of these traverse paths is, of course, possible.
Traverse paths travelled at feed speed are displayed green, those travelled at rapid traverse with a red frame.

In the ’HANDWHEEL’ work step it is possible to graphically change the tool position interactively in some input fields under ’Position’ (marked with) and under ’Manual’:

- For this switch on the "keyboard handwheel" first. <Shift>+<F4>
- Handwheel increment increase <Shift>+<F5>
  Handwheel increment reduction <Shift>+<F6>
  Here, observe the pointer position
  underneath the actual value display: 

- Moving in the positive direction <Alt>+<Scroll down>
  Moving in the negative direction <Alt>+<Scroll up>

Clamping



If several clamping situations are required, the originally created clamping situation can be cancelled here and a new clamping situation defined.
This work step can also be used any time you wish to change the return level or the tool change point.
When the workpiece can be produced in one clamping position and without changing the return level or the workpiece change point, you do not require this work step. All required clamping data has already been entered when creating the work plan.

Macro

You know better than anybody else how different the individual workpieces and machining possibilities are.
SYM*plus* includes the corresponding work steps for all standard machining processes when milling.

For an extended functionality we also delivery machining macros, such as for instance ’SURFACE MILLING’.
Even if you have to produce a special part or look for a lower cost way of production, this will be no problem. In most cases we are able to create special work steps in the form of customised machining macros.

4.5.3 ’NC output’ menu

This subject was also dealt with in chapter ’General complementary information’ under the point ”’NC output’ menu” on page 36.

4.6 ’Teacher’ operating mode

This subject was also dealt with in chapter ’General complementary information’ under the point ”’Teacher’ operating mode” on page 36.

4.7 ’Set-up’ operating mode

This subject was also dealt with in chapter ’General complementary information’ under the point "’Set-up’ operating mode" on page 39.

4.8 ’Transfer’ operating mode

This subject was also dealt with in chapter ’General complementary information’ under the point "’Transfer’ operating mode" on page 45.

5 Solutions to the exercises in the work documents

All data to be read from files (work plans, workpieces, tools) refer to the sample files supplied with the SYMplus version 5.1. Deviations can result from changes which have been made to the installation or if you work with a different version.

5.1 Solutions to exercises in the "Workbook Turning"

Page	Chapter	Solution	Remark
9	1.1.1	Did we surprise you?: I can choose between coffee or cappuccino. Sometimes the vending machine dispenses the drink before the cup is in position.	
10	1.1.2	Measuring range:[0-150] Measuring accuracy:[0.05] Accuracy of the vernier:[0.05] Figure on the left:[20] + [9] + [0.4] = [29.4] Figure on the right:[20] + [3] + [0.65] = [23.65] LH measuring value = [24.7] RH measuring value = [37.55]	
11	1.1.2	Measuring range:[25-50] Measuring accuracy:[0.01] Spindle pitch:[0.5] Figure on the left:[46] + [0.35] = [46.35] Figure on the right:40] + [0.5] + [0.23] = [40.73] LH measuring value = [27.43] Centre measuring value = [36.71] RH measuring value = [47.59]	
12	1.1.2	Red = [reject side] Yellow = [go-side] Nominal size:[42] Tolerance:[16] Minimum dimension:[41.984] Maximum dimension:[42] Tolerance range: [h6] Workpiece A Left: The go-side is OK. The maximum dimension is not exceeded. Right: The reject side is not OK. The minimum dimension is not gone below. Result: The workpiece is OK. Workpiece B Right: The go-side is not OK. The maximum dimension is exceeded. Result: The workpiece needs to be reworked. Workpiece C Left: The go-side is OK ... Right: The reject side is OK ... Result: The workpiece is REJECT.	

13	1.1.2	Red = [reject side] Grey = [go-side] Nominal size:[50] Tolerance:[25] Minimum dimension:[50] Maximum dimension:[50.025] Tolerance range: [H7] Workpiece A Left: The go-side is OK. The minimum dimension is not gone below. Right: The reject side is not OK. The maximum dimension is not exceeded. Result: The workpiece is OK. Workpiece B Right: The go-side is not OK. The minimum dimension is gone below. Result: The workpiece needs to be reworked. Workpiece C Left: The go-side is OK ... Right: The reject side is OK ... Result: The workpiece is REJECT.	
15	1.1.3	Forces and deformations: Operating force Pre-clamping force Δl_1 Change of screw length Δl_2 Change of workpiece thickness Diagram: Joining point Yield point Breaking point Elastic deformation Plastic deformation	
16	1.1.4	Clamping situation: Number 1 Reason: The maximum possible lever arm length l_1 is reached at this position, the maximum clamping force is reached. Clamping possibility 1: When a support angle between 0° and max. 5° of the clamping element can be reached with the given stepping of the clamping supports. Clamping possibility 2: When the coarse adjustment according to the stepping is completed, the upper section of the clamping support can be infinitely precision adjusted within a 2 mm range by lateral sliding on an inclined plane. Clamping possibility 3: At an angle of >0°, the clamping element only makes contact with one tangent point and could easily slip off. Typically, a vice is used to support and align all types of workpieces.	
17	1.1.4	Clamping pressure: 400 bar Force: 28274 N Ensure stable support directly beneath the clamping points. Advantage: Reduction of the down/make-ready times on production machines. Positioning precision: <0.005mm	

19	1.2.2	Clamping pressure: Material, clamping position, overhang, cutting data v_{max} = [60 m/min] a_{max} = [15 m/sec^{2}] dual: [0]·2^3+[1]·2^2+[1]·2^1+[1]·2^0 decimal value of 0111:[0]+[4]+[2]+[1]=[7]	
20	1.2.2	Voltage supply 1. Filter 2. Transformer 3. Rectifier 4. Filter network 5. Regulation Drive module 1. NC reads 1st block 2. Setpoint value transmission to the axis module 3. Motor starts 4. Setpoint/actual value comparison 5. If setpoint value equals actual value PLC Never bridge the safety switches fitted to the doors. What caused the crash? Values at G54 were not set. What does Z198 at G54 mean? Zero point offset between MNP and WNP.	
21	1.2.3	Texts referring to the keys are displayed when clicked on with the mouse.	
22	1.3.1	Turn on the machine using the main switch on the rear and wait until the machine has started. 1. Rearm the EMERGENCY STOP button 2. Switch on the drives for the spindle and advance axes 3. Delete the alarm number [+X] The reference point in X is moved to [+Z] The reference point in Z is moved to	
23	1.3.1	[Jog] operating mode: Machine in manual mode [+T]: Swivel the turret to the next station (tool change) G97: Constant speed S1000: Set the speed to 1,000 rpm M4: Switch on spindle (CCW run) [MDA] operating mode: Entry and machining of individual blocks [Cycle Start]: NC block machining Crash between tool and workpiece	
24	1.3.1	[Section toggle]: Invoke basic menu Select the [Parameter] softkey Select the [Zero point offset] softkey Select the [Determine] softkey [Jog] operating mode: Manual mode, move axes [VAR] operating mode: Increment mode, set the increment width Select the [Compensate] softkey Offset value Z: 151.277	
26	1.4	- Long hair implies high risks - Wearing of necklaces or similar is dangerous - Clothes must be tight fitting - Always wear safety shoes	

27	1.4	- Never operate on the grinding block without wearing protection goggles - Most severe injury can occur if you do not wear a hat or hair net - Never try to hold workpieces with your bare hands, always use suitable clamping means - Never remove chips with your bare hands or even when wearing gloves	
30	2.3.1	Exercise 1 N G X Z I K N1 G0 X0 Z1 N2 G1 Z0 N3 G1 X14 N4 G1 X20 Z-3 N5 G1 Z-10 N6 G2 X40 Z-20 I10 K0 N7 G1 Z-30 N8 G2 X60 Z-40 I10 K0 N9 G1 X70 N10 G1 X80 Z-60 N11 G1 Z-80 N12 G1 X82 N13 G0 X120 Z10	
30	2.3.2	Exercise 2 N G X Z R N1 G0 X0 Z1 N2 G1 Z0 N3 G1 X16 N4 G3 X20 Z-2 R10 N5 G1 Z-20 N6 G2 X40 Z-30 R10 N7 G3 X60 Z-40 R10 N8 G1 Z-50 N9 G2 X80 Z-60 R10 N10 G1 Z-80 N11 G1 X82 N12 G0 X120 Z10	
31	2.3.3	Exercise 5 N G X Z I K R N1 G0 X82 Z0 N2 G1 X0 N3 G1 Z1 N4 G0 X64 N5 G1 X70 Z-2 N6 G1 Z-9 N7 G2 X72 Z-10 I1 K0 N8 G1 X75 N9 G1 X76 Z-10.5 N10 G1 Z-20 N11 G1 X72 Z-30 N12 G1 Z-37 N13 G2 X78 Z-40 I3 K0 N14 G1 Z-50 N15 G2 X78 Z-70 R20 N16 G1 Z-75 N17 G1 X82 N18 G0 X120 Z10	
36	2.5.3.1	V = [0.551] dm^3 m = [4.327] kg	

38	2.5.4.1	Exercise 6 N1 G54 N2 G92 S3000 N3 G96 S200 F0.3 T1 M4 N4 G0 X82 Z0 M8 N5 G1 X-1.6 N6 Z1 N7 G0 X75 N8 G1 Z-72 F0.3 N9 X81 N10 G0 Z1 N11 X70 N12 G1 Z-48 N13 X76 N14 G14 M9 N15 M30	
39	2.5.4.2	Exercise 7 N1 G54 N2 G92 S3000 N3 G96 S200 F0.3 T1 M4 N4 G0 X82 Z0 M8 N5 G1 X-1.6 N6 Z1 N7 G14 M9 N8 G97 S1500 F0.1 T8 M3 N9 G0 X0 Z1 N10 G1 Z-50 N11 G0 Z1 N12 G14 M9 N13 G96 S180 F0.2 T4 M4 N14 G0 X23 Z1 M8 N15 G1 Z-32 N16 X19 N17 G0 Z1 N18 X26.011 N19 G1 Z-32 N20 X22 N21 G0 Z1 N22 G14 M9 N23 M30	
40	2.5.5.1	Exercise 8 Actual value 1. D = [70] mm 2. D = [76] mm 3. D = [72] mm 4. D = [78] mm All axis parallel diameters are dimensional precise.	
41	2.5.5.2	Exercise 9 Actual value A E Bevel X[66.467] X[70] Z[0] Z[-1.766] Radius X[70] X[72.8] Z[-8.6] Z[-10] All elements, which are not axis parallel, are NOT dimension precise.	
42	2.5.6.1	Grey: Equidistant line Yellow: Set point contour Red: Path of the theoretic tool tip	

43	2.5.6.2	Exercise 10 Actual value A E Bevel X[66] X[70] Z[0] Z[-2] Radius X[70] X[72] Z[-9] Z[-10] All elements, which are not axis parallel, are dimension precise due to G42.	
45	2.5.7.2	Advantages: 1. shorter 2. better legible 3. easier editable	
46	2.5.7.3	Exercise 11 N1 G54 N2 G92 S3000 N3 G96 S200 F0.3 T1 M4 N4 G0 X82 Z0 M8 N5 G1 X-1.6 N6 Z1 N7 G0 X80 N8 G81 D2.5 AX0.5 AZ0.2 N9 G1 X0 Z0 N10 X16 N11 X20 Z-2 N12 Z-30 N13 X44 N14 G3 X50 Z-33 R3 N15 G1 Z-50 N16 X80 Z-62 N17 G80 XA16 N18 G14 M9 N19 G96 S240 F0.1 T3 M4 N20 G0 X0 Z1 M8 N21 G42 N22 G23 N9 N16 N23 G40 N24 G1 X82 Z-63 N25 G14 M9 N26 M30	
47	2.5.7.4	Exercise 12 N1 G54 N2 G92 S3000 N3 G96 S180 F0.15 T3 M4 N4 G0 X82 Z0 M8 N5 G1 X-0.8 N6 Z1 N7 G0 X80 N8 G81 D1.5 AX0.5 AZ0.2 N9 G1 X0 Z0 N10 G3 X24 I0 K-20 N11 G1 Z-20 RN5 N12 X40 N13 X60 Z-30 N14 Z-35 N15 G2 Z-55 R20 N16 G1 Z-70 RN2 N17 X70 RN2 N18 X80 Z-85 N19 G80	

47	2.5.7.4	N20 G96 S240 F0.1 N21 G0 X0 Z1 N22 G42 N23 G23 N9 N18 N24 G40 N25 G1 X82 Z-86 N26 G14 M9 N27 M30	
50	2.5.9.2	Exercise 13 N1 G54 N2 G92 S3000 N3 G96 S180 F0.15 T3 M4 N4 G0 X82 Z0 M8 N5 G1 X-0.8 N6 Z1 N7 G0 X80 N8 G81 D1.5 AX0.5 AZ0.2 N9 G1 X0 Z0 N10 G1 X36 RN2 N11 Z-5 N12 G2 Z-25 R15 N13 G1 Z-30 RN2 N14 X64 N15 X70 AS150 N16 Z-40 RN10 N17 G3 Z-60 R15 RN10 N18 G1 Z-70 RN2 N19 X80 N20 G80 XA30 N21 G96 S240 F0.1 N22 G0 X0 Z1 N23 G42 N24 G23 N9 N19 N25 G40 N26 G1 X82 N27 G14 M9 N28 M30	
51	2.5.9.3	Exercise 14 N1 G54 N2 G92 S3000 N3 G96 S180 F0.15 T3 M4 N4 G0 X62 Z0 M8 N5 G1 X-0.8 N6 Z1 N7 G0 X60 N8 G81 D1.5 AX0.5 AZ0.2 N9 G1 X0 Z0 N10 G3 X20 I0 K-20 N11 G1 Z-10 RN2 N12 X30 N13 G3 X36 IA20 KA-20 RN5 O2 N14 G1 Z-40 RN4 N15 X60 RN-2 N16 Z-43 N17 G80 N18 G96 S240 F0.1 N19 G0 X0 Z1 N20 G42 N21 G23 N9 N16 N22 G40 N23 G1 X82 N24 G14 M9 N25 M30	

52	2.5.10.1	Exercise 15 N1 G54 N2 G92 S3000 N3 G96 S200 F0.3 T1 M4 N4 G0 X82 Z0 M8 N5 G1 X-1.6 N6 Z1 N7 G14 M9 N8 G97 S1500 F0.1 T8 M3 N9 G0 X0 Z1 M8 N10 G84 ZA-80 N11 G14 M9 N12 G96 S180 F0.1 T10 M4 N13 G0 X18 Z1 M8 N14 G81 D1.5 AX-0.5 AZ0.2 N15 G1 X64 Z0 N16 X60 Z-2 N17 Z-15 N18 X30 AS-120 N19 Z-40 N20 G3 X30 Z-60 R18 N21 G1 Z-72 RN3 N22 X18 N23 G80 N24 G96 S240 F0.1 N25 G0 X64 Z1 N26 G41 N27 G23 N15 N22 N28 G40 N29 G0 Z1 N30 G14 M9 N31 M30	
53	2.5.10.2	Exercise 16 N1 G54 N2 G92 S3000 N3 G96 S200 F0.3 T1 M4 N4 G0 X52 Z0 M8 N5 G1 X-1.6 N6 Z1 N7 G14 M9 N8 G97 S800 F0.1 T30 M3 N9 G0 X0 Z1 M8 N10 G84 ZI-6 N11 G14 M9 N12 G97 S1200 F0.1 T25 M3 N13 G0 X0 Z1 M8 N14 G84 ZI-16 N15 G14 M9 N16 G97 S400 T28 M3 N17 G0 X0 Z3 M8 N18 G32 Z-10.5 F1.5 N19 G14 M9 N20 M999	

53	2.5.10.2	N21 G96 S200 F0.3 T1 M4 N22 G0 X50 Z1 M8 N23 G81 D2.5 AX0.5 AZ0.2 N24 G1 X0 Z0 N25 X30 RN-2 N26 G85 X30 Z-35.2 I1 K4 N27 G1 X50 RN-1.5 N28 Z-37 N29 G80 XA26 N30 G14 M9 N31 G96 S240 F0.1 T3 M4 N32 G0 X0 Z1 M8 N33 G42 N34 G23 N24 N28 N35 G40 N36 G14 M9 N37 G97 S1900 T5 M3 N38 G0 X30 Z4.5 M8 N39 G31 X30 Z-34 F1.5 D0.92 Q8 N40 G14 M9 N41 M30	
56	2.5.10.2	Exercise 17 N1 G54 N2 G92 S3000 N3 G96 S200 F0.3 T1 M4 N4 G0 X82 Z0 M8 N5 G1 X-1.6 N6 Z1 N7 G14 M9 N8 G97 S1500 F0.1 T8 M3 N9 G0 X0 Z1 M8 N10 G84 ZA-70 N11 G14 M9 N12 G96 S180 F0.2 T4 M4 N13 G0 X18 Z1 M8 N14 G81 D1.5 AX-0.5 AZ0.2 N15 G1 X54 Z0 N16 Z-40 N17 X38.38 RN-1.5 N18 G85 X38.38 Z-60 I1.06 K5.6 N19 G1 X18 N20 G80 N21 G14 M9 N22 G96 S180 F0.1 T10 M4 N23 G0 X54 Z1 M8 N24 G41 N25 G23 N15 N19 N26 G40 N27 G0 Z1 N28 G14 M9 N29 G97 S1000 T12 M4 N30 G0 X38.38 Z1 M8 N31 Z-34 N32 G31 X38.38 Z-58 F1.5 D0.81 Q11 O1 N33 G0 Z1 N34 G14 M9 N35 M30	
55	2.5.11	1234: Invoke subroutine 1234 2: Repeat twice	

56	2.5.12.1	Exercise 18 Main program: N1 G54 N2 G92 S3000 N3 G96 S200 F0.3 T1 M4 N4 G0 X81 Z0 M8 N5 G1 X-1.6 N6 Z1 N7 G0 X80 N8 G81 D2.5 AX0.5 AZ0.2 N9 G1 X0 Z0 N10 X37 RN4 N11 Z-22 N12 X44 RN2 N13 X54 AS160 N14 Z-48 RN4 N15 X64 N16 Z-60 RN10 N17 X80 AS150 N18 G80 XA30 N19 G14 M9 N20 G96 S240 F0.1 T3 M4 N21 G0 X0 Z1 M8 N22 G42 N23 G23 N9 N17 N24 G40 N25 G14 M9 N26 G96 S100 F0.1 T7 M4 N27 G0 X38 Z-10 M8 N28 G22 L50 N29 G0 Z-14 N30 G22 L50 N31 G14 M9 N32 M30 Subroutine: N1 G91 N2 G1 X-1.75 F0.1 N3 G4 U0.5 N4 X1.75 F0.3 N5 Z0.8 N6 X-0.8 Z-0.8 F0.05 N7 X0.8 F0.3 N8 Z-0.8 N9 X-0.8 Z0.8 F0.05 N10 X0.8 N11 G90 N12 M17	
56	2.5.12.1	Width: [3] mm Cutting radius: [0.2] mm Bevel length: [0.259] mm Bevel width: [0.183] mm Elements, which are not axis parallel, are not dimension precise without cutting radius correction.	

57	2.5.12.1	Exercise 19 N1 G54 N2 G92 S3000 N3 G96 S200 F0.3 T1 M4 N4 G0 X82 Z0 M8 N5 G1 X18 N6 Z2 N7 G0 X72 N8 G42 N9 G1 Z0 N10 X78 RN2 N11 Z-40 N12 X82 N13 G40 N14 G14 M9 N15 G97 S1500 F0.1 T8 M3 N16 G0 X0 Z2 M8 N17 G84 ZA-70 N18 G14 M9 N19 G96 S180 F0.2 T4 M4 N20 G0 X20 Z1 M8 N21 G81 D1.5 N22 G1 X54 Z0 N23 X50.013 RN-1 N24 Z-35 N25 X40 RN-0.5 N26 Z-55 RN2 N27 X34 N28 X20 AS215 N29 G80 N30 G14 M9 N31 G96 S240 F0.1 T10 M4 N32 G0 X54 Z1 M8 N33 G41 N34 G23 N22 N28 N35 G40 N36 G0 Z1 N37 G14 M9 N38 G96 S100 F0.1 T6 M3 N39 G0 X48 Z1 M8 N40 Z-5 N41 G86 X50.013 Z-9 ET54 EB4 RO-0.3 D2 N42 G0 Z-30 N43 G86 X50.013 Z-32 ET54 EB4 RO-0.3 D2 N44 G0 Z1 N45 G14 M9 N46 M30	
57	2.5.12.1	External: [4.6] mm Internal: [4] mm Bevel : [0.3] mm G86 internally contains the cutting radius correction.	

58	2.5.12.2	Exercise 20 N1 G54 N2 G92 S3000 N3 G96 S200 F0.3 T1 M4 N4 G0 X82 Z0 M8 N5 G1 X-1.6 N6 Z1 N7 G14 M9 N8 G96 S180 F0.15 T3 M4 N9 G0 X80 Z1 M8 N10 G81 D1.5 AX0.5 AZ0.2 N11 G1 X0 Z0 N12 X30 RN-2 N13 G85 X30 Z-25 I1.15 K3.8 N14 G1 X55 RN3 N15 Z-35 N16 G2 X55 Z-65 R25 N17 G1 Z-75 RN5 N18 X80 AS120 N19 G80 XA25 N20 G96 S240 F0.1 T3 M4 N21 G0 X0 Z1 M8 N22 G42 N23 G23 N11 N18 N24 G40 N25 G1 X82 N26 G14 M9 N27 G97 S800 T11 M3 N28 G0 X35 Z5 M8 N29 G31 X30 Z-24 F1.5 D0.92 Q11 O1 N30 G14 M9 N31 G96 S100 F0.1 T7 M4 N32 G86 X80 Z-95 ET74 EB6 D1.5 AS15 RO1 RU1 M8 N33 G14 M9 N34 M30	
58	2.5.12.2	To be preferred: [A] Only cutting-in reduces the workpiece cross section for the following operating steps.	
59	2.5.12.3	Exercise 21 N1 G54 N2 G92 S3000 N3 G96 S200 F0.3 T1 M4 N4 G0 X82 Z0 M8 N5 G1 X-1.6 N6 Z1 N7 G14 M9 N8 G96 S180 F0.15 T3 M4 N9 G0 X80 Z1 M8 N10 G81 D1.5 AX0.5 AZ0.2 N11 G1 X0 Z0 N12 G3 X20 I0 K-20 RN5 O2 N13 G1 X40 Z-50 N14 X80 RN-2 N15 Z-52 N16 G80 N17 G14 M9	

59	2.5.12.3	N18 G96 S240 F0.1 T3 M4 N19 G0 X0 Z1 M8 N20 G42 N21 G23 N11 N15 N22 G40 N23 G1 X82 N24 G14 M9 N25 G96 S100 F0.1 T9 M4 N26 G88 X58 Z-50 ET-53 EB8 AS30 AE30 RO1 RU1 D1.5 M8 N27 G14 M9 N28 M30 Time: [7:00] min	
59	2.5.12.3	Using DAL80 is not reasonable because: The tool should not cut in because residual material is left anyway.	
59	2.5.12.3	Exercise 21 (optimised) N1 G54 N2 G92 S3000 N3 G96 S200 F0.3 T1 M4 N4 G0 X82 Z0 M8 N5 G1 X-1.6 N6 Z1 N7 G0 X80 N8 G81 D2.5 AX0.5 AZ0.2 AE180 AV0 N9 G1 X0 Z0 N10 G3 X20 I0 K-20 RN5 O2 N11 G1 X40 Z-50 N12 X80 RN-2 N13 Z-52 N14 G80 N15 G14 M9 N16 G96 S180 F0.15 T3 M4 N17 G0 X42 Z-20 M8 N18 G81 D1.5 AX0.5 AZ0.2 N19 G23 N9 N13 N20 G80 N21 G0 Z1 N22 G96 S240 F0.1 N23 G0 X0 Z1 N24 G42 N25 G23 N9 N13 N26 G40 N27 G1 X82 N28 G14 M9 N29 G96 S100 F0.1 T9 M4 N30 G88 X58 Z-50 ET-53 EB8 AS30 AE30 RO1 RU1 D1.5 M8 N31 G14 M9 N32 M30 Time: [2:59] min	
65	2.5.15	The solution is saved in the software under the "Teacher" operating mode.	
66	2.5.15	The solution is saved in the software under the "Teacher" operating mode.	
67	2.5.15	The solution is saved in the software under the "Teacher" operating mode.	

5.2 Solutions to exercises in the "Workbook Milling"

Page	Chapter	Solution	Remark
9	1.1.1	Did we surprise you? I can choose between coffee or cappuccino. Sometimes the vending machine dispenses the drink before the cup is in position.	
10	1.1.2	Measuring range:[0-150] Measuring accuracy:[0.05] Accuracy of the vernier:[0.05] ?Figure on the left:[20] + [9] + [0.4] = [29.4] Figure on the right:[20] + [3] + [0.65] = [23.65] LH measuring value = [24.7] RH measuring value = [37.55]	
11	1.1.2	Measuring range:[25-50] Measuring accuracy:[0.01] Spindle pitch:[0.5] ?Figure on the left:[46] + [0.35] = [46.35] Figure on the right:40] + [0.5] + [0.23] = [40.73] LH measuring value = [27.43] Centre measuring value = [36.71] RH measuring value = [47.59]	
12	1.1.2	Red = [reject side] Yellow = [go-side] Nominal size:[42] Tolerance:[16] Minimum dimension:[41.984] Maximum dimension:[42] Tolerance range: [h6] Workpiece A Left: The go-side is OK. The maximum dimension is not exceeded. Right: The reject side is not OK. The minimum dimension is not gone below. Result: The workpiece is OK. Workpiece B Right: The go-side is not OK. The maximum dimension is exceeded. Result: The workpiece needs to be reworked. Workpiece C Left: The go-side is OK ... Right: The reject side is OK ... Result: The workpiece is REJECT.	

13	1.1.2	Red = [reject side] Grey = [go-side] Nominal size:[50] Tolerance:[25] Minimum dimension:[50] Maximum dimension:[50.025] Tolerance range: [H7] Workpiece A Left: The go-side is OK. The minimum dimension is not gone below. Right: The reject side is not OK. The maximum dimension is not exceeded. Result: The workpiece is OK. Workpiece B Right: The go-side is not OK. The minimum dimension is gone below. Result: The workpiece needs to be reworked. Workpiece C Left: The go-side is OK ... Right: The reject side is OK ... Result: The workpiece is REJECT.	
15	1.1.3	Forces and deformations: Operating force Pre-clamping force Δl_1 Change of screw length Δl_2 Change of workpiece thickness Diagram: Joining point Yield point Breaking point Elastic deformation Plastic deformation	
16	1.1.4	Clamping situation: Number 1 Reason: The maximum possible lever arm length l_1 is reached at this position, the maximum clamping force is reached. Clamping possibility 1: When a support angle between 0° and max. 5° of the clamping element can be reached with the given stepping of the clamping supports. Clamping possibility 2: When the coarse adjustment according to the stepping is completed, the upper section of the clamping support can be infinitely precision adjusted within a 2 mm range by lateral sliding on an inclined plane. Clamping possibility 3: At an angle of >0°, the clamping element only makes contact with one tangent point and could easily slip off. Typically, a vice is used to support and align all types of workpieces.	
17	1.1.4	Clamping pressure: 400 bar Force: 28274 N Ensure stable support directly beneath the clamping points. Advantage: Reduction of the down/make-ready times on production machines. Positioning precision: <0.005mm	

<table>
<tr><td>19</td><td>1.2.2</td><td>
<table>
<tr><th></th><th>Knee-type milling machine</th><th>Cross slide table milling machine</th></tr>
<tr><td>+X</td><td>Table to the left</td><td>Table to the left</td></tr>
<tr><td>-X</td><td>Table to the right</td><td>Table to the right</td></tr>
<tr><td>+Y</td><td>Tool to the rear</td><td>Table to the front</td></tr>
<tr><td>-Y</td><td>Tool to the front</td><td>Table to the rear</td></tr>
<tr><td>+Z</td><td>Table down</td><td>Tool up</td></tr>
</table>
Air supply/air return:
Free air supply, throttled air return.
Advantages:
Fast unclamping, slow (=careful) clamping.

dual: $[0]\cdot 2^3+[1]\cdot 2^2+[0]\cdot 2^1+[1]\cdot 2^0$
decimal value of 0101:[0]+[4]+[0]+[1]=[5]
</td><td></td></tr>
<tr><td>20</td><td>1.2.2</td><td>
Voltage supply
1. Filter
2. Transformer
3. Rectifier
4. Filter network
5. Regulation

Drive module
1. NC reads 1st block
2. Setpoint value transmission to the axis module
3. Motor starts
4. Setpoint/actual value comparison
5. If setpoint value equals actual value

PLC
Never bridge the safety switches fitted to the doors.

What caused the crash? Values at G54 were not set
WNP calculation: By measuring with a 3D probe or radio measuring probe
</td><td></td></tr>
<tr><td>21</td><td>1.2.3</td><td>Texts referring to the keys are displayed when clicked on with the mouse.</td><td></td></tr>
<tr><td>22</td><td>1.3.1</td><td>
Turn on the machine using the main switch on the rear and wait until the machine has started.

1. Rearm the EMERGENCY STOP button
2. Switch on the drives for the spindle and advance axes
3. Delete the alarm number

[+Z] The reference point in Z is moved to
[+Y] The reference point in Y is moved to
[+X] The reference point in X is moved to
</td><td></td></tr>
<tr><td>23</td><td>1.3.1</td><td>
[MDA] operating mode: Entry and machining of individual blocks
T3: Select station 3 in the magazine
M6: Change tool
S1000: Set the speed to 1,000 rpm
M3: Switch on spindle (CW run)
[Jog] operating mode: Machine in manual mode
Table to the right: Key [-X]
</td><td></td></tr>
</table>

24	1.3.1	[Section toggle]: Invoke basic menu Select the [Parameter] softkey Select the [Zero point offset] softkey Select the [Determine] softkey Select the [OK] softkey [Jog] operating mode: Manual mode, move axes [VAR] operating mode: Increment mode, set the increment width Select the [Compensate] softkey Select the [Next axis] softkey Offset value X: 56.789 Offset value Y: -123.664 Offset value Z: -34.678	
26	1.4	- Long hair implies high risks - Wearing of necklaces or similar is dangerous - Clothes must be tight fitting - Always wear safety shoes	
27	1.4	- Never operate on the grinding block without wearing protection goggles - Most severe injury can occur if you do not wear a hat or hair net - Never try to hold workpieces with your bare hands, always use suitable clamping means - Never remove chips with your bare hands or even when wearing gloves	
30	2.3.1	Exercise 1 N G X Y I J N1 G0 X50 Y20 N2 G1 Y30 N3 G3 X40 Y40 I-10 J0 N4 G1 X20 N5 G2 X20 Y50 I0 J5 N6 G1 X40 N7 G3 X50 Y60 I0 J10 N8 G1 Y70 N9 G2 X70 Y70 I10 J0 N10 G1 Y60 N11 G3 X80 Y50 I10 J0 N12 G1 X100 N13 G2 X100 Y40 I0 J-5 N14 G1 X80 N15 G3 X70 Y30 I0 J-10 N16 G1 Y20 N17 G2 X50 Y20 I-10 J0	
30	2.3.2	Exercise 2 N G X Y R N1 G0 X20 Y20 N2 G1 Y70 N3 G2 X30 Y80 R10 N4 G1 X80 N5 G2 X90 Y70 R10 N6 G1 Y20 N7 G2 X80 Y10 R10 N8 G1 X30 N9 G2 X20 Y20 R10 N10 G0 X55 Y30 N11 G2 X55 Y60 R15 N12 G2 X55 Y30 R15	

31	2.3.3	Exercise 5 N G X Y Z I J R N1 G0 X35 Y25 Z1 N2 G1 Z-2 N3 G2 X5 I-15 J0 N4 G1 Y65 N5 G2 X35 I15 J0 N6 G0 Z1 N7 G0 X45 Y10 N8 G1 Z-2 N9 G1 Y80 N10 G1 X75 Y10 N11 G1 Y80 N12 G0 Z1 N13 G0 X115 Y65 N14 G1 Z-2 N15 G3 X85 R15 N16 G1 Y25 N17 G3 X115 R15 N18 G0 X-10 Y-10 Z100	
36	2.5.3.1	Volume [0.165] dm^3 Mass [1.295] kg	
39	2.5.4.1	Exercise 6 N1 G54 N2 T8 F150 S3000 M13 N3 G0 X11 Y11 Z1 N4 G1 Z-5 N5 G1 X61 F250 N6 Y37 N7 X11 N8 Y11 N9 X16 Y19 N10 X57 N11 Y28 N12 X12 N13 G0 Z1	
39	2.5.4.1	Exercise 7 N14 G0 X94 Y65 N15 G1 Z-5 F150 N16 X101 F250 N17 G2 I-7 J0 N18 G1 X109.01 N19 G2 I-15.01 J0 N20 G1 X94 Y65 N21 G1 Z-10 F150 N22 G1 X101 F250 N23 G2 I-7 J0 N24 G1 X109.01 N25 G2 I-15.01 J0 N26 G0 Z1	
39	2.5.4.1	Exercise 8 N27 G0 X22 Y68 N28 G1 Z-2.5 F150 N29 X48 F250 N30 G0 Z1 N31 G0 X35 Y81 N32 G1 Z-2.5 F150 N33 Y55 F250 N34 G0 Z100 M9	

40	2.5.4.2	Exercise 9 N35 T11 F300 S3500 M13 N36 G0 X22 Y55 Z1 N37 G1 Z-18 N38 G0 Z1 N39 G0 X48 N40 G1 Z-18 N41 G0 Z100 M9	
40	2.5.4.2	Exercise 10 N42 T39 F200 S2800 M13 N43 G0 X109 Y24 Z1 N44 G1 Z-19 N45 G0 Z1 N46 G0 X101.5 Y36.99 N47 G1 Z-19 N48 G0 Z1 N49 G0 X86.5 N50 G1 Z-19 N51 G0 Z1 N52 G0 X79 Y24 N53 G1 Z-19 N54 G0 Z1 N55 G0 X86.5 Y11.01 N56 G1 Z-19 N57 G0 Z1 N58 G0 X101.5 N59 G1 Z-19 N60 G0 X150 Y150 Z100 N61 M30	
41	2.5.5.1	Exercise 11 N1 G54 N2 T8 F150 S3000 M13 N3 G72 ZI-5 LP60 BP36 D5 V1 DB90 O2 E150 F300 N4 G79 X36 Y24 Z0 N5 G73 ZI-10 R20.005 D5 V1 O2 E150 F300 N6 G79 X94 Y65 Z0 N7 G0 Z100 M9 N8 T7 S4500 M13 N9 G74 ZI-2.5 LP36 BP10 D2.5 V1 E150 F250 N10 G79 X22 Y68 Z0 AR0 N11 G79 X35 Y55 Z0 AR90 N12 G0 Z100 M9 N13 T11 F250 S3500 M13 N14 G81 ZA-18 V1 N15 G76 X22 Y55 Z0 AS0 D26 O2 N16 G0 Z100 N17 T39 F200 S2800 M13 N18 G81 ZI-20 V1 N19 G77 IA94 JA24 Z0 R15 AN0 AI60 O6 N20 G0 X150 Y150 Z100 N21 M30	

42	2.5.5.2	Exercise 12 N1 G54 N2 G59 XA60 YA45 N3 T2 F480 S1000 M13 N4 G0 X-52.5 Y-58 Z-7 N5 G1 Y58 N6 G0 X52.5 Z-3 N7 G1 Y-58 N8 G0 Z100 M9 N9 T1 F120 S800 M13 N10 G81 ZI-5 V1 F120 N11 G76 X-50 Y-30 Z-7 AS90 D30 O3 W1 N12 G76 X50 Y-30 Z-3 AS90 D30 O3 N13 G0 Z100 M9 N14 T10 F200 S1200 M13 N15 G81 ZA-19 V1 N16 G76 X-50 Y-30 Z-7 AS90 D30 O3 W1 N17 G76 X50 Y-30 Z-3 AS90 D30 O3 N18 G0 X150 Y150 Z100 N19 M30	
42	2.5.5.2	A crash will occur (if a protruding step exists between the approach positions).	
43	2.5.5.3	Exercise 13 N1 G54 N2 T7 F200 S3200 M13 N3 G72 ZA-8 LP55 BP25 D2 V1 RN4 N4 G79 X25 Y45 Z0 AR110 N5 G72 ZA-5 LP45 BP30 D2.5 V1 AK0.2 AL0.4 H14 N6 G79 X85 Y45 Z0 W1 N7 G74 ZA-17 LP45 BP9 D2 V1 W1 EP0 N8 G79 X25 Y45 Z-8 AR110 N9 G73 ZA-9 R12.5 D2 V1 N10 G79 X85 Y45 Z-5 W1 N11 G0 Z100 M9 N12 T1 F120 S800 M13 N13 G81 ZA-4.5 V1 N14 G76 X60 Y70 Z0 AS0 D50 O2 N15 G76 X60 Y20 Z0 AS0 D50 O2 N16 G0 Z100 M9 N17 T10 F200 S1200 M13 N18 G81 ZA-19 V1 N19 G76 X60 Y70 Z0 AS0 D50 O2 N20 G76 X60 Y20 Z0 AS0 D50 O2 N21 G0 X150 Y150 Z100 N22 M30	
44	2.5.6.1	Exercise 14 N1 G54 N2 T2 F480 S1000 M13 N3 G0 X135 Y-2.5 Z-5 N4 G1 X10 N5 G2 X-2.5 Y10 I0 J12.5 N6 G1 Y80 N7 G2 X10 Y92.5 I12.5 J0 N8 G1 X110 N9 G2 X122.5 Y80 I0 J-12.5 N10 G1 Y59 N11 G2 X110 Y46.5 I-12.5 J0 N12 G1 X80 N13 G3 Y43.5 I0 J-1.5 N14 G1 X110	

44	2.5.6.1	N15 G2 X122.5 Y31 I0 J-12.5 N16 G1 Y-14 N17 G0 X150 Y150 Z100 N18 M30	
44	2.5.6.1	Exercise 15 N1 G54 N2 T3 F480 S1200 M13 N3 G0 X132 Y0 Z-5 N4 G1 X10 N5 G2 X0 Y10 I0 J10 N6 G1 Y80 N7 G2 X10 Y90 I10 J0 N8 G1 X110 N9 G2 X120 Y80 I0 J-10 N10 G1 Y59 N11 G2 X110 Y49 I-10 J0 N12 G1 X80 N13 G3 Y41 I0 J-4 N14 G1 X110 N15 G2 X120 Y31 I0 J-10 N16 G1 Y-12 N17 G0 X150 Y150 Z100 N18 M30	
44	2.5.6.1	..., a new equidistant line must be created for each milling cutter Ø.	
47	2.5.7.2	Exercise 16 N1 G54 N2 T2 F480 S1000 M13 N3 G0 X135 Y-5 Z-5 N4 G41 N5 G1 X110 Y10 N6 X10 N7 Y80 N8 X110 N9 Y59 N10 X80 N11 G3 Y31 I0 J-14 N12 G1 X110 N13 Y-15 N14 G40 N15 G0 X150 Y150 Z100 N16 M30	
47	2.5.7.2	We recommend to adapt the approach and retract point to the smaller Ø in addition to the T-number.	
48	2.5.7.3	N1 G54 N2 T2 F480 S1000 M13 N3 G0 X135 Y-5 Z1 N4 G41 G45 D25 X110 Y10 Z-5 N5 G1 X24 N6 G2 X10 Y24 R14 N7 G1 Y66 N8 G2 X24 Y80 R14 N9 G1 X110 N10 G1 Y59 N11 G1 X80 N12 G3 Y31 I0 J-14 N13 G1 X110 N14 G1 Y10 N15 G46 G40 D25 N16 G0 X150 Y150 Z100 N17 M30	

48	2.5.7.3	The approach and retract functions result in a higher programming comfort.	
49	2.5.7.4	Exercise 18 N1 G54 N2 T2 F480 S1000 M13 N3 G0 X40 Y110 Z1 N4 G41 G47 R10 X50 Y85 Z-5 N5 G1 X105 N6 G2 X115 Y75 I0 J-10 N7 G1 Y20 N8 G2 X100 Y5 I-15 J0 N9 G1 X20 N10 G3 X5 Y20 I-15 J0 N11 G1 Y45 N12 G1 X20 Y85 N13 G1 X50 N14 G48 G40 R10 Z1 N15 G0 X150 Y150 Z100 N16 M30	
52	2.5.9.2	Exercise 19 N1 G54 N2 T2 F480 S1000 M13 N3 G0 X45 Y-20 Z1 N4 G41 G47 R8 X40 Y5 Z-5 N5 G1 X5 RN-5 N6 Y50 N7 Y85 AS60 RN5 N8 X50 RN2 N9 G3 X75 Y80 R-16 RN2 N10 G1 X105 N11 X115 AS-30 N12 G1 Y35 RN2 N13 G3 X100 Y20 R15 N14 G1 Y15 N15 X62 Y5 N16 X40 N17 G48 G40 R8 N18 G0 X150 Y150 Z100 N19 M30	
52	2.5.9.2	Exercise 20 N1 G54 N2 T2 F480 S1000 M13 N3 G0 X45 Y-20 Z1 N4 G42 G47 R15 X40 Y5 Z-5 N5 G1 X62 N6 X100 Y15 N7 Y20 N8 G2 X115 Y35 R15 RN2 N9 G1 Y74.226 N10 X105 Y80 N11 X75 RN2 N12 G2 X50 Y85 R-16 RN2 N13 G1 X25.207 RN5 N14 X5 AS-120 N15 Y5 RN-5 N16 X40 N17 G48 G40 R8 N18 G0 X150 Y150 Z100 N19 M30	

53	2.5.9.3	Exercise 21 N1 G54 N2 T2 F480 S1000 M13 N3 G0 X-15 Y25 Z1 N4 G41 G47 R8 X5 Y30 Z-5 N5 G1 Y60 N6 Y85 AS80 RN3 N7 X15 N8 G3 X50 R22 RN3 N9 G1 X60 RN3 N10 G3 X95 R18 N11 G1 X110 RN-3 N12 Y65 RN2 N13 G3 X115 IA110 JA45 RN2 N14 G1 Y5 RN-3 N15 X100 N16 G3 X65 R20 N17 G1 X45 RN3 N18 G3 X20 AO270 RN3 N19 G1 X5 N20 Y30 N21 G48 G40 R8 N22 G0 X150 Y150 Z100 N23 M30	
53	2.5.9.3	Exercise 22 N1 G54 N2 T2 F480 S1000 M13 N3 G0 X-15 Y25 Z1 N4 G42 G47 R8 X5 Y30 Z-5 N5 G1 Y5 N6 X20 RN3 N7 G2 X45 AO270 RN3 N8 G1 X65 N9 G2 X100 R20 N10 G1 X115 RN-3 N11 Y25.635 RN2 N12 G2 X110 Y65 IA110 JA45 RN2 N13 G1 Y85 RN-3 N14 X95 N15 G2 X60 R18 RN3 N16 G1 X50 RN3 N17 G2 X15 R22 N18 G1 X9.408 RN3 N19 X5 Y60 N20 Y30 N21 G48 G40 R8 N22 G0 X150 Y150 Z100 N23 M30	
54	2.5.10	Bracket: [60]°	
55	2.5.10	Length: [11,538] mm Spacing: [109.983] mm Spacing: [32.661] mm Station: [2]	
56	2.5.11	Tool radius: [12.5] mm "Allowance" by TR: [1] mm Current tool radius: [13.5] mm	
57	2.5.12	Value for TR: [-0.01] Cutting depth in Z: Z[-2]	

58	2.5.13	Tool length: [80] mm "Allowance" by TL: [0.5] mm Current tool length: [80.5] mm	
59	2.5.14	Exercise 23 N1 G54 N2 T15 F400 S1500 M13 N3 G0 X-2 Y-12 Z1 N4 Z-5 N5 G41 G45 D16 X10 Y5 N6 G1 Y35 N7 G3 X10 Y60 R20 N8 G1 Y90 RN6 N9 X25 N10 X95 AS-8 RN12 N11 X115 Y95 N12 G2 X145 Y65 R30 N13 G1 Y35 N14 G2 X115 Y5 R30 N15 G1 X10 Y5 N16 G46 G40 D21 N17 G0 Z1 N18 G72 ZI-8 LP60 BP30 D3 V1 RN10 E200 F300 N19 G79 X50 Y65 Z0 AR-8 N20 G79 X50 Y30 Z0 AR-8 N21 G0 Z100 M9 N22 T10 F200 S1200 M13 N23 G81 ZI-15 V1 N24 G77 IA115 JA65 Z0 R20 AN0 AP100 O4 N25 G0 Z100 M9 N26 T17 F200 S3500 M13 N27 G75 ZI-9 BP8 RP20 AN0 AO90 D3 V1 EP1 E75 N28 G77 IA115 JA35 Z0 R20 AN-90 AI120 O3 N29 G0 Z100 M9 N30 T1 F120 S800 M13 TR1 N31 G0 X0 Y0 Z1 N32 G0 Z-2 N33 G23 N5 N16 N34 G0 X150 Y150 Z100 N35 M30	
60	2.5.15	1234: Invoke subroutine 1234 2: Repeat twice	
61	2.5.15	Exercise 24 Main program: N1 G54 N2 T11 F225 S2000 M13 N3 G0 X30 Y15 Z1 N4 G22 L73 H1 N5 G0 X50 Y50 Z1 N6 G22 L73 H1 N7 G0 X80 Y50 Z1 N8 G22 L73 H1 N9 G0 X110 Y30 Z1 N10 G22 L73 H1 N11 G0 X150 Y150 Z100 N12 M30	

61	2.5.15	Subroutine: N1 G91 N2 G1 Z-5 F150 N3 G1 X-15 F300 N4 G1 Y30 N5 G1 Z-4 N6 G1 Y-30 N7 G1 X15 N8 G0 Z9 N9 G90 N10 M17	
64	2.5.17	Exercise 25 N1 G55 N2 T8 F200 S2400 M13 N3 G0 X60 Y10 Z1 N4 G41 G45 D10 X50 Y8 Z-5 N5 G1 XI-10 N6 G3 YI-16 AO180 N7 G1 XI10 N8 G46 G40 D10 N9 G59 AR120 N10 G23 N3 N8 N11 G59 AR120 N12 G23 N3 N8 N13 G50 N14 G73 ZI-5 R30 D5 V1 N15 G79 X0 Y0 Z0 N16 G73 ZI-5 R30 D5 V1 W1 RZ18 N17 G79 X0 Y0 Z-5 N18 G75 ZI-21 BP12 RP40 AN30 AO60 D5.25 V1 EP1 N19 G77 IA0 JA0 Z0 R40 AN30 AI120 O3 N20 G0 Z100 M9 N21 T10 F150 S2000 M13 N22 G81 ZA-23 V1 W1 N23 G77 IA0 JA0 ZA-5 R40 AN0 AI120 O3 H2 N24 G81 ZA-23 V1 W1 N25 G77 IA0 JA0 ZA-10 R24 AN30 AI60 O6 H3 FP1000 N26 G0 X150 Y150 Z100 N27 M30	
65	2.5.17	Exercise 26 N1 G54 N2 T2 F700 S2000 M13 N3 G72 ZA0 LP102 BP72 D5 V1 H2 N4 G79 X-25 Y0 Z10 N5 F400 S2000 M13 N6 G0 X-90 Y0 Z1 N7 G0 Z1 N8 G1 Z-10 N9 G1 X-25 N10 G2 X-10 Y15 R15 N11 G1 XI5 N12 G1 YI-5 N13 G3 X5 Y0 R10 N14 G3 X-5 Y-10 R10 N15 G1 YI-5 N16 G1 XI-5 N17 G2 X-25 Y0 R15 N18 G0 Z1 N19 G0 X-90 N20 G0 Z-20 N21 G1 X-35 N22 G0 Z1 N23 G0 Z100 M9	

65	2.5.17	N24 T8 F250 S2300 M13 N25 G41 G45 D20 X-75 Y-20 Z-10 W1 N26 G61 AS0 N27 G61 XA-45 YA-16 AS30 N28 G61 YI0 N29 G63 XA-10 IA-10 JA0 R30 N30 G1 X10 N31 G61 XI0 N32 G63 XA10 IA0 JA0 R20 N33 G1 Y30 N34 G1 X-10 N35 G3 Y16 IA-10 JA0 N36 G1 X-45 N37 G1 Y20 AS150 N38 G1 X-75 N39 G46 G40 D11 N40 G41 G45 D11 X-75 Y-16 Z-20 W-20 N41 G61 YI0 N42 G62 YA16 IA0 JA0 R20 N43 G1 X-75 N44 G46 G40 D11 W1 N45 G0 Z100 M9 N46 F220 S1600 T1 M13 N47 G81 ZA-2.5 V1 N48 G79 X-40 Y25 Z0 N49 G79 YI-50 N50 G0 Z100 M9 N51 F250 S5000 T35 M3 N52 G82 ZA-15 D10 V1 N53 G23 N48 N49 N54 G0 Z100 M9 N55 T4 F250 S2300 M13 N56 G87 ZA-31 R11.25 D4 V1 N57 G79 X0 Y0 Z-10 N58 G0 Z100 M9 N59 T22 F250 S2300 M13 N60 G88 ZA-31.5 DN24 D1.5 Q20 V3 N61 G23 N57 N57 N62 G0 X150 Y150 Z100 N63 T0 M30	
69	2.5.19	The solution is saved in the software under the "Teacher" operating mode.	
70	2.5.19	The solution is saved in the software under the "Teacher" operating mode.	
71	2.5.19	The solution is saved in the software under the "Teacher" operating mode.	

Keyboard assignment of the SYM*plus*

System

(3D-)Additional function selection	<F11>
Configuration selection	<Ctrl>+<F12>
Operating mode selection	<Ctrl>+<F10>
Direct operating mode selection	<Ctrl>+<Fx> (x = 1...9)
Changing to the next operating mode	<Ctrl>+<Tab>
Changing to the previous operating mode	<Ctrl>+<Shift>+<Tab>
Window edge display	<Alt>+<Pos1>
Software exit	<Alt>+<F4>
Switching dialogs transparent <Alt>+<F9>	
Help system	<F12>
Scrolling through help masks	<Ctrl>+<LH arrow key> or <RH arrow key>
Changing a value in the input field	<F9>
Confirms dialogs	<F10>
Abort dialogs/inputs	<ESC>
Display selection field options	<F9>
Next option in the selection field	<+>
Previous option in the selection field	<–>
Application of entries in the input field	<Enter> or <Tab>
Next entry or selection field	<Tab>
Previous entry or selection field	<Shift>+<Tab>

Direct additional function selection

(Only with the respective configuration)

Zoom	<Alt>+<1>
General view	<Alt>+<2>
Machining compartment view	<Alt>+<3>
Calculator	<Alt>+<4>
Point determination	<Alt>+<5>
Measuring	<Alt>+<6>
Element information	<Alt>+<7>
Screen print	<Alt>+<8>
Adjusting dimensions	<Alt>+<9>

Work step handwheel

Switch on "Handwheel keyboard"	<Shift>+<F4>
Increasing the hand wheel increment	<Shift key>+<F5>
Reducing the hand wheel increment	<Shift key>+<F6>

Use the handwheel to move in the positive direction<Alt>+<Scroll down>
Use the handwheel to move in the negative direction<Alt>+<Scroll up>

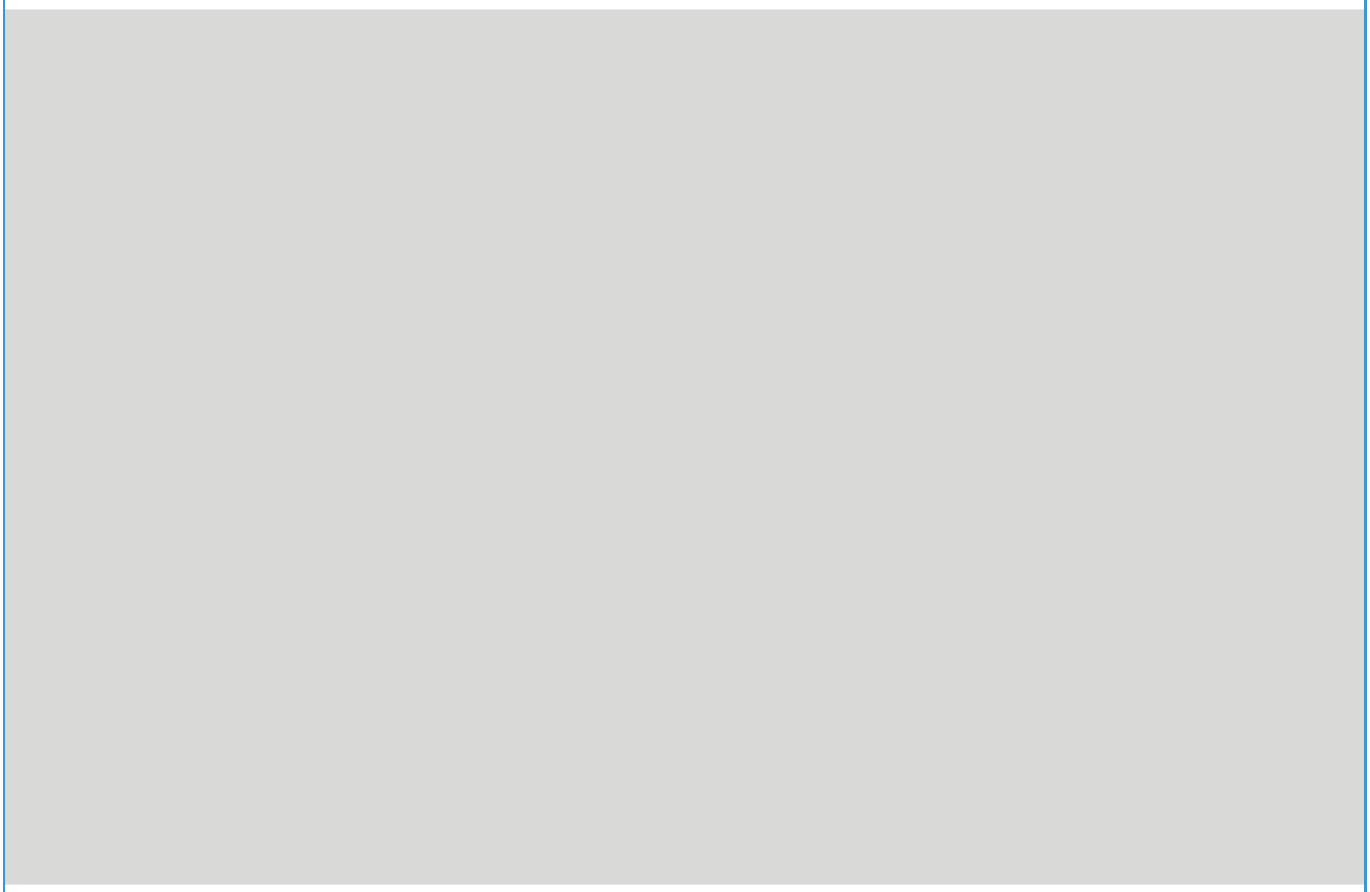
Navigation

Start of page / list	<Pos1>
End of page / list	<End>
Select option / contour...	<+> or <–>
Cursor left / right	<LH arrow key> or <RH arrow key>
Cursor up / down	<Arrow key up> or <Arrow key down>

Simulation

Increase / reduce feed override	<+> or <–>

Remark:
The '+' sign between the keys (<Key1>+<Key2>) indicates
that all specified keys have to be pressed simultaneously.



www.cnc-keller.com

CNC KELLER GmbH

Vorm Eichholz 2, 42119 Wuppertal, Germany · Fon: +49 202 40 40 0 · Fax: +49 202 40 40 99