HEIDENHAIN

Klartext

Klartext – The TNC Newsletter · Edition 42 · 6/2004 METAV Edition

start smart.

smarT.NC, the New User Interface

Interactive Preparation for an iTNC 530 Basic Course

Editorial

Dear Klartext Reader,

This issue of Klartext is dedicated to smarT.NC, our new user interface for the iTNC, which enables beginners as well as experienced plain language programmers to create executable NC programs faster, more safely, and more conveniently.
But what makes smarT.NC so special is that it does not mean that we've abandoned the concept of plain language programming. Instead, we've made it even more user friendly. This means that you decide whether to create and run a program in the plain language level or in the new user interface. The machining steps are defined in machining units and listed in the working plan. The benefit is both unique and striking: even if a TNC program has been created with smarT.NC, it can be edited as before with the TNC (plain language) editor. You can even insert "genuine" plain language blocks between individual smarT.NC machining blocks. This procedure is possible because both smarT.NC and the plain language editor access one and the same file: the plain language program!

Are you familiar with the iTNC programming station? If you'd like to know how it can be a useful addition to your existing machinery and dramatically increase your productivity, read this issue's article on the Wintec Company.

Have a good read.

Content

Imprint

Publisher
DR. JOHANNES HEIDENHAIN GmbH
Postfach 1260
D-83292 Traunreut
Tel: (0 86 69) 31-0
HEIDENHAIN im Internet:
www.heidenhain.de

Editor
Frank Muthmann
Fax: (0 86 69) 31-18 88
e-mail: info@heidenhain.de

Klartext on the Internet
www.heidenhain.de/english/print_0.htm

Layout
Expert Communication GmbH
Inselkammerstraße 5
82008 Unterhaching/München
Tel: (0 89) 66 63 75 0
e-mail: info@expert-communication.de
www.expert-communication.de

Plain Language Gets Even Simpler with smarT.NC, the New User Interface for the iTNC 530

Introduced more than 25 years ago, plain-language is still the standard programming language for all TNC controls. When you hear someone talking about shop-floor programming, he very often means plain-language conversational programming from HEIDENHAIN. Thanks to HEIDENHAIN's consistent development efforts over many years, conversational programs written on earlier TNC contouring controls can also be run on the newest HEIDENHAIN control, the iTNC 530.

But the people at HEIDENHAIN wanted more than this. In addition to their classic conversational programming format, they have now developed a new user interface oriented as much to the NC beginner as to the seasoned conversational programming professional. As an intuitive, self-explaining interface, smarT.NC—as the new user interface is called—guides the machinist through the complete NC programming process. So far, so good. But the key point is that smarT.NC can be used as an alternative to conversational programming. This means that smarT.NC lets the machinist decide whether he wants to create and run a program in the conversational level or in the new user interface. In other words, all previous TNC programming features remain as available and functional as they've ever been.

With smarT.NC, HEIDENHAIN is abiding by its principle: the machinist need only learn those features that are really new—he does not have to relearn features that he has already mastered.

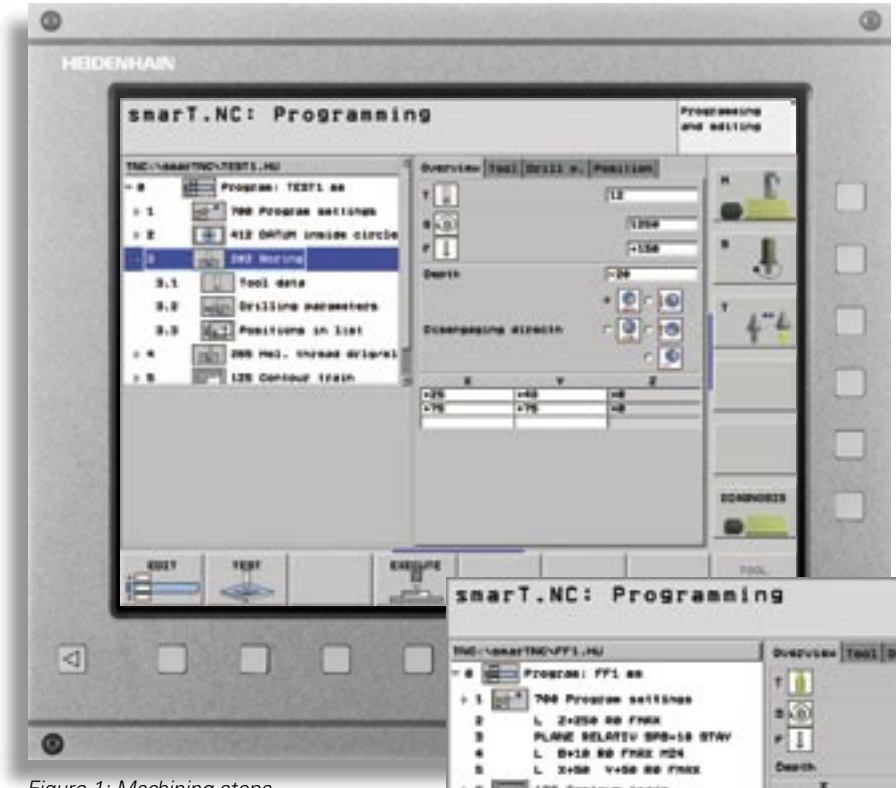


Figure 1: Machining steps are displayed in the form of units

What seems so simple to the user is the result of a very sophisticated program structure. While the user follows the logical dialog guidance, smarT.NC simultaneously saves the data in structured plain-language programs. The machining steps defined in machining units and listed in the working plan. (Figure 1)

The benefit to the machinist becomes evident: even if a TNC program has been created with smarT.NC, it can be edited as before with the TNC (plain language) editor. He can even insert "genuine" plain language blocks between individual smarT.NC machining blocks. Of course HEIDENHAIN leaves it to the machinist

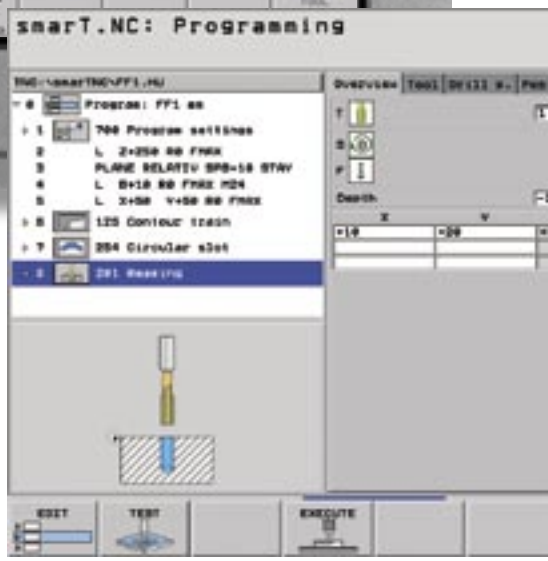
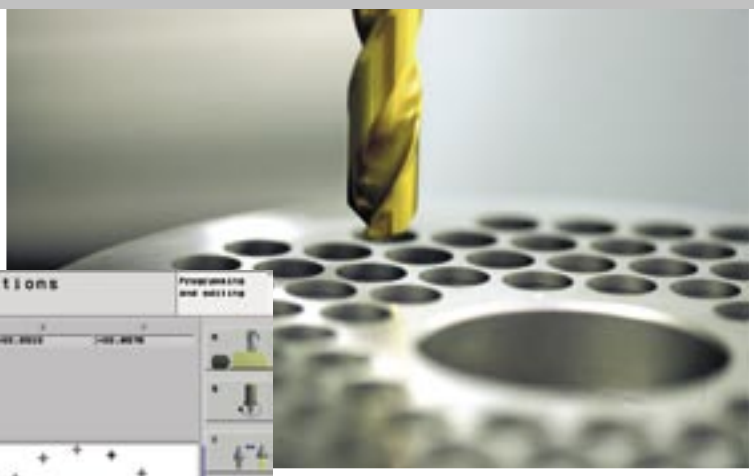


Figure 2: Plain language blocks between smarT.NC machining units

to decide whether to edit in the conversational editor or in smarT.NC. This procedure is possible because both smarT.NC and the conversational editor access one and the same file: the plain language program! (Figure 2)

smarT.NC



But smarT.NC offers more than just an alternative, albeit simpler, type of programming. Many new features were integrated—for example the pattern generator. This geometry function has something special about it. Machining positions that are definable by regular patterns are especially easy to program with smarT.NC. The new user interface provides a comprehensive selection of patterns. (Figure 3).

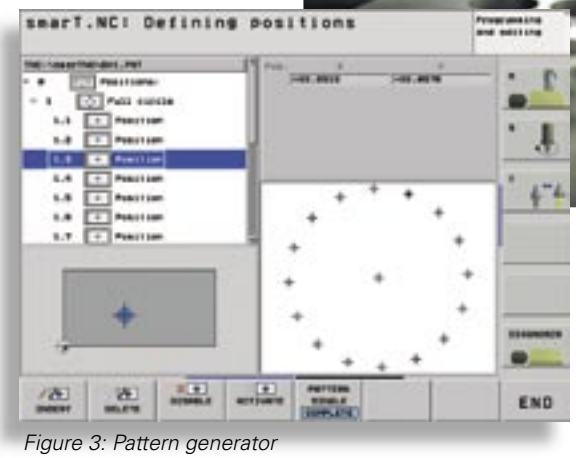


Figure 3: Pattern generator

This, too, is a special feature of smarT.NC: if certain predefined machining positions within a machining step are not required, they can be simply skipped during program and test run, or even deleted from the program.

Several defined patterns can be combined with each other so that, for example, patterns can be superposed on circle segments or circles with linear patterns. The individual patterns can be assigned to and machined at various heights.

Figure 4: Overview form

Figure 5: Subform with additional machining options

Key Features of smarT.NC

- **Fast programming of simple operations**

  The simpler the operation, the fewer entries are required. With smarT.NC, the machinist defines all required machining steps in a single-page form.(Figure 4). The preconfigured, globally effective machining parameters such as safety clearance, etc. guarantee that the TNC adheres to these global parameters with each selected machining step.

- **Fast programming of complex operations**

  If additional machining options become necessary, smarT.NC automatically offers an appropriate subform. (Figure 5)

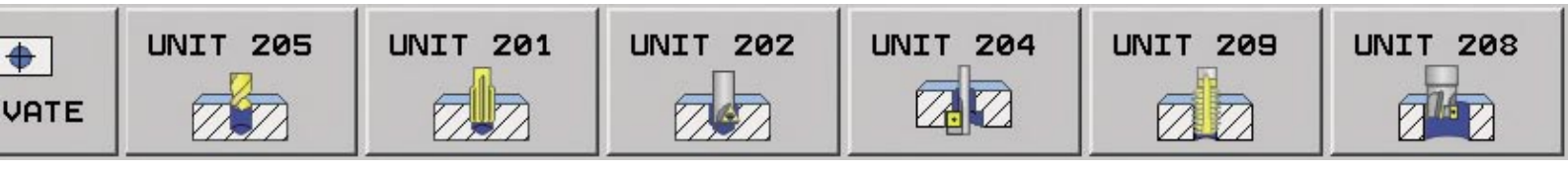
- **Expanded functions**, such as measuring cycles or multi-plane machining in conjunction with a tilted working, are provided in smarT.NC by appropriate "machining units" plane.

- **smarT.NC generates plain language format**

  Plain language programs from smarT.NC can be edited both in the smarT.NC user interface and in the conversational mode. Any function available in the TNC plain language format can be inserted between the machining units. (Figure 2)

Key to start smarT.NC

Key to page through input forms



- Clear depiction of the program structure
  - smarT.NC's split screen enables fast navigation within the strucktured working plan in the left window. Opening and closing various levels shows the well ordered machining steps. (Figure 6)
  - Clearly arranged input forms on the right side of the screen immediately show the defined machining parameters.
  - The soft-key row shows the input options most useful for the active input field and lets you find more by soft key. (Figure 7)
- Fewer input parameters
  - Global program parameters such as setup clearances, positioning feed rates, etc. obviate multiple definitions and remain effective until they are redefined in the program. (Figure 8)
  - smarT.NC shows only those input parameters relevant to the scope of operation selected by soft key.

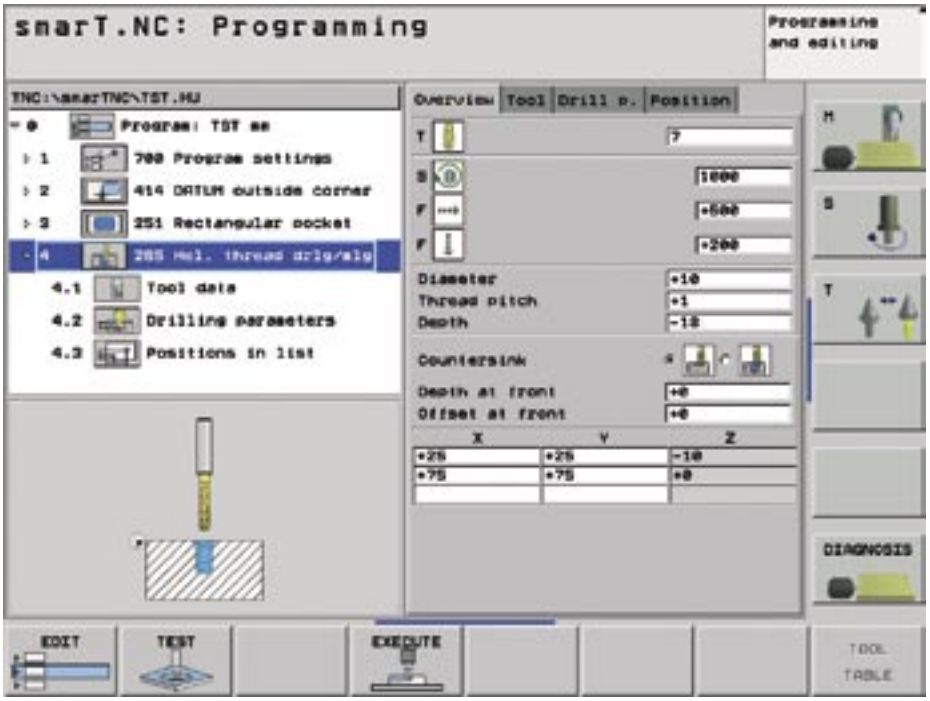


Figure 6: Work plan shown in outline

- Fast editing of input parameters

smarT.NC allows you to change any machining parameters within the input form with only a few keystrokes. The forms are divided into different sections delineated by frames.

Figure 7: Switching between input options by soft key

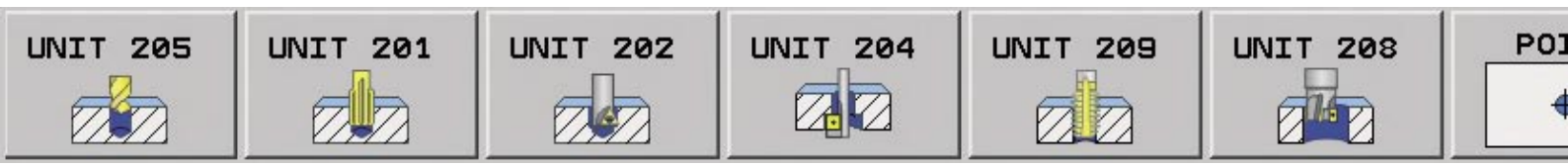
Figure 8: Global data in the program header

The navigation keys bring you to the first entry of the respective following or preceding sections:


Jump to the first entry of the next section


Jump to the first entry of the previous section


Switch to the next form view



- **Stay fast and flexible when programming machining positions**

  Every machining position is graphically illustrated in the machining pattern. (Figure 9)

  - Only a few entries are needed to define machining patterns of single points, single rows, surfaces, frames, full circles, and circle segments.
  - Immediately upon definition, the machining pattern is displayed with respect to the datum. A zoom function is also available.
  - Any number of different point patterns containing any number of points can be defined in one file. Just as in the part program, the point patterns are shown in an outline.

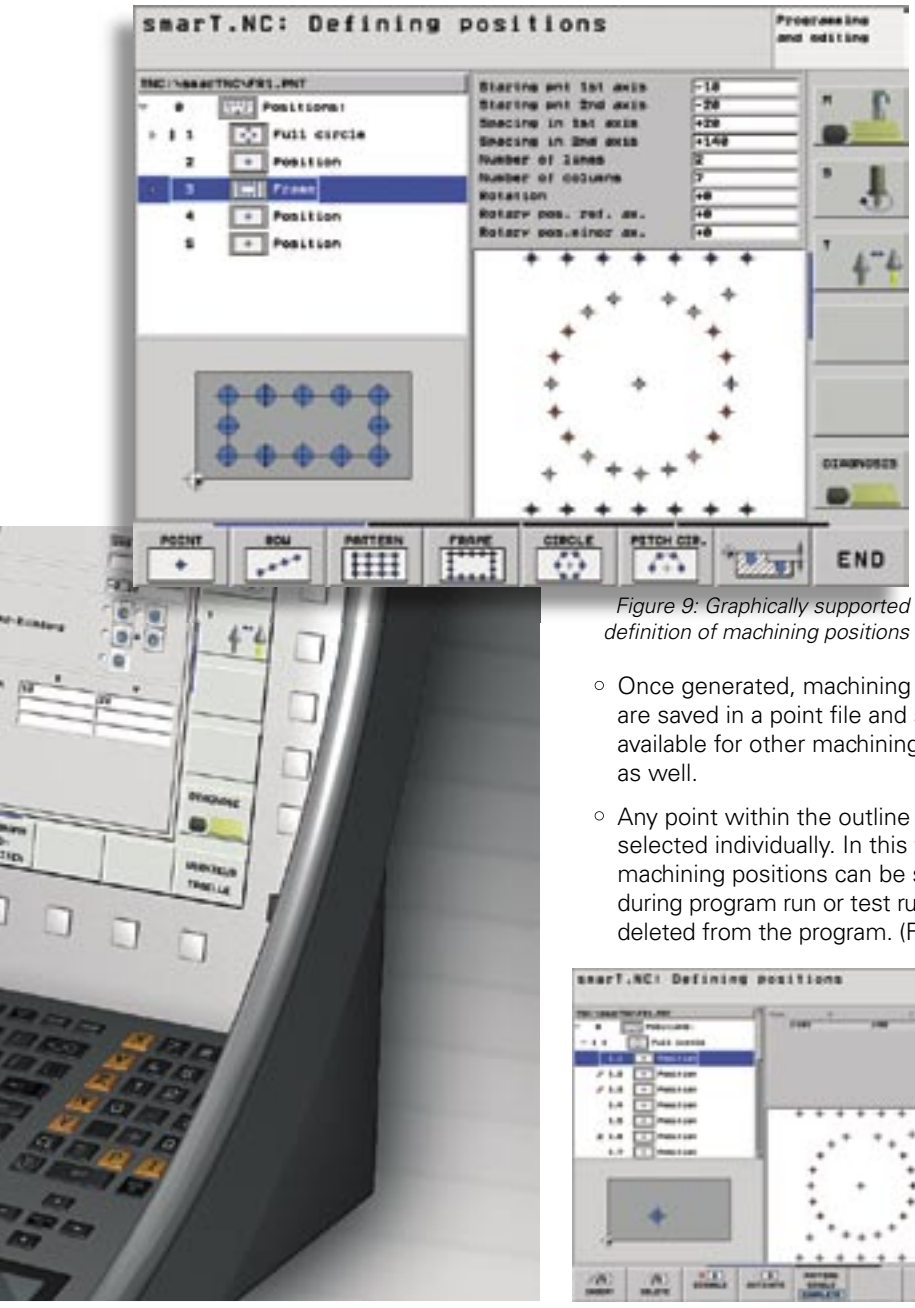


Figure 9: Graphically supported definition of machining positions

  - Once generated, machining positions are saved in a point file and stay available for other machining steps as well.
  - Any point within the outline can be selected individually. In this way the machining positions can be skipped during program run or test run, or deleted from the program. (Figure 10)

Figure 10: Skipping individual machining positions

  This is particularly helpful when individual positions in a regular pattern are to be left unmachined.
  - Even the coordinates of the workpiece surface can be changed as desired between the individually defined machining patterns.
  - The machinist selects by soft key whether all defined point patterns are displayed or only the currently active one.
  - Every single machining position calculated by the TNC can be read directly in Cartesian coordinates.

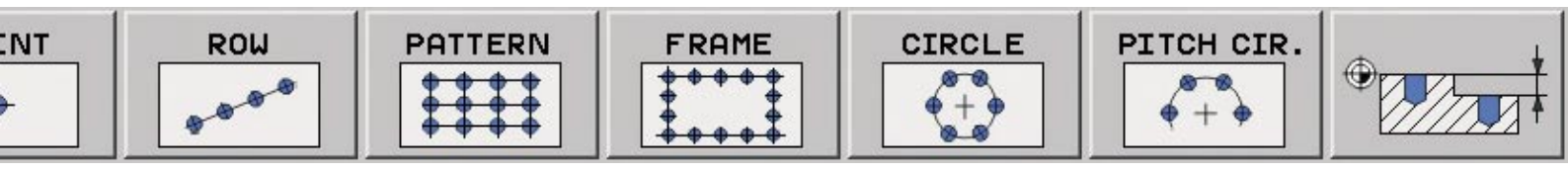
- **Stay fast and flexible when programming contours**

  The definition of contours is just as self-explanatory as the creation of part programs through graphically aided forms.

  The individual contour elements are likewise displayed in the outline; the associated data are shown in a form. The TNC saves the contour itself as a plain language program in a separate file. Since these contour descriptions do not include tool radius compensation, they can be individually reused later for various operations. The programming graphic immediately displays the contour element as it was entered. (Figure 11)

  If a workpiece is not dimensioned for conventional NC programming, smarT.NC also features the powerful HEIDENHAIN FK free contour programming. Vivid help graphics aid the machinist in finding which of the many entry options he needs.

Figure 11: Contour programming supported by help graphics



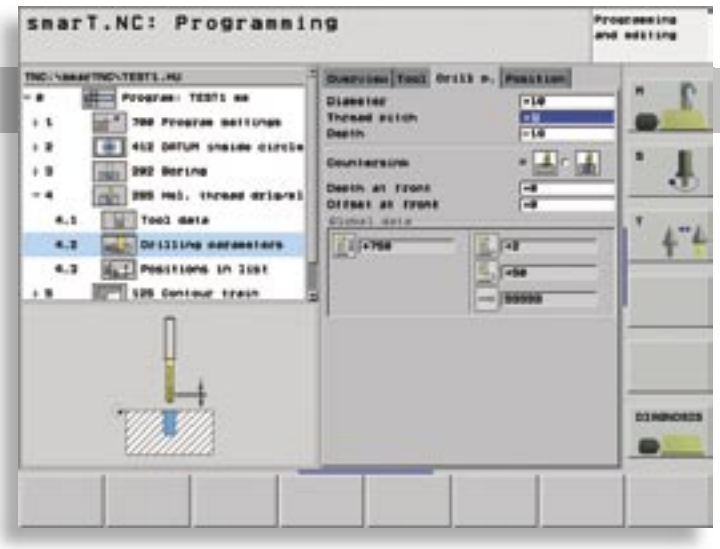


Figure 12: Detailed help graphics

- **Perfect programming support**

  Beginners need to learn quickly how to write programs, and without any extensive training. To accelerate learning, smarT.NC supports the user with:

  - Clearly arranged help graphics that illustrate all required input (Figure 12)
  - Graphic, easily understood symbols in the forms so that types of input information are quickly recognized for different operations (Figure 13)
  - Explanatory tips that appear on contact with the mouser pointer (Figure 14)

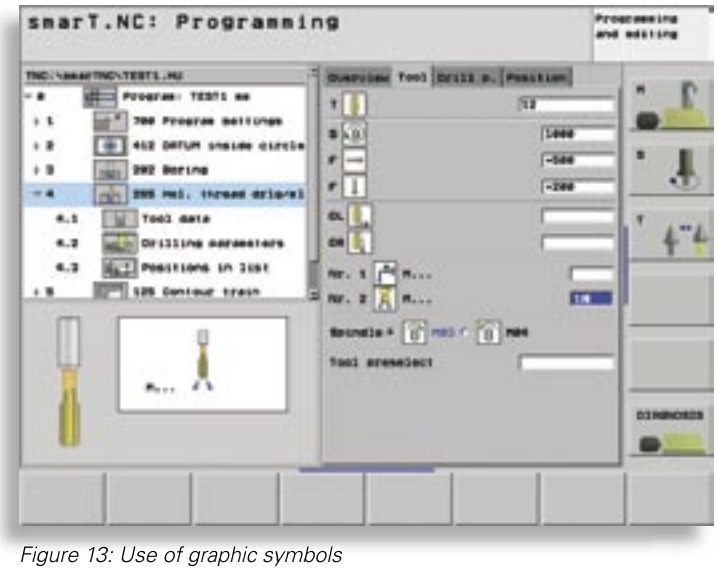


Figure 13: Use of graphic symbols

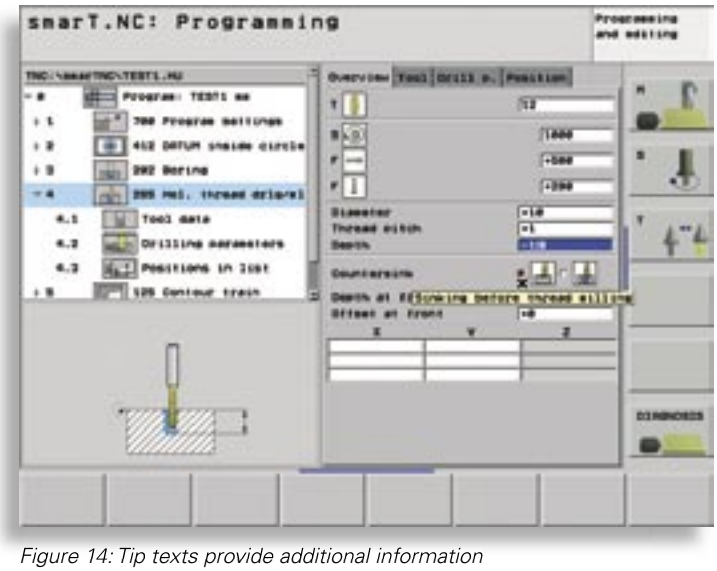


Figure 14: Tip texts provide additional information

- **Program, test, and run without changing modes**

  From creation of an individual program, to the test run, and even including the execution of a complex program—with smarT.NC you can complete all of your programming work centrally in one operating mode.

  Of course the test run also includes the new 3-D graphics, which can display operations defined in a tilted working plane.

  The following modes are available for running a part program:

  - In the program run mode, defined machining units are executed either individually in succession or all together from beginning to end.
  - In the flexible single-block mode, any machining unit can be selected individually by cursor and run without necessarily following a certain sequence

Goals

In the medium term it is planned to make nearly all available plain language functions accessible through forms in smarT.NC.

2003 CNC Machining World Champion Visits HEIDENHAIN

Every two years, young professionals from around the world, accompanied by their teachers and trainers, take part in a public competition where they measure their skills against their peers in the various occupations, while at the same time maintaining high international standards.

At the 37th World Skills Competition in St. Gallen, Switzerland naturally had home advantage, and secured victory with a total of 20 medals. HEIDENHAIN took special note of this event when the media mentioned that the machinists had performed the CNC portion of the competition on machine tools equipped with HEIDENHAIN TNC controls. Beat Güller, a 20-year-old from Würenlos, became World Champion by besting the other 18 participants in the 4-day competition.

Beat Güller acquired his CNC machining skills in the training centers near Oerlikon and in his father Leo's factory in Würenlos. The small but mighty company, founded in 1960, has five employees. The product spectrum includes the CNC machining of single parts and series, vacuum filling of transformers, manufacture of foil capacitors, and development of specialized solutions in cooperation with customers.

Beat Güller also works on a machine tool equipped with a HEIDENHAIN TNC in his father's factory. Asked how he managed to become the best CNC machinist in the world, he modestly answered, "It was an advantage that the World Skills Competition took place in Switzerland, and also that the milling machine itself was from Switzerland. That way I had to chance to practice at the Oerlikon training center for three months." What he didn't mention is that this alone is not enough to become a world champion in milling, but that you also have to be an excellent milling machinist.

However, his start into the competition couldn't have been worse. In the first exercise he noticed after already half of the machining time had past that a datum had been set incorrectly, so he had to start over completely. Everything went smoothly from there, and Beat Güller produced the best results. The new workpiece was manufactured in record time, and he did not exceed the total time permitted. Naturally five points were deducted because of having to start over, but he still achieved 93 out of the 100 points possible for this workpiece. A total of four "fantasy workpieces" had to be programmed and milled. Beat Güller almost received the maximum number of points possible for the fourth workpiece, scoring 99.41 points.

HEIDENHAIN wanted to congratulate Mr. Güller on his achievement, and so invited him and his girlfriend to come visit Traunreut. But this excellent performance actually deserved something more, and so the two of them enjoyed a very relaxing wellness weekend before heading back home to Switzerland.

We wish Beat Güller much continued success in his career.

The idea of holding a competition for young professionals originated in Spain in 1950. At the beginning it was a purely Iberian showdown between twelve-man teams from Spain and Portugal. Diplomatic observers from several European countries were present, and they became very enthusiastic about the idea. This led to the founding of the International Vocational Training Organization (IVTO) and the international job skills competition. Since 2001 the IVTO has been known as the World Skills (WS) association, and the vocational world championships are now the Wold Skills Competition (WSC).

Beat Güller's visit to JH

Workpiece from the third day of competition: CK45 (steel), 2.5 hours for programming + 3.5 hours for machining; total: 6 hours. Note the three-sided machining

Drilling Really Deep Holes.

When very deep holes must be drilled — where the tool is at least 20 times as long as it is wide — then neither normal tools nor standard cycles can be used. The techniques are so sophisticated that some companies have specialized themselves in only this type of machining. For example, they drill 1-m deep coolant holes with a diameter of 10 mm into injection molding tools.

Tools

Sets of extra-long drills, where each drill is "responsible" for a certain depth range, are used for these machining operations.

Production process

First you center, and then make a pilot drill hole, which can be performed with the usual tools and standard cycles.

The drill used for the actual deep drilling is so long that its length makes it too unstable to be able to use it directly on the surface. In order to be used, the tool must be guided via a pilot hole at least as deep as one-third the length of the tool.

The second tool can then increase the depth, for example, from 100 mm to 300 mm. Another tool is then necessary for each subsequent 300 mm section, i.e., for 300 mm to 600 mm, and from 600 mm to 900 mm. A longer, unguided length, on the other hand, would lead to tool breakage.

The exchange of such long tools can no longer take place automatically. Therefore, if a number of holes must be drilled to varying depths, then the production process must be tool-oriented, meaning that as few tool-changing procedures as possible should be programmed.

What is required:

1. A special drilling procedure
2. A tool-oriented production strategy that is oriented on the natural sequence of the tool set.

The "ingredients" used could be as follows on a current TNC:

1. A drilling cycle to be programmed with the help of Q parameters.
2. The declaration of this program as a cycle using Cycle Definition 12.
3. The use of a freely definable table containing all information about each drill hole.
4. A basic program that contains all calls and values to be loaded, and that ensures that each tool located in the spindle remains there until all machining steps for which it is suited have been performed.

Organizing the machining steps

> The free table is assigned.

> The centering tool is inserted.

> The values from the special drilling procedure are loaded and the program is declared as a cycle.
(Because of the simple program structure, the centering and pilot drilling are performed as part of the special procedure, even though they would be just as possible with the system cycles.)
The values from the table are read row-by-row, but only from the columns X and Y. One centering drill hole is performed per line. Only the plane position in X/Y is assumed from the table. The drilling information comes from the cycle. This means that all drill holes are centered identically.

> The standard drill is inserted. A single cycle value (e.g., the drilling depth) is loaded. All plane positions from the table are again machined in the same manner (uniform 100-mm deep pilot holes).

> The first extra-long drill is inserted. Again the rows of the table are read, this time with additional information, especially about the depth and material surface.

Special attention must be paid to each depth:

If the 300-mm drill is in the spindle, it must

1. directly drill a hole from the pilot depth to 300 mm,
2. and plunge only 300 mm if the hole is deeper.

> The second extra-long drill (300 mm to 600 mm), as well as any longer drills, if required, are then inserted.

Using the 600-mm drill as an example, the following must be kept in mind:

1. Ignore depths less than 300 mm, because they are already finished.
2. Directly drill depths from 300 mm to 600 mm.
3. Plunge only 600 mm if the holes are deeper.
4. Drilling instructions that go deeper than the longest tool must be limited to that tool's maximum length.

Using this method, a drilling position only 250 mm deep is only approached by the first extra-long drill, whereas a drill hole that is 1100 mm deep is ap-

proached four times, once each time when the currently appropriate tool is in the spindle. This avoids unproductive motions and unnecessary tool changes.

If the table is structured correctly, then the information for each drilling position can be changed to suit the present needs. The operation remains clear and easily understandable, since all relevant values are stored in the table.

The drilling cycle

Parameters to be loaded

```
Q1 =+0 ;Material surface                Absolute
Q2 =+0 ;Pilot-drilling depth            Incremental
Q3 =-10 ;Chip-removal height            Incremental
Q4 =+1000 ;Feed rate for plunging
Q5 =+200 ;Feed rate for drilling
Q6 =+2000 ;Shaft speed
Q7 =+5 ;Infeed                          Incremental
Q8 =+0 ;;Depth
Q9 =+0.5 ;Dwell time
Q10 =+1 ;Chip-breaking value            Incremental
Q11 =+2 ;Safety value 1                 Incremental
Q12 =+30 ;Safety value 2                Incremental
Q13 =+5 ;Number of infeeds
```

Selection of the drilling positions from the table

```
Q51 =+0 ;Starting line
Q52 =+4 ;Subsequent drill holes
```

The drilling process

At the beginning of the cycle the control checks whether the drilling depth is deeper than the pilot depth, and whether it has been entered as a negative value. If one of these requirements is not fulfilled, the cycle is not performed.

> The first safety height is approached in the Z axis at rapid traverse.

> The tool rotates counterclockwise.

> The predrilled depth plus the safety height are approached at the feed rate for plunging.

> After the spindle rotation has been reversed, drilling is performed at that feed rate and with chip breaking until the programmed number of chip breaks has been reached.

> The tool rises at rapid traverse to the chip-removal height (remaining in the material), where it dwells briefly.

> New infeed to the reached depth plus safety height.

> Drilling in stages with chip breaking and chip removal.

> The tool reaches the final depth and dwells there.

> Reduction of the shaft speed and retraction at an increased rate to the second safety height.

> Then spindle stop.

You'll find our complete program under www.heidenhain.de/klartext

Brand new for participants of HEIDENHAIN seminars: An interactive training program helps to prepare you for the iTNC 530 Basic Course. This is fully in the spirit of "The transfer of knowledge must be kept simple, and is allowed to be fun!"

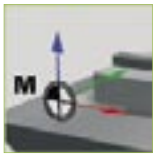
Interactive Preparation for an iTNC 530 Basic Course

People interested in HEIDENHAIN seminars have a new ace up their sleeves: They can use a new, interactive training program to acquire important **fundamental knowledge** before the course, and then test their knowledge without any pressure.

The interactive training program is intended to establish as uniform a basic knowledge as possible, so that all participants can begin the course as **smoothly as possible**. That is why each person who registers for one of the BAS 530 TNC basic courses receives an **iTNC Training** CD with their registration confirmation.

Contents

The training system is very simple to use: Each topic is selected with a simple mouse click. You are then led step by step through each training section. Quite a large spectrum of knowledge is covered, so the system limits itself to just the most important information on each topic. The topics are:



Coordinate Systems

The first section conveys fundamental knowledge about the most important reference points on the machine and workpiece. Animations show how to determine positions using a coordinate system, and how reference points on workpieces are determined and set.

NC Axes

Everyone who wants to know how machine tools control tool motions and determine their traverse paths learns important technical background information in this section.

Tools

The various types of tool systems on machines as well as the reference points of tools are discussed here.

The TNC

Under no circumstances should future course participants without TNC experience skip this section. This is where interactive functions teach you about the most important operating elements on the control and screen—essential for a smooth start in the basic course!

Programming Fundamentals

This is an important requirement for anybody who has never used HEIDENHAIN conversational programming before. Amusing animations explain the most important "building blocks" of an NC program, and show you how to create and manage machining programs with the TNC.

Frequently Used Functions

These functions are needed repeatedly during program creation. This section discusses the most important machine functions, approach and departure motions, and the handling of tool tables.

Interactive training must be simple, and should make you curious

The following example, using the approach and departure motions, is intended to show that using the interactive training system can be easy, even when the material becomes somewhat complex. After all, the user should be able to concentrate on what's important: the contents of the presentation.

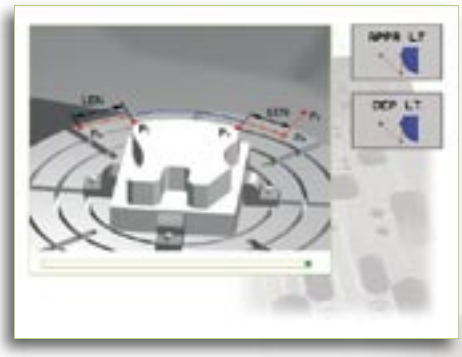
Simple interaction
One click and the film starts running: While the 3D animation is being shown, the function of the tool motion is described in synchronized texts.

Independent control of progress
Complex functions are broken up into smaller steps and explained simply. This helps you to gain a quick overview. In each case the user determines the speed of progress by clicking the mouse to start each subsequent step of the animation.

Many 3-D animations and interactive sequences transform the preparation into amusing edutainment.

Easy-to-remember sequences
After the function has been completely explained, the path traversed in this example remains on the screen as a graphic—making it easier for you to remember typical tool motions.

The 3-D animations are based on machining operations frequently used in practice.



Learning and testing on our own

With the HEIDENHAIN training system, the user determines which topics he would like to learn (more) about. The tests at the end of each section are also taken just as independently. Along with multiple choice tests, where you simply click the correct statements, there are also a series of tests where one must find the correct solution with the help of the animation.

Interactive test with animation

It is impossible to "get stuck" on any test. It is always possible to continue even if you don't get the correct answer immediately.

Interactive courses in the future?

Of course all courses will continue to take place in the HEIDENHAIN seminar rooms. But even now, given the elaborate technical equipment in the seminar rooms, the participants expect features like live video transmissions of workpiece machining processes on the machine, and—more and more often—interactive animations or simulations.

You will find information about the HEIDENHAIN courses on the Training Portal of the HEIDENHAIN website. You can quickly find the appropriate course, and determine if there is still space available: Go to *www.heidenhain.de* and click "Training Center."

Why have an interactive training system?

We are convinced that the use of interactive media increases the quality of our training seminars. They offer practical information, and the animations make it easy to convey complex material. Along with an independent preparation for the course, the medium can be used during the course to quickly convey difficult subjects to each participant, as well as to expand any current knowledge. We expect that this will improve the individual advancement of each participant, as well as save time. In the future we would also like to test the material covered in the course interactively, so that we may document the learning success. This gives us more time for individual attention to each participant, and makes room for the constantly increasing scope of the courses.

Hannes Wechselberger,
Director of HEIDENHAIN Training

WINTEC Präzisionsmechanik GmbH Opts for External Programming Station from HEIDENHAIN

Originally, oil used to be produced here, later superseded by a reservoir for natural gas. Meanwhile, an industrial park has been established on the site of the former oil field just outside Assling, a village near Munich in Bavaria. This is where Rudolf Winter, the founder, owner and managing director of ***WINTEC Präzisionsmechanik GmbH***, found the ideal premises for the company's new head office and workshop in 2001.

Founded in 1997, the company offers CNC milling and turning in the µm range, laser writing services, and the manufacture of prototypes and special machinery in quantities from one to several thousand. A further focus is the manufacture of components and small devices.

Within only a few years, high investments were made into machinery. The ultramodern workshop comprises five machining centers and two tool milling machines for CNC milling, two turning lathes for CNC turning with driven tools, and one laser marker. This amounts to a total of nine CNC-controlled machine tools, seven of which are equipped with TNCs.

Streamlined workflows and sophisticated logistics are the distinguishing features of the company. WINTEC is of course certified to DIN EN ISO 9001:2000. The on average twenty employees have been working in two shifts since 1998 and, since April 2004, even in 3 shifts.

WINTEC's customers are primarily German companies specialized in the development and manufacture of broadcast products for film and photo technology as well as in medical components and computer-chip handling systems, to name only a few.

Consistency in machine tool control

For Rudolf Winter, the machine tool control is the be-all and end-all. It comes as no surprise, then, that all the milling machines use controls from HEIDENHAIN.

HEIDENHAIN has been a supplier of contouring controls for milling and drilling operations for over 20 years. The basic operating concept of all TNC controls has always remained the same throughout this time. All TNC controls are compatible with the predecessor models in both operation and programming. This has the advantage that users are immediately familiar with new control versions, which greatly facilitates work with the machine tool.

At the same time, the know-how the WINTEC team has gathered at the TNC controls is further developed and extended more quickly. Learning how to program controls from other manufacturers would require additional investments in time and capital. Permanent changeovers in the handling are therefore avoided in the workshop.

Advantages of external programming

WINTEC implemented an external HEIDENHAIN programming station already quite a while ago. Here in the programming room, the staff can concentrate on specific solutions without being disturbed. The programmer checks the programs, simulates the workpiece machining, and verifies all technological parameters as well as the part geometry. Possible errors are thereby revealed early on, thus eliminating machine downtimes.

Mr. Rudolf Winter, owner and manager of WINTEC Präzisionsmechanik GmbH

All machines are networked online. With three TNC controls connected directly to the external programming station over the network, the programs can be transferred straight to the machine tools. While the programmer is entering or testing part programs at the external programming station, the machines in the workshop are free for other projects. This parallel procedure allows optimally utilizing the machine capacities.

In the area of CNC milling, the external programming station is additionally used for training purposes. Trainees and interns can conduct programming tests and run trial programs without taking up the time of colleagues or occupying machinery.

Fit for the future!

For Rudolf Winter, quality, short lead times and a fair price are the key criteria to remaining competitive. Nowadays, customers hardly ever accept costs for testing. With an external programming station, specific machining operations can be simulated without wasting precious time and material.

In its upcoming expansion and modernization effort, too, WINTEC will thus continue to bet on controls from HEIDENHAIN!

RTS-RIEGERTEAM.DE

Find your way more quickly: smarT.NC

Never losing orientation is the key to attaining goals quickly and surely. The same applies to plain-language programming on your TNC control. HEIDENHAIN can show you how to work even more easily in the future with plain-language orientation using the new smarT.NC user interface. Never has programming, testing, and working been simpler. NC programs created with smarT.NC can be used in the conversational interface as well. So all the features that conversational professionals know and esteem have stayed, and beginners find their way even faster.

start smart.

DR. JOHANNES HEIDENHAIN GmbH, 83292 Traunreut
Germany, Phone +49 8669 31-0, Fax +49 8669 5061
www.heidenhain.de, E-Mail info@heidenhain.de

HEIDENHAIN