HEIDENHAIN

HSCI

CNC PILOT 640

The Control for Lathes and
Turning-Milling Machines

CNC PILOT 640 with 15.6-inch touchscreen

CNC PILOT 640 with 19-inch touchscreen

Windows 7, Windows 8, and Windows 10 are trademarks of Microsoft Corporation

Options are functions that are integrated into the control and with which you can adapt the feature range of the CNC PILOT 640 to your actual requirements, including at a later time if necessary. Some options need to be adapted by the machine manufacturer.

The features and specifications described here apply to the following control and NC software versions:
CNC PILOT 640 with NC software versions
688946-06 (export license required)
688947-06 (no export license required)

This brochure supersedes all previous editions, which thereby become invalid. **Subject to change without notice.**

Contents

The CNC PILOT 640 ...

Versatile and powerful
– CNC PILOT 640, the control for lathes and turning-milling machines

Thanks to its flexible design and numerous programming features, the CNC PILOT 640 always provides you with optimum support. Regardless of whether you are manufacturing one-off parts or large batches of simple or complex workpieces, the CNC PILOT 640 is characterized by its ease of use and user-friendly programming. It is easy to learn and requires minimum training time.

The CNC PILOT 640 was designed for CNC lathes and is ideal for both horizontal and vertical lathes as well as for vertical boring and turning mills.

The CNC PILOT 640 supports lathes with a main and counter spindle, C axis, or positionable spindle, as well as driven tools and machines with Y and B axes.

In multi-channel machining, different machining steps can be performed simultaneously using multiple slides.
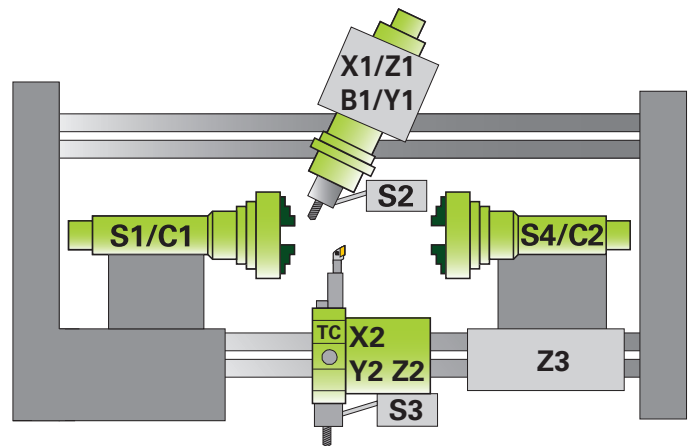
Regardless of whether you are turning simple parts or complex workpieces, the CNC PILOT 640 provides you with the benefits of graphical contour input and convenient programming with smart.Turn.

Programming with variables, controlling special machine components, or using externally created programs, etc., is no problem: simply switch to DIN PLUS. With DIN PLUS you'll find the solution for your special tasks.

Compatibility
You can also run existing NC programs from old HEIDENHAIN lathe controls on the CNC PILOT 640. With a convenient import filter, you can easily transfer older programs (e.g., from the CNC PILOT 4290) to the new control for continued use.
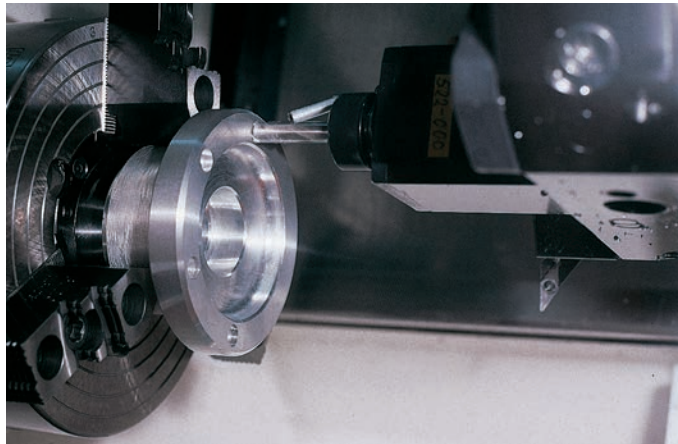
From simple jobs on a compact machine ...
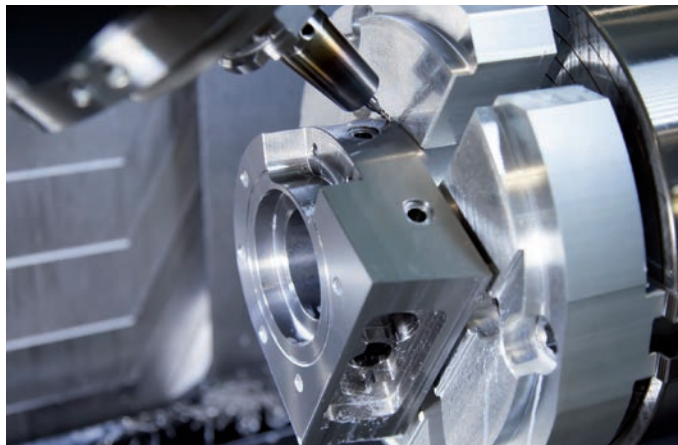
... to complex tasks,

... to serial production

Intuitive and user friendly
– State-of-the-art multitouch operation and clear-cut display

The screen

The CNC PILOT 640 is equipped with a practical touchscreen designed for harsh shop conditions. It is splash-proof, scratch-resistant, and features an IP54 protection rating. The screen can be operated by means of gestures such as those commonly used on mobile devices.

The CNC PILOT 640 is available in two screen formats. The 19-inch monitor particularly lends itself to large or multi-channel machines requiring a larger field of view. The smaller screen in the 15.6-inch widescreen format is particularly well-suited for compact machines. The widescreen format ensures a clear overview of the programs—particularly in programming mode.

The CNC PILOT 640 gives you a clear view of all the information you need for programing, operating, and checking the control and machine.

During program input, the required parameters are illustrated in help graphics. In the simulation, all movements of the tool are displayed in realistic detail. During program run, the CNC PILOT 640 shows you complete information on the tool position, speed, and load of the drives, as well as the current machine status.

The keyboard

The CNC PILOT 640 needs very few keys. The functions are clearly identifiable thanks to easily understood symbols.

The keys on the numeric keypad are used for both data input and selection of the functions. The menu window graphically displays the available functions. You can use soft keys to modify the selected functions, confirm position and technology values, and control the data input.
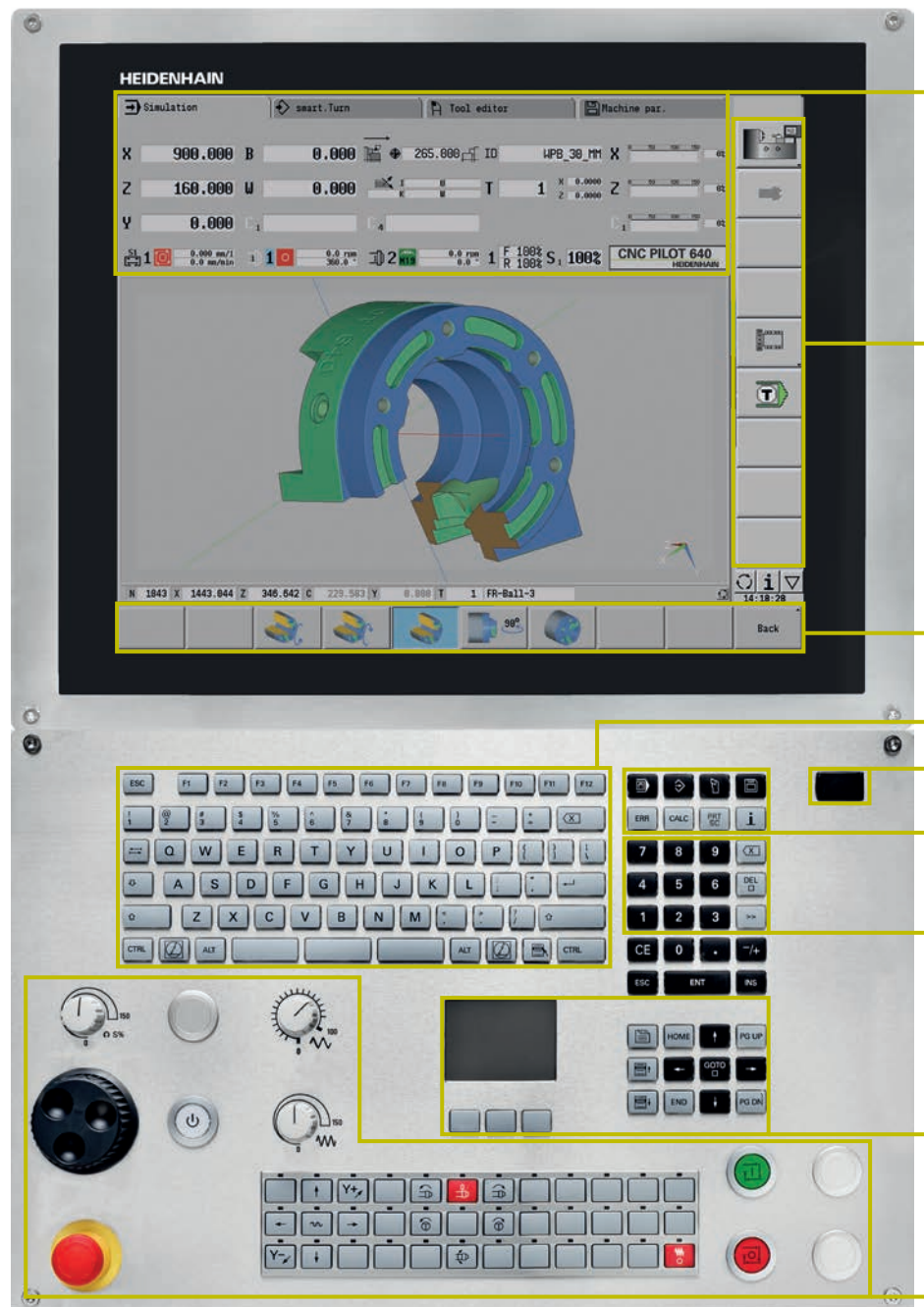
Display of operating modes and the machine status (configurable)
You can select the suitable display field for each of the up to 25 segments
and save different display fields for the automatic and manual modes.

PLC soft keys for machine functions

Self-explanatory soft keys for the respective operating mode

Alphanumeric keyboard for comments

USB interface

Keys for operating modes and functions

Keypad for numerical input and fast, direct menu selection

Navigation keys

Machine operating panel with override potentiometers

Ergonomic and sturdy design
The optimized stainless-steel operating panel of the CNC PILOT 640 features a special protective coating, making it particularly resistant to contamination and wear. The labeling of the ergonomically formed keys can withstand extreme shop conditions. With the handy control knobs, you can make fine adjustments to the feed rate, rapid traverse, and spindle speed.

Intuitive touchscreen operation
The conventional way of operating the CNC PILOT has proven itself over many years. Operators around the world use their CNC PILOT with dialog keys, navigation keys, and soft keys.

In a touchscreen version, the CNC PILOT 640 now supports you with a particularly innovative and user-friendly operating interface. It combines the proven benefits of the HEIDENHAIN controls with a modern method of operation based on tapping, swiping, and dragging gestures.

Practical touchscreen
The touchscreen is ideally suited to a shop environment. It can still be operated even when exposed to cooling lubricant, oil, etc. If you wish to clean your screen, you can simply select the "Touchscreen Cleaning" mode to lock it. The touchscreen can also be operated with different types of work gloves.

Gestures for multitouch operation
The screen of the CNC PILOT 640 can be operated with gestures like those commonly used on your smartphone or tablet. For example, you can zoom in or out with two fingers. By swiping, you can very quickly navigate through soft-key rows, programs, or menus.

Symbol	Gesture
	Tap
	Double tap
	Long press
	Swipe
	Drag
	Two-finger drag
	Spread
	Pinch

Quick and reliable machining with high contour fidelity
– Uniformly digital control design

Thanks to its digital design, the CNC PILOT 640 has the machine's entire drive system under control. Not only does the field-proven digital drive technology from HEIDENHAIN enable high contour fidelity and rapid machining at high speeds but also, with the CNC PILOT 640, all of the control components are connected over a digital interface.

Digital drive technology
The position controller, speed controller, and current controller (if needed) are integrated into the CNC PILOT 640. Digital motor control makes it possible to attain very high feed rates.

High contour fidelity
The CNC PILOT 640 dynamically calculates the contour in advance. The lathe control thereby adapts the axis speed to the contour transitions early enough. It controls the axes with special algorithms that ensure path control within the speed and acceleration limits.

High availability
In the uniformly digital control solution of the CNC PILOT 640, all components are connected over purely digital interfaces: the control components over HSCI (HEIDENHAIN Serial Controller Interface),

the HEIDENHAIN real-time protocol for Fast Ethernet, and the encoders over EnDat 2.2, the bidirectional interface from HEIDENHAIN.

This achieves a high degree of availability for the complete system. It is diagnosable and immune to noise—from everything from the main computer to the encoder.

The uniformly digital solution from HEIDENHAIN ensures exceptionally high accuracy and surface quality at high traversing speeds.
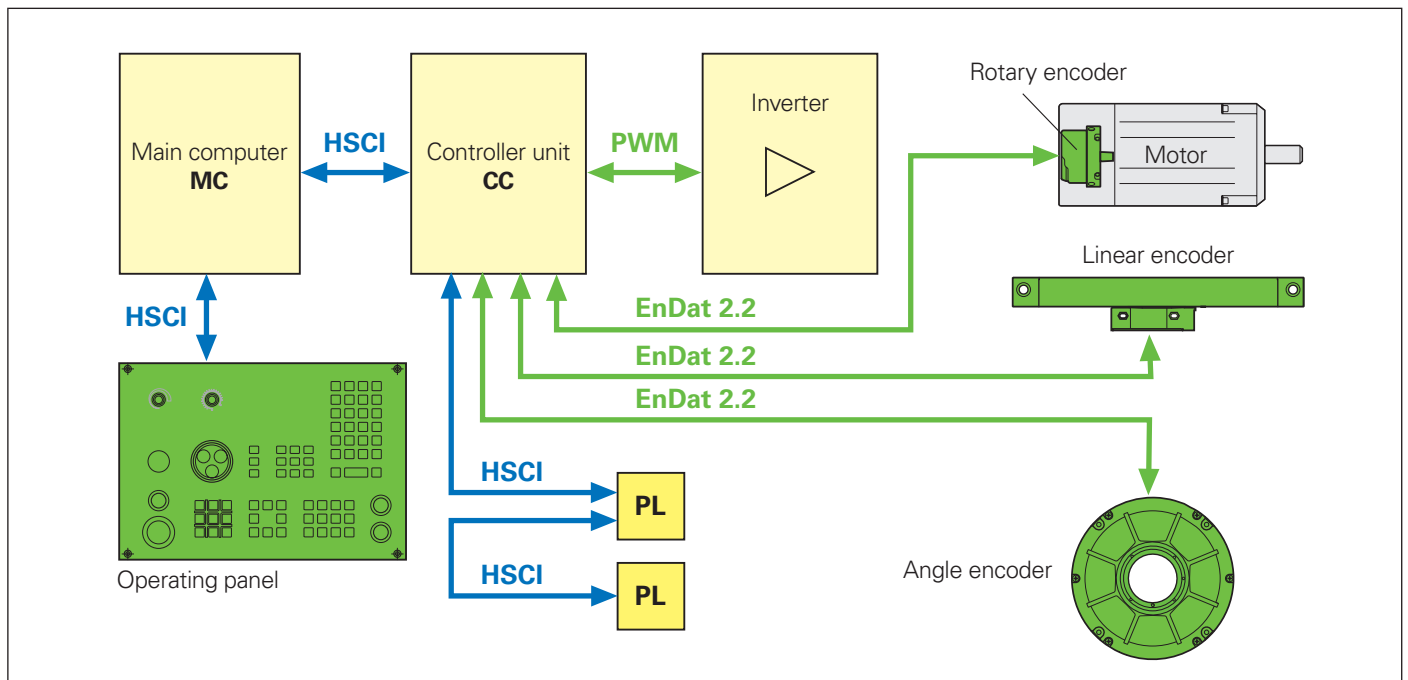
– Intelligent machining

Load monitoring – detection of tool wear and breakage during machining*
Load monitoring tracks the machine's spindle and motor loads while comparing them with the load values of a reference operation. The CNC PILOT 640 can graphically display the load values in a separate window.

You can set two limit values that trigger different error reactions. After the first limit value is exceeded, the current tool is flagged as worn out, and the control automatically exchanges it on the next tool call with a previously defined replacement tool. After the second limit is exceeded, the CNC PILOT 640 assumes that there is an impermissible load (e.g., tool breakage) and stops the machining process. This improves process reliability during machining, particularly during unmanned shifts.

* These functions must be implemented in the machine and control by the machine manufacturer.

Batch mode
The batch mode function allows you to run multiple main programs automatically. You can also define the NC programs and the respective quantities in a program list. The CNC PILOT 640 can then execute this list on its own and without you having to select and start programs in between. Batch mode is particularly suitable for different operations performed with the same raw material supplied by bar loaders or robots.
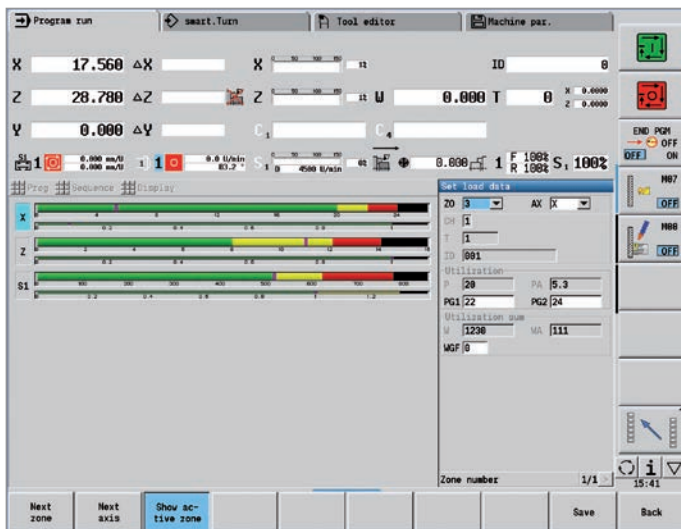
Graphical display of the load values

Effective, well structured, and flexible
– Convenient programming with smart.Turn (option)

Has the safety clearance been correctly entered, is the speed limit taken into account, and how are the oversizes defined? All of these items need to be considered not only by the beginner but also by the experienced NC programmer when creating conventional DIN programs.

The smart.Turn principle

The working block—called a unit—plays a pivotal role in smart.Turn programs. A unit completely and unambiguously defines a single working step. The unit includes the tool call, the technology data, the cycle call, the approach and departure strategies, and global data such as the safety clearance, etc. All of these parameters are displayed together in a clearly structured dialog box.

The smart.Turn principle gives you the reassurance that the working block is defined correctly and completely. In the NC program, smart.Turn lists the DIN PLUS commands of the unit. This gives you an overview of all of the working-block details at any time.

The unit

All of the parameters of a smart.Turn unit are brought together in simple and well-designed fillable forms. The overview form provides you with an overview of the selected unit, while subforms provide information on the details of a working block. Clearly arranged help graphics illustrate all of the required inputs. If input options are available, smart.Turn displays a list of the available options for selection.

By the way, you do not need to stop the manufacturing process to program with smart.Turn. You can create and test the smart.Turn program while a program is running.
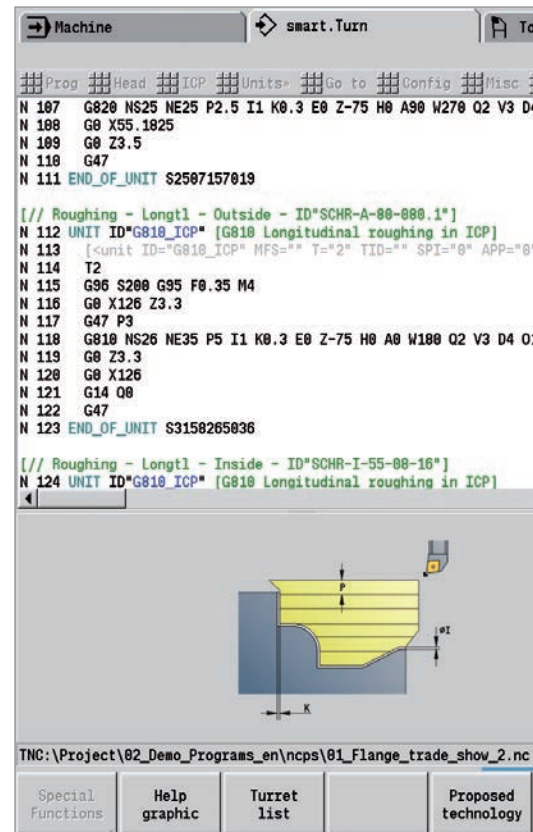
Straightforward dialogs; help graphics illustrate the parameters.
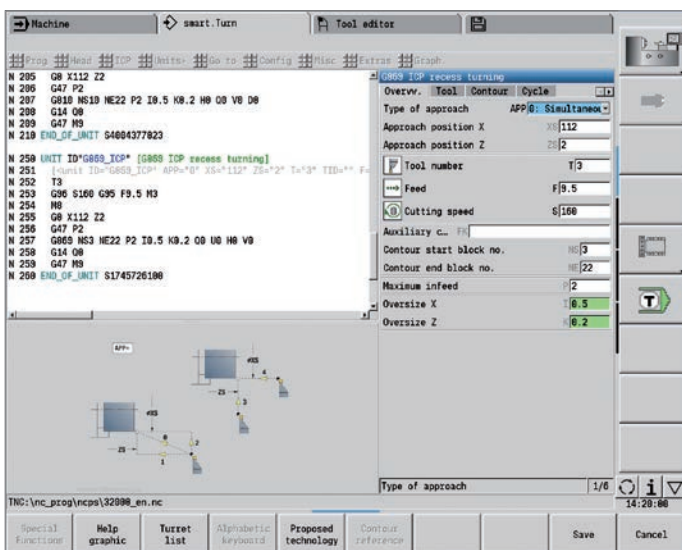
Input form in smart.Turn

smart.Turn unit in the NC program
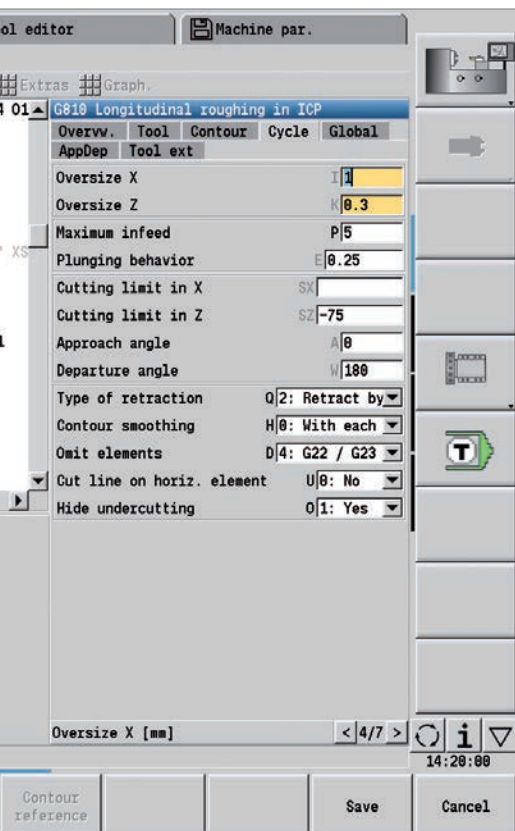
Structured and easy-to-read

Programs created with smart.Turn are characterized by their clear structure and easy understandability. The smart.Turn tool uses section codes that clearly distinguish between the program head with the setup information, the turret assignment, the workpiece description, and the actual machining operation.

Guided by the dialog, you enter the following data in this sequence:
• Program head
• Tool assignment in the turret
• Description of workpiece blank
• Description of finished part
• Individual machining steps

The smart.Turn method not only ensures that the program is easy to read; it also makes it possible to save all of the information required for producing the workpiece in the NC program.

Production data at a glance

The program head includes all of the key information about the workpiece (e.g., drawing number, date, programmer, material, fixture, etc.).

All of the information that is important for setting up and machining the workpiece, such as the tool assignment in the turret, is included in the NC program.

Programming in more than one window

Up to six NC programs can be opened simultaneously in the DIN PLUS editor. The NC program to be displayed is selected via the smart keys. This enables you to transfer program blocks from one NC program to another and allows you to quickly gain an overview of complex NC programs, including subprograms.
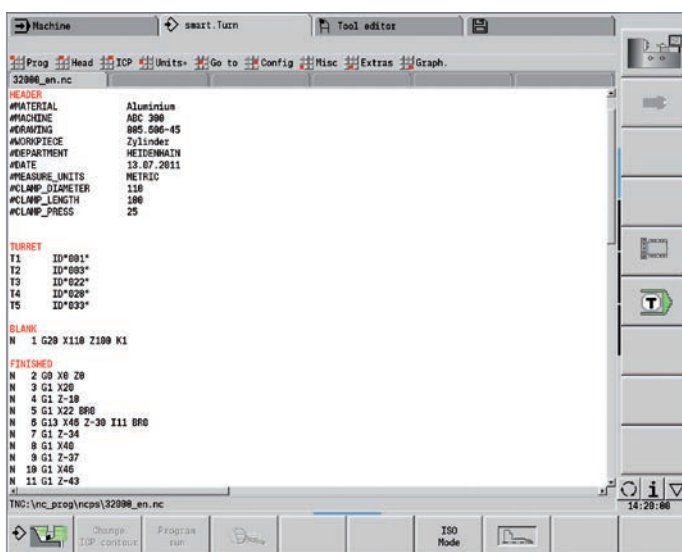
A smart.Turn program with section codes
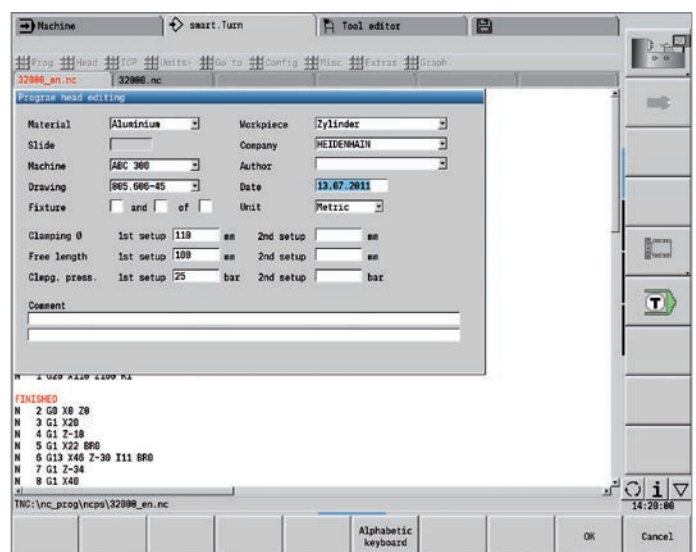
Input form for program head

Effective, well structured, and flexible
– Convenient programming with smart.Turn (option)

Programming made simple

Global parameters, such as oversizes, safety clearances, coolants, etc., are defined once in the start unit. Then smart.Turn transfers these parameters to the other units.

In the NC program, smart.Turn lists the DIN PLUS commands of the unit. This not only gives you an overview of all of the working block details but also provides you with a clearly arranged and well-structured NC program.

The smart.Turn tool supports units for roughing, finishing, recessing, recess turning, thread cutting, drilling, tapping, and milling, as well as special units for program start, program end, moving the C axis in/out, subprograms, and program-section repeats.

Programming contours

The smart.Turn tools enables you to work simply and flexibly. Simple contours can be defined with just a few entries in the cycle. Complex contours are described by means of ICP graphical interactive programming. Workpiece descriptions that are available in DXF format can be easily imported. Contours are always stored in the NC program such that they are "readable" and "editable." This gives you the benefit of choosing either smart.Turn or the ICP editor to edit the programs.

Contour follow-up

Another highlight of the CNC PILOT 640 is its contour follow-up feature. If you define the workpiece blank at the beginning of your smart.Turn or DIN PLUS program, the control then computes the resulting new blank for each new cut. The machining cycles always adapt to the current workpiece blank. Contour follow-up helps you to avoid air cuts and to optimize approach paths, even through space where workpiece material has been removed.

Technology data as suggested values

The CNC PILOT 640 saves your cutting data according to the criteria of workpiece material, tool material, and machining mode. Because you specify the cutting material in the tool definition, all that remains is to enter the material of your workpiece. This provides smart.Turn with all of the data it needs in order to set default values for the cutting data.
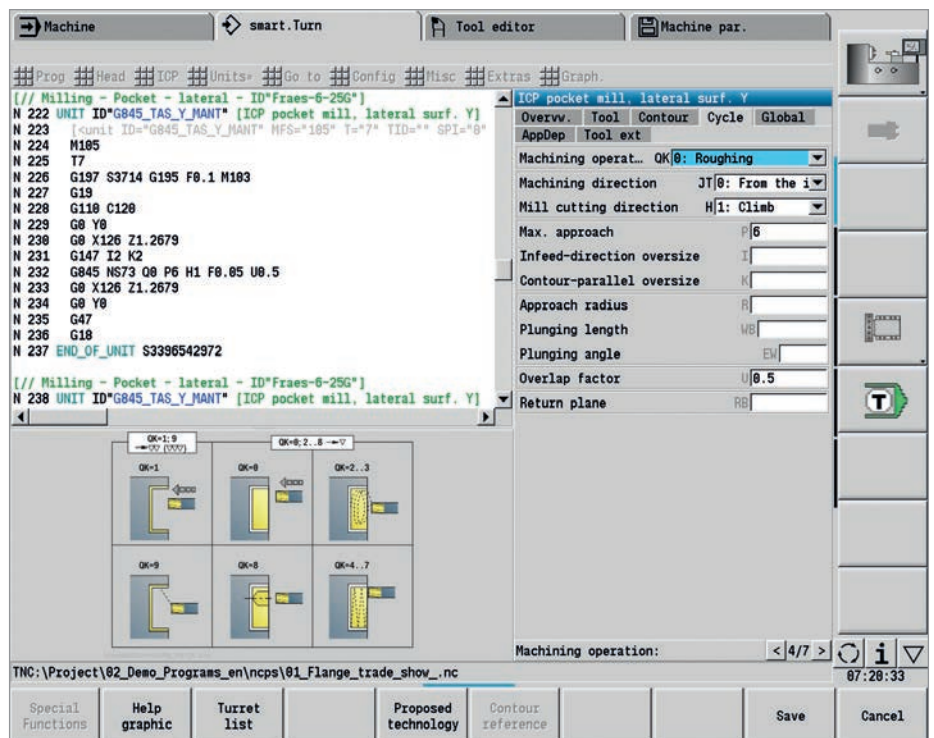
Dialogs of the smart.Turn tool with help graphic

– Powerful NC programs with DIN PLUS

Programming in DIN PLUS

The smart.Turn tool offers units for all machining tasks, as well as units for special functions. However, if you wish to control special machine components or use the variable programming function or other complex functions that are not provided by smart.Turn, then use DIN PLUS. It provides powerful machining cycles, program branching, and programming with variables. You can switch back and forth between the smart.Turn and DIN PLUS programming modes within a program.

Because the units are based on DIN PLUS, you can "dissolve" a unit into blocks at any time in order to modify and optimize the resulting DIN PLUS program section.

Of course, the CNC PILOT 640 also allows you to create a DIN program, or to import and use externally created programs.

Powerful cycles in DIN PLUS

In the fixed cycles of DIN PLUS, you define the contour section to be machined. You simply mark the area to be machined in the control graphic. Then you can immediately test each working step in the simulation.

You select the respective commands from a menu or enter them directly in G-code. The screen displays a dialog box in which you enter the related parameters. All of the input is explained on the screen in plain language and with graphical illustrations.

Thanks to the powerful fixed cycles and the assignment of cycles to the sections to be machined, DIN PLUS allows you to increase effectiveness and flexibility compared with conventional NC programming.
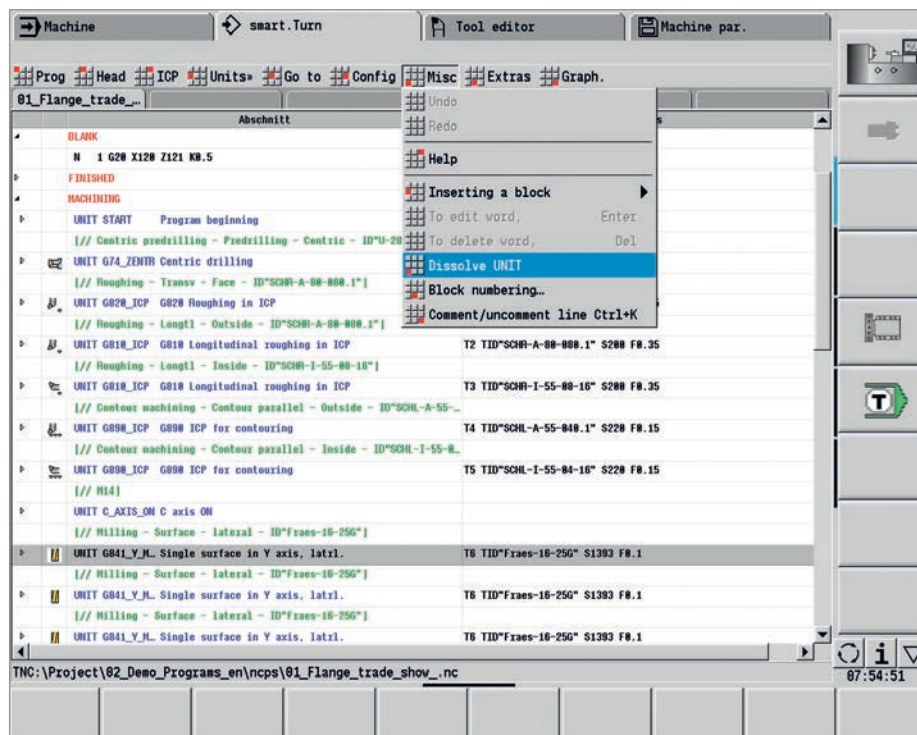
smart.Turn program in tree view with a dissolved smart.Turn unit

Effective, well structured, and flexible
– An NC program at the stroke of a key with TURN PLUS (option)

With TURN PLUS, you can create NC programs in very little time. After you have described the contour of the workpiece blank and finished part, you need only select the material and clamping devices. TURN PLUS does everything else automatically: it generates the working plan, selects the machining strategy, selects the tools and cutting data, and generates the NC blocks.

The result is a comprehensively commented smart.Turn program with working blocks (units). This gives you leeway for optimization and safety when first running the NC program.

TURN PLUS can do all of this for milling and drilling operations with the C or Y axis on front face and lateral surfaces, as well as on rear-face surfaces on machines with counter spindles.

The NC program at a keystroke
If short programming times are important, you can generate all of the working steps with only a single keystroke. Based on the entered contour and the information from the technology database, TURN PLUS creates the working plan and selects suitable machining strategies, tools, and cutting data, all on its own. The whole operation takes only a few seconds. You can monitor each individual step in the control graphic.

TURN PLUS uses a reasonable sequence of possible operations, such as "first roughing transverse, then roughing longitudinal" or "finishing outside, then finishing inside." However, you can also adapt this sequence on your own to suit various tasks. In this way, the CNC PILOT 640 can also harness your company's know-how during automatic working plan generation.

Automatic program generation for the complete contour
The CNC PILOT 640 automatically generates the NC program even for complex workpieces with machining operations on the front face, back face, and lateral surfaces. After defining the geometry, you can benefit from time savings of approximately 90%—time otherwise needed for the programming of the machining operations.

Automatic program generation for the second setup
TURN PLUS knows the contours of the clamping devices and takes them into account when generating the working plan. It thereby automatically limits the tool path to a safe distance from the clamping device. Once the program for the first setup is complete, you can "rechuck" using interactive graphics. The control then automatically generates the program for the second setup without you needing to re-enter the workpiece geometry.
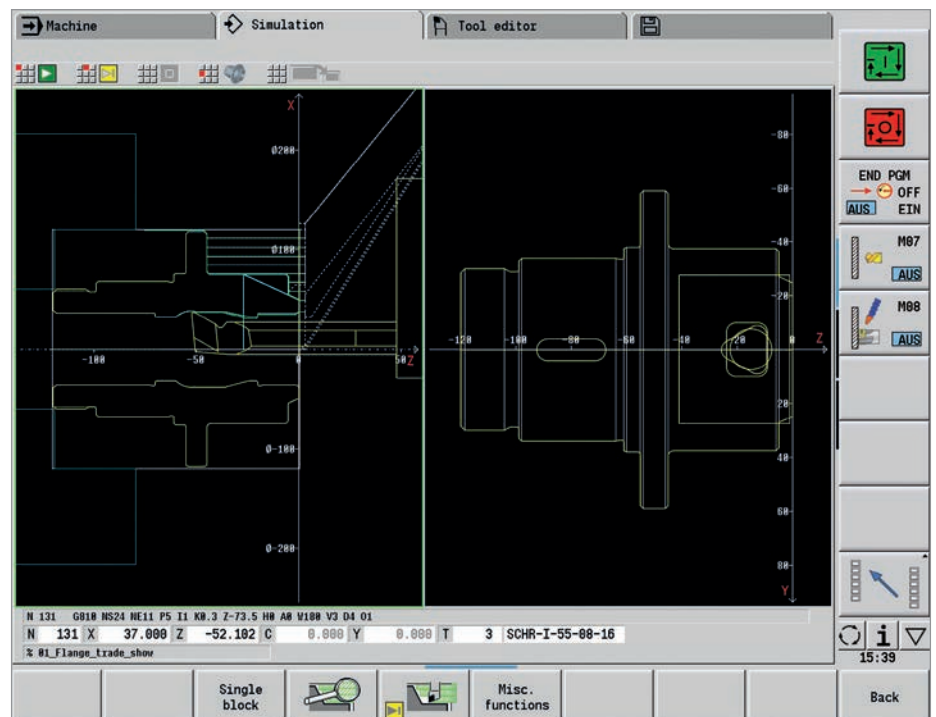
Inclined contours are no problem
When it comes to automatically generating the NC program for inclined contours, the control is confronted with unique demands. The angle at which the contour falls is often steeper than the angle of the tool tip. In this case, the control automatically selects a different tool and performs the machining operation in the opposite direction or as a recess. In any case, the result is an executable NC program.

Defining the machining sequence
You can define a standard machining sequence in the TURN PLUS dialog. Various machining sequences, such as for chuck parts, shaft machining, etc., can be stored.

From the global main types of machining, such as "roughing," "finishing," or "drilling," to details such as defining a tool for a specific operation, the automatic working plan generation (AWG) feature can be adapted to suit the user's requirements.

Technology data

Material

Velocity

Feed rate

Tools

TURN PLUS

```
//Contour machining – Contour parallel – Outside
N    52 UNIT ID"G890_ICP" [G890 ICP for contouring]
N    53    [<unit   ID"G890_ICP" MFS="" T="4" TID="" SPI
N    54    T4
N    55    G96 S240 G95 F0.25 M4
N    56    G0 X6 Z3
N    57    G47 P2
N    58    G890 NS4 NE13 E0.2 V3 Q0 M0 Z-70 D1 I45 K-67
N    59    G14 Q0
N    60    G47
N    61 END_OF_UNIT  S3005992634
```
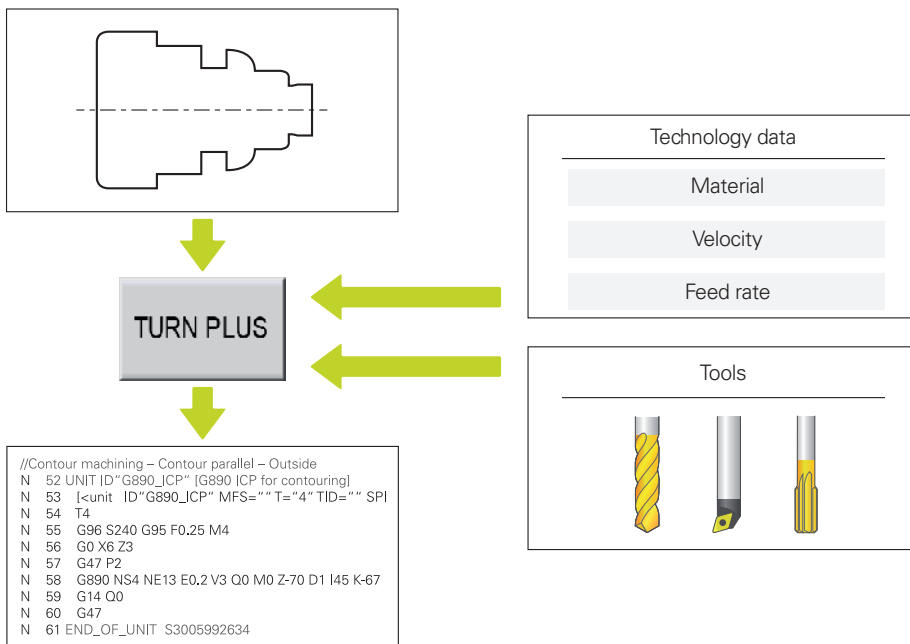
Go from programming to the first cut in very little time with TURN PLUS

Describing and importing contours
– ICP interactive contour programming

For jobs that cannot be machined with the standard cycles due to the complexity of the workpiece or the lack of certain dimensions in the workpiece drawing, you need ICP. You describe the contour elements directly as they appear in the workpiece drawing. Or—if the drawing is available in DXF format—you simply import the contour.

Contour programming with ICP
You define an ICP contour by entering the contour elements one after the other in the graphic editor. When selecting the contour elements, you specify the direction of the line or the direction of rotation of the circular arc. As a result, the CNC PILOT 640 requires very little information about the contour element.

When entering the data, you decide whether the coordinates are to be absolute or incremental, whether lines are defined by their end point or length, and whether circular arcs are to be specified by their center point or radius. You also specify whether the path to the next contour element should be tangential or non-tangential.

The CNC PILOT 640 then calculates missing coordinates, intersections, center points, etc., provided that they are mathematically defined. If the entered data permit multiple mathematically possible solutions, then you can view the alternative proposals and select the one you prefer. You can also add to or modify existing contours.

Superimposing form elements
The ICP editor recognizes the chamfer, rounding, and undercut form elements (DIN 76, DIN 509 E, DIN 509 F, etc.). You can enter these form elements during the course of sequential contour definition. However, it is often easier to first define the "rough" contour and then superimpose the form elements. This is done by selecting the corner on which the form element is to be placed and then inserting the element.

ICP contours for smart.Turn and DIN PLUS
In smart.Turn, you have various possibilities for describing the contour to be machined. You can describe simple contours right within the unit, or you can use ICP to define complex turning or milling contours as well as linear or circular drilling and milling patterns. The contour defined with ICP is incorporated into the smart.Turn program. Within the unit, you enter a reference to the contour section to be machined. ICP contours are saved in the NC program in a readable and editable form. This gives you the benefit of choosing either smart.Turn or the ICP editor to edit the contours.

If you are working in DIN PLUS mode, you can likewise use ICP to describe the turning and milling contours, as well as the linear and circular patterns. In the contour-related cycles, you enter a reference to the contour section to be machined.

The ICP editor can be called immediately from within smart.Turn.
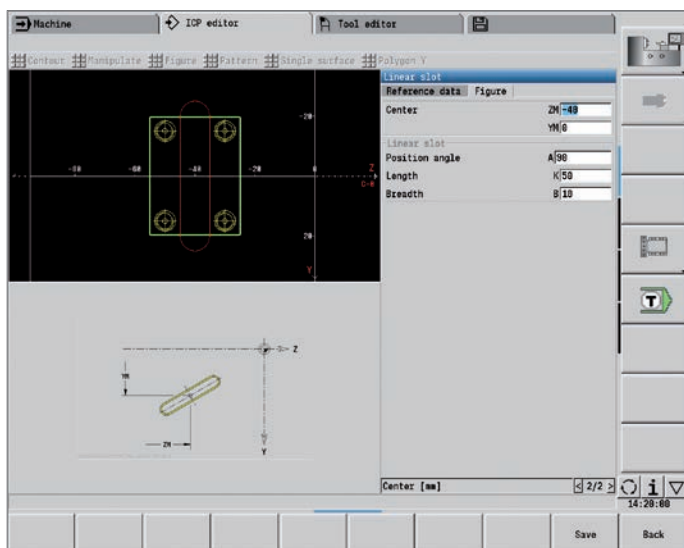
Define the contour element in ICP
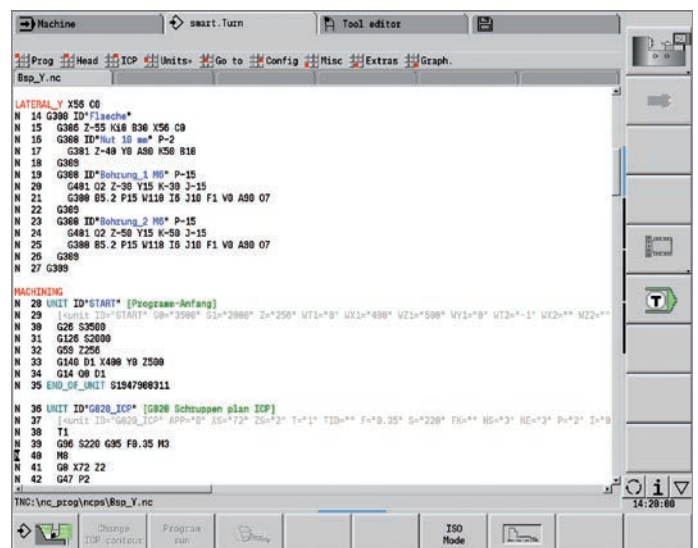
ICP contour description in the NC program

DXF import of contours (option)

Why should you painstakingly enter contour elements if the data already exists in the CAD system? ICP makes it possible to import contours in DXF format directly into the CNC PILOT 640. Not only do you save time that would otherwise be spent on programming and testing but you also ensure that the finished contour exactly corresponds to the design engineer's specification. DXF contours can describe workpiece blanks, finished parts, contour trains, and milling contours. They must be available as two-dimensional elements within a separate layer (i.e., without dimension lines, wrap-around edges, etc.).

First, you download the DXF file to the CNC PILOT 640 over the network or via a USB stick. Since the DXF format is fundamentally different from the ICP format, the contour is converted from DXF to the ICP format during the import. This contour is then treated as a normal ICP contour and is available for smart.Turn or DIN PLUS programming.
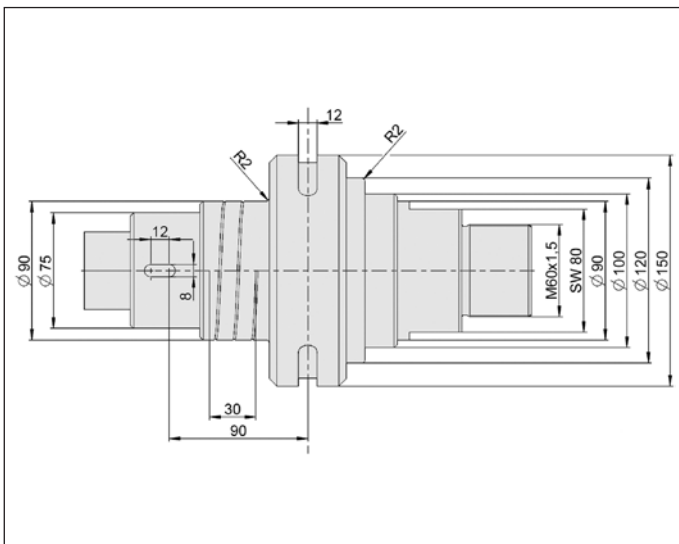
Workpiece drawing
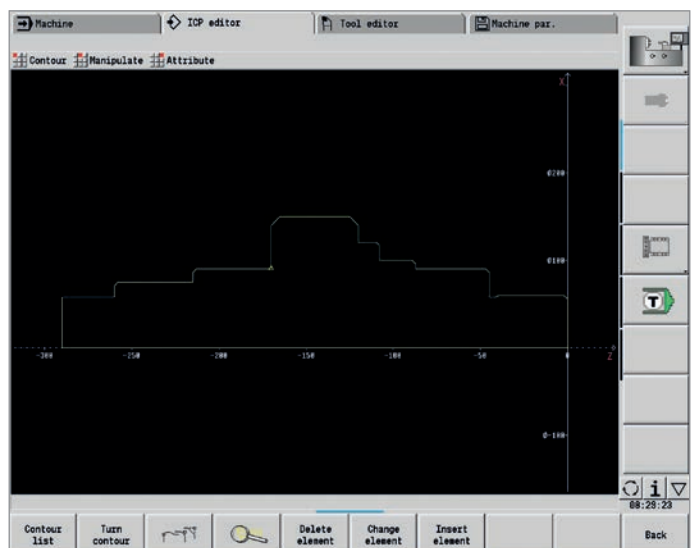
Contour description in the ICP editor

17

Realistic testing before machining
– Graphical simulation

The timely detection of errors is very important in NC programming. The CNC PILOT 640 supports you during debugging by means of the graphical simulation. This simulation is exact and provides the actual dimensions of the contour and of the tool cutting edge because the simulation works with the geometry values from the tool database.

Graphical simulation

Before actual machining, you can use the graphical simulation to inspect the following:
- Approach and departure behavior
- The machining sequence
- The proportioning of cuts
- The finished contour

The graphical simulation allows you to display the tool's cutting edge. You can view the cutting-edge radius, width, and position rendered true to scale. This helps you recognize machining details or collision risks in advance.

Wire-frame or cutting-path graphics, and machining simulation

The CNC PILOT 640 supports various views of the tool paths and the machining process. You can choose the type of verification best suited to the tool or machining process used.

The **wire-frame graphics** are particularly convenient if you need only a rapid overview of the approaching and departing movements and the proportioning of cuts. The wire-frame graphics illustrate the path of the theoretical cutting point.

A more accurate method of contour verification is provided by the **cutting-path graphics**. The cutting-path graphics account for the exact geometry of the tool tip. You immediately see whether material will be left behind, the contour will be damaged, or whether the overlaps will be too large. The cutting-path graphics are particularly helpful in recessing, milling, or drilling operations in which the tool shape has a decisive influence on the final outcome.

The **machining simulation** (material removal graphics) realistically demonstrates the cutting process. The workpiece blank is shown as a filled-in surface. The CNC PILOT 640 simulates every tool movement at the programmed cutting speed and removes the "machined" material.
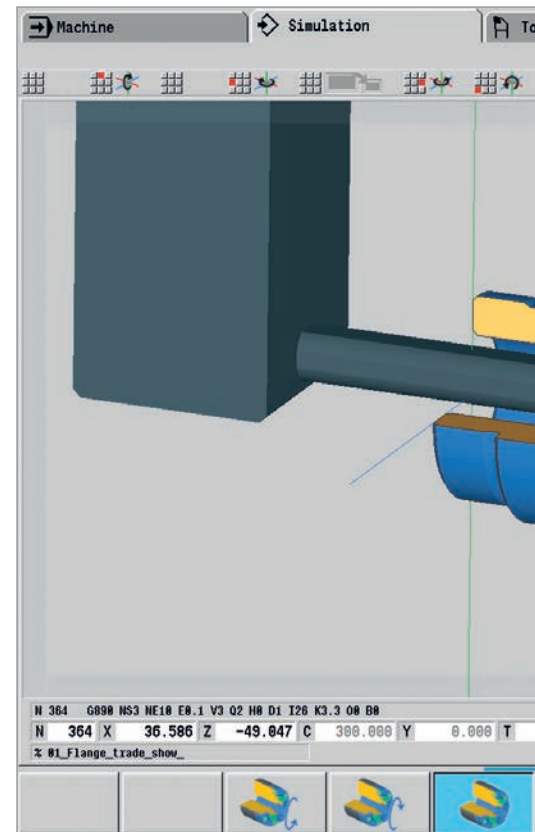
Finished part in a 3-D graphical image

Wire-frame graphics
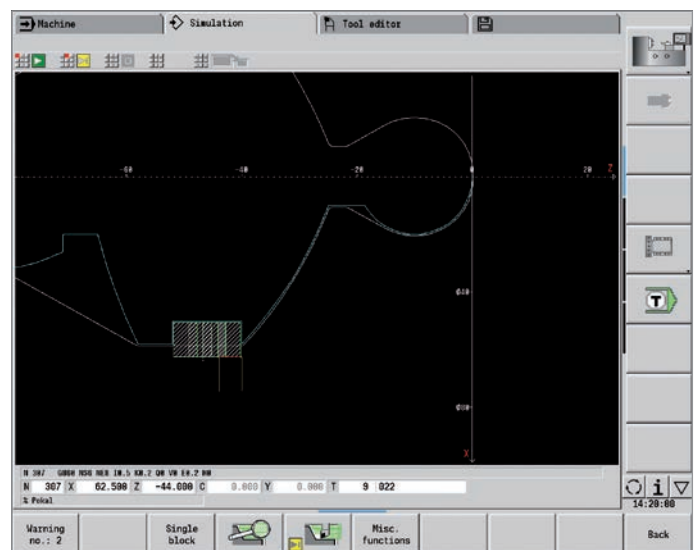
Cutting-path graphics
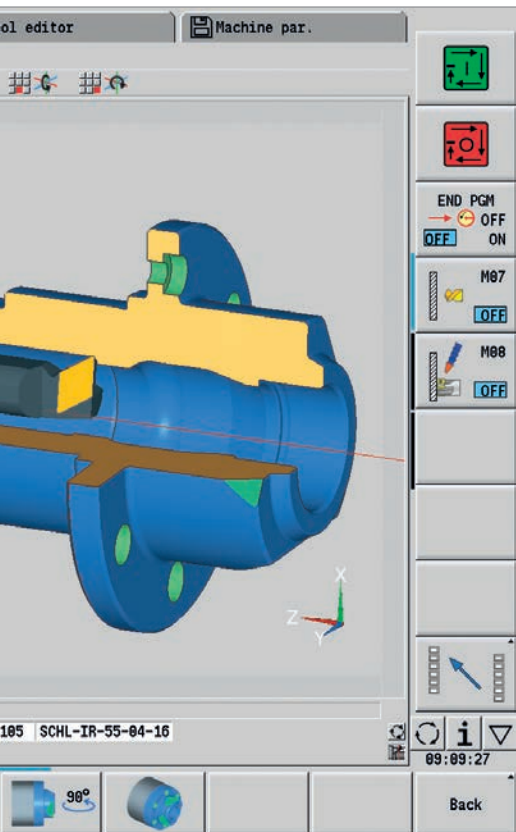
Configuring the views

If your lathe is equipped with driven tools and a positionable spindle, C axis, or Y axis, then the CNC PILOT 640 also simulates the machining of the front face and lateral surface, or the XY and YZ plane. You can select the combination of windows best suited to the job. This gives you everything you need for a precise inspection of your drilling and milling operations.

The CNC PILOT 640 depicts C-axis machining of the cylindrical surface as an "unrolled" plane surface.

3-D simulation

With the high-resolution, realistic 3-D simulation, you can precisely evaluate the outcomes of drilling, turning, or milling operations, even prior to actual machining. This also applies to complex multi-channel operations, for which all of the workpieces and tool movements of all of the slides are displayed. Milling and turning operations are shown in different colors.

The view, which is freely rotatable about all axes, permits visual inspection of the workpiece blank and finished part from all angles. With its intuitive gesture-controlled operation, you can navigate and zoom in on any programmed detail—including, of course, for C-axis contours on the cylindrical surface or front face, and for Y-axis contours in the tilted plane. In this manner, the 3-D simulation makes it possible for you to detect even very small errors prior to machining.

Calculating the machining time

If your customer needs an offer in a hurry, and you require exact information within very little time, the CNC PILOT 640 is a valuable aid thanks to its machining time calculator. During simulation of the smart.Turn or DIN PLUS program, the CNC PILOT 640 calculates the time per piece for the programmed machining work.

Along with the total time, the table displays the machining time and idle time of each cycle or of each tool insert. And this can help with more than just cost calculations, since you can also tell at a glance whether there are ways to further optimize the machining process.
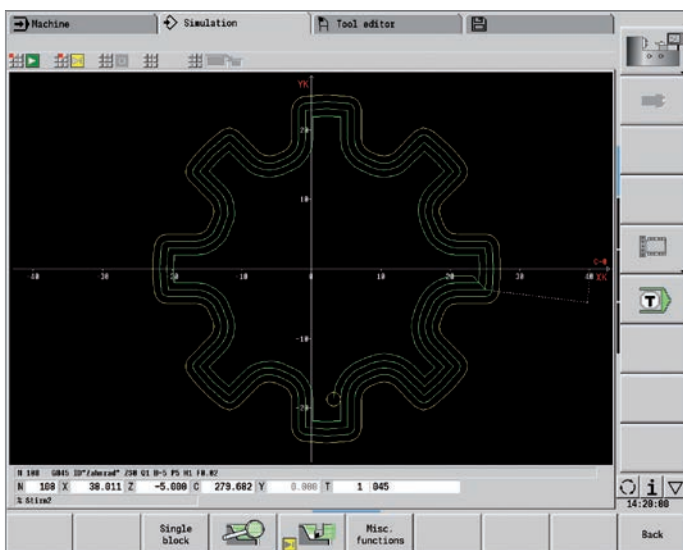
Machining the front face

Calculation of machining time

19
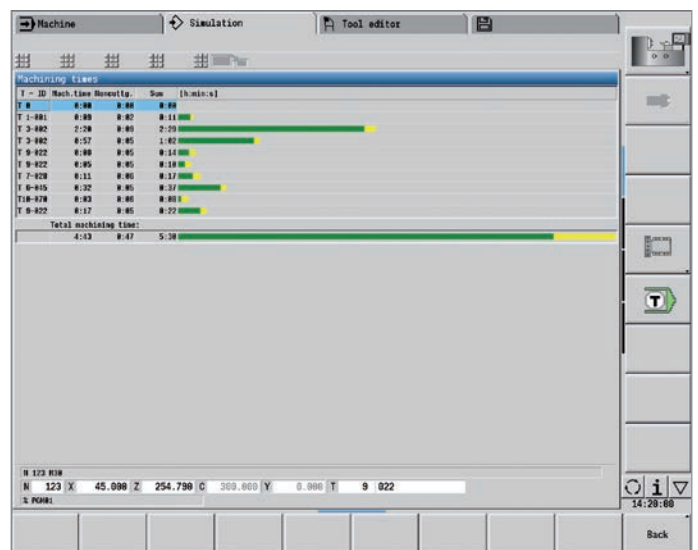
Expandable for complex tasks
– Turning, drilling, and milling in a single setup (option)

You can use the CNC PILOT 640* to drill and mill your workpiece on its front face and lateral surface, all within a single setup. In addition, the control offers numerous functions and practical cycles.

* Optional. The machine and the CNC PILOT 640 must be adapted to this function by the machine manufacturer.

Drilling, deep-hole drilling, tapping
The CNC PILOT 640 drills, performs deep-hole drilling, and taps individual holes with the C or Y axis. Using parameters, you can easily program infeed reductions when drilling partly or completely through the workpiece.

Drilling and milling patterns
If holes, slots, or ICP milling cycles are located at regular intervals along a straight line or circular arc, the CNC PILOT 640 greatly simplifies your work: you can create these patterns on the front face or lateral surface with just a few keystrokes.

Thread milling
On lathes equipped with a C or Y axis, you can take advantage of thread-milling because the CNC PILOT 640 supports special thread-milling tools.

Milling slots and simple figures
Slot milling with the CNC PILOT 640 is very simple. You define the position and depth of the slot, as well as the cutting values, and the milling cycles then take care of the rest automatically.

For simple contours such as circles, rectangles, and equilateral polygons, defining the figure and position likewise requires only a few keystrokes.
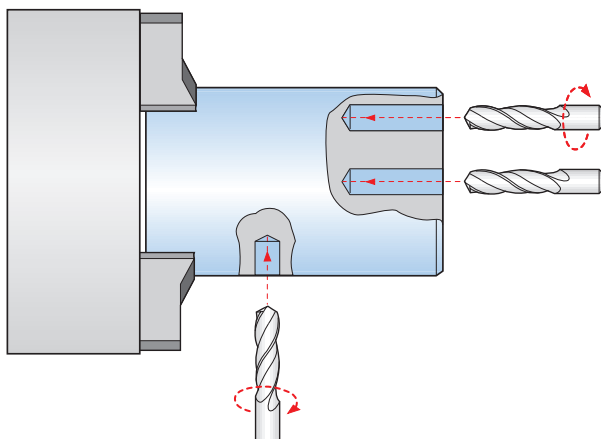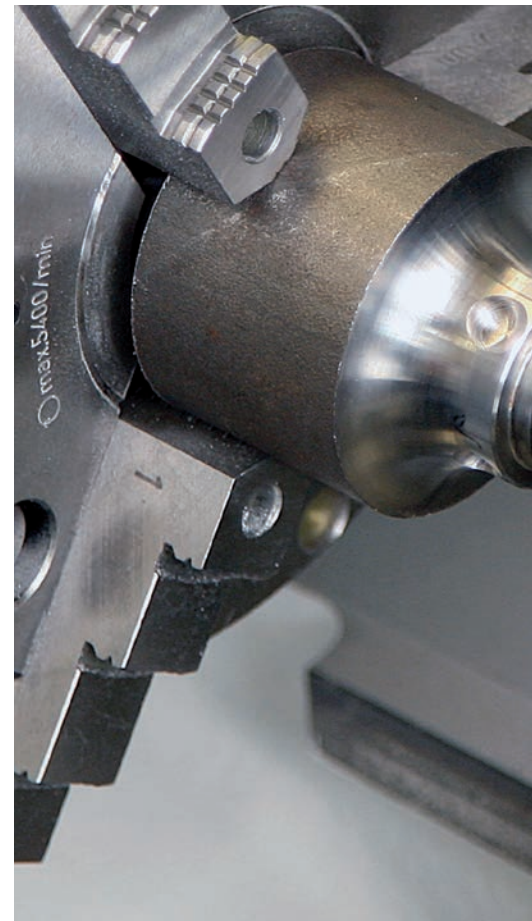
Drilling

Deep-hole drilling

Tapping

Thread milling

Drilling or tapping

smart.Turn units and DIN PLUS cycles for drilling
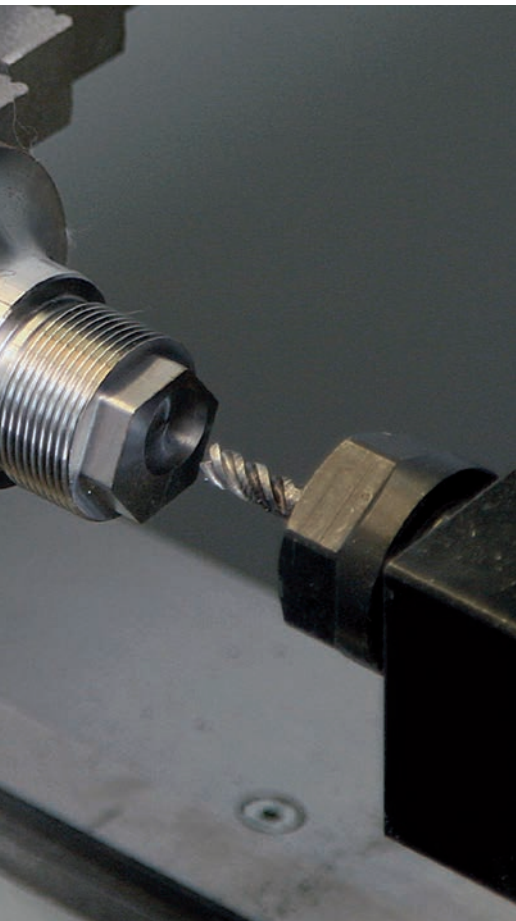
Contour and pocket milling

The CNC PILOT 640's milling cycles support both contour and pocket milling. You determine all of the essential details, such as the machining direction, milling direction, approach and departure behavior, infeeds, etc. The CNC PILOT 640 automatically compensates for the tool radius.

You can mill the pocket in two stages—first roughing and then finishing. The result is high accuracy and excellent surface quality.

For programming in smart.Turn and G-code, the CNC PILOT 640 supports various infeed strategies. You can choose between direct, reciprocating, or helical infeed, or infeed at the predrilling position.

Face milling

The face milling cycle machines individual surfaces, equilateral polygons, or a circle—even off-center.

Helical slot milling

The helical slot milling cycle is useful for machining lubrication grooves. You specify all of the important parameters such as the pitch, milling at multiple infeeds, etc.

Engraving cycles

Do you want to "inscribe" your workpieces? That's no problem with the CNC PILOT 640. The smart.Turn engraving units require only a few parameters in order to engrave characters of any size on the front face or lateral surface, or on the XY or YZ plane.

On the front face of the workpiece, you can arrange the characters along a line or an arc. On the lateral surface, and when engraving with the Y axis, you can define the angle at which the characters are to be arranged. With the time function, you can engrave the current time and date on your workpiece.

Of course, the engraving cycles are also available as DIN PLUS cycles.

Deburring

The CNC PILOT 640 supports special units or DIN PLUS cycles for deburring. You enjoy the benefit of being able to program this operation with only a few parameters.

Trochoidal milling

The trochoidal milling method allow you to efficiently machine pockets in different shapes (circle, rectangle, and polygon), as well as any desired slot. With this technique, roughing is performed with circular movements that are superimposed onto a linear forward motion. Trochoidal milling enables high removal rates and reduces chatter.

Slot milling

Figure milling (circles, rectangles, regular polygons)

ICP contour milling

Face milling (single surfaces, flat surface, polygon)

Helical slot milling

smart.Turn units and DIN PLUS cycles for milling
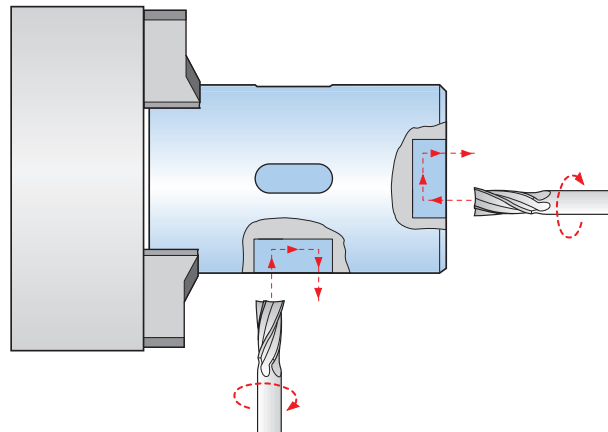
Front face and lateral-surface milling

Expandable for complex tasks
– Full-surface machining including the C axis and Y axis (option)

The CNC PILOT 640 provides a solution for any machining task and machine configuration: it enables complex machining tasks with a C or Y axis. It also controls full-surface cutting on dual-spindle machines.

Likewise, for C-axis, Y-axis, and full-surface machining, you can select from the DIN PLUS, smart.Turn or Teach-in programming modes.

C axis or positionable spindle*
For more complex tasks, the CNC PILOT 640 can be expanded to also control a C axis or positionable spindle and a driven tool. The driven tool makes it possible to drill off-center and to tap holes while the main spindle is at rest. The C axis or positionable spindle permits milling and drilling operations on the front face and lateral surface of the workpiece. For the programming and inspection of these machining sections, the workpiece is shown in the front face view and in the unrolled lateral surface view.

Y axis*
With the Y-axis option of the CNC PILOT 640, you can machine slots or pockets with plane bottoms and perpendicular slot corners. By defining the spindle angle, you can determine the position of the milling contours on the workpiece. For programming and verification of these machining sections, the workpiece is shown in the side and front face views. The Y axis is supported in the smart.Turn and DIN programming feature.

Dual-spindle option
For full-surface lathes, the CNC PILOT 640 provides the following features:
- Opposing spindle with second C axis
- Movable tailstock (W axis)

These features are complemented by additional functions such as coordinate transformation, spindle synchronization, and traversing to a stop surface.

* These functions must be implemented in the machine and control by the machine manufacturer.

Coordinate transformation
Contours of workpiece blanks and finished parts can be mirrored about the X axis or shifted relative to the workpiece datum.

Spindle synchronization
Counter spindles are electronically coupled and rotate synchronously. This makes it possible to transfer the workpiece from one spindle to the other while they are rotating, thereby saving time otherwise spent braking and starting the spindles. The CNC PILOT 640 detects any angular offset and compensates for it during subsequent milling on the counter spindle.

Traversing to a fixed stop
To ensure that the workpiece is firmly pressed onto the opposite spindle surface, the control monitors the nominal and actual positions while the longitudinal axis is in motion, thereby detecting the fixed stop. The CNC PILOT 640 monitors the motor torque and uses it to attain the programmed contact force.
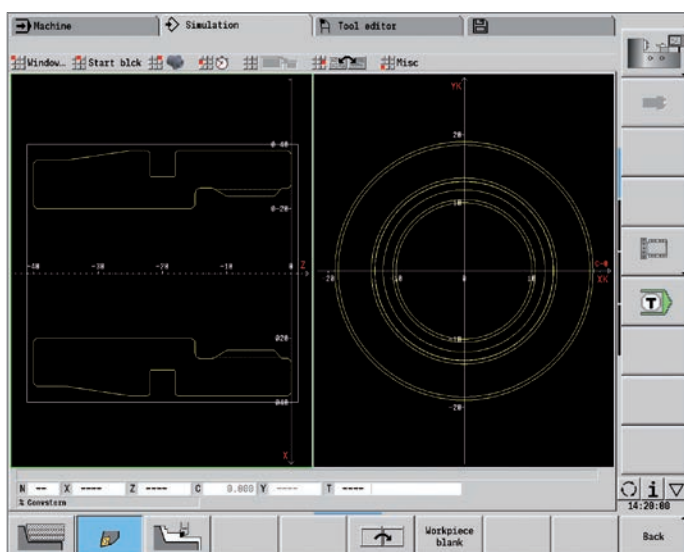
Graphical contour programming for C-axis machining (milling and drilling)
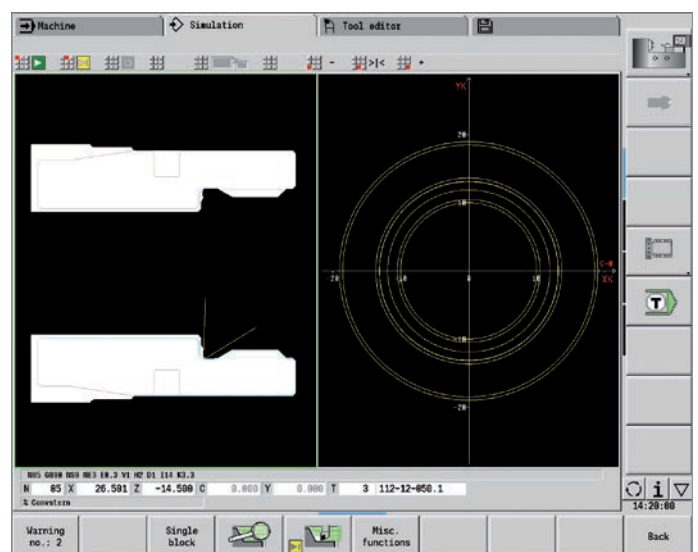
First clamping

Eccentric turning and non-circular turning*

The CNC PILOT 640 offers convenient cycles for eccentric turning and for the manufacture of oval and polygonal parts. In this case—besides the actual contour machining—additional traversing movements of the X and Y axes are superimposed in synchronism with the rotational movement of the spindle. The manufacture of cams and non-circular parts is thereby possible without additional machine elements.

* These functions must be implemented in the machine and control by the machine manufacturer.

Hobbing of external gears

Cycle G808 HOBBING allows you to machine external cylindrical gears or helical gears in any angle. During hobbing, the rotation of the tool and the spindle are synchronized. In addition, the cutter moves along the workpiece axially. Cycle G808

automatically controls these complex movements and allows you to easily and practically enter all of the relevant values. You can use gear parameters directly from your drawing, and the cycle will then use them to calculate the sequence of motion.

Full-surface machining: counter spindle (S3) with C axis (C2) on minor axis (W) and driven tool (S2)
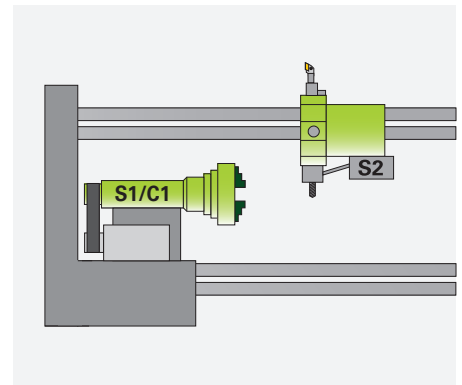
Spindle 1 (S1) with C axis (C1), as well as driven tool (S2)
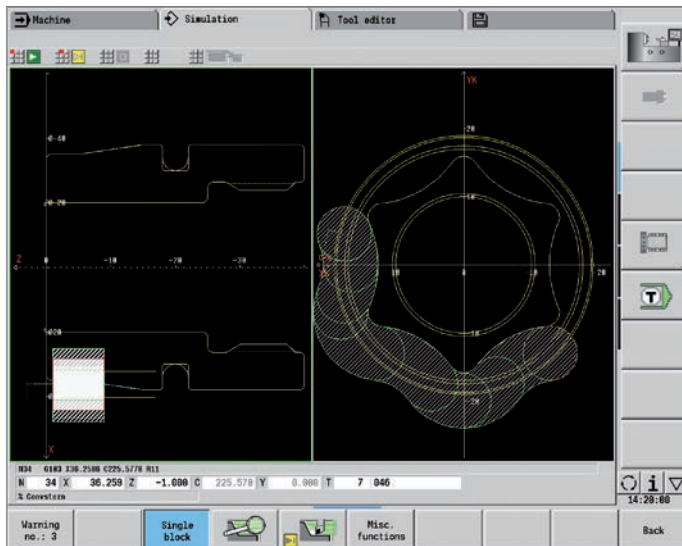
Rear-face machining on the counter spindle after automatic workpiece transfer

Expandable for complex tasks
– Working in a tilted plane with the B axis (option)

The B axis makes it possible to perform drilling and milling operations in oblique planes. At first glance, the programming work for such operations seems very complex and compute-intensive. But with the CNC PILOT 640, you simply tilt the coordinate system to the required position and program the machining operation as usual in the working plane. The machine will then execute the machining operation in the tilted working plane.

The B axis also provides benefits in turning operations. By tilting the B axis and rotating the tool, you can bring it into positions that enable you to use a single tool to perform machining in the longitudinal and transverse directions on the main and counter spindles. You can thereby reduce the number of tools needed as well as eliminate certain tool changes.

Programming
The usual separation of the contour description from the machining operation on the CNC PILOT 640 also applies to milling and drilling operations in a tilted plane.

First, you rotate and shift the coordinate system so that it lies in the tilted plane. Then you describe the hole pattern or the milling contour as you would in the YZ plane. The hole pattern and figure definitions of the CNC PILOT 640 are available. This means that, for linear or circular patterns as well as for simple figures (circles, rectangles, regular polygons, etc.), only a few entries are needed in order to describe the pattern or figure in the tilted plane.

Simulation
In the "side view" window, the simulation shows the hole pattern and milling contour perpendicularly to the tilted plane, and thus without distortion. This ensures the easy verification of programmed hole patterns and milling contours. You can also verify the tool movements in the Side View window. If you want to check the machining operation in the tilted plane with respect to the turning contour or the front face, then add the Lathe Window or Front Window. In the position display (below the simulation window), the CNC PILOT 640 displays the angle of the tilted plane and the tilt angle of the B axis. Do you want to view the active coordinate system? No problem—with a simple keystroke, the CNC PILOT 640 shows the current datum and the direction of the active coordinate system.

Machining in the tilted plane

Flexible use of tools*

If your machine is equipped with a B axis, then you can use your tools much more efficiently than before. On conventional lathes, you need four different tools for longitudinal and transverse turning on main and counter spindles, but you can perform these tasks with just a single tool when you employ a B axis.

You simply tilt the B axis and rotate the tool to the normal position or to the position for machining from behind the workpiece—whichever is required for longitudinal or transverse turning on the main or counter spindle. All you need is a single call: the CNC PILOT 640 calculates the tool lengths, the tool angle, and the other tool data for you.

Tool-use flexibility is increased significantly when multiple tools are mounted in one holder. For example, with a roughing, finishing, and recessing tool you can perform essential portions of turning and recessing operations on a main spindle and counter spindle without changing the tool. And programming is very easy. You simply indicate which tooth of the tool to use and then define the tilting angle and the tool position. This is sufficient because the CNC PILOT 640 already contains the rest position and the data of each tool tooth in its database.

This type of flexibility reduces the number of your tools, and you save on machining time by reducing the number of tool changes.

* The machine and CNC PILOT 640 must be adapted for this function
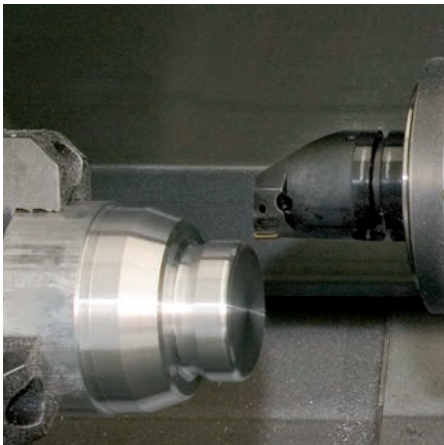
Facing and ...
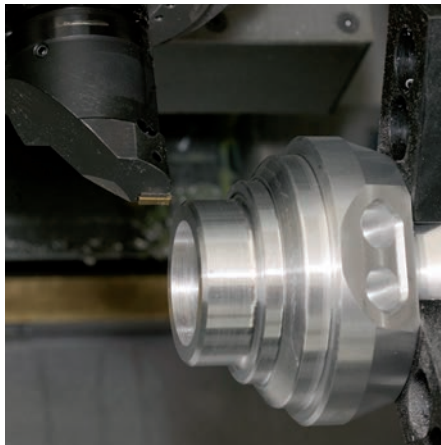
... longitudinal turning with the same tool ...
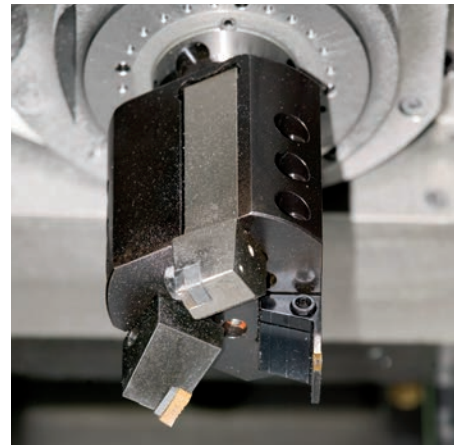
... and with multiple tools on one holder.

25

Powerful Teach-in mode (option)
– Cycles with preprogrammed working steps

Workpiece machining in Teach-in mode
For simple, non-recurring tasks, rework, or thread repair, the cycles of the CNC PILOT 640 simplify your work.

The help graphics illustrate the few entries needed for the cycles. Before cutting, use the simulation to assure yourself that the machining will run as planned.

Thread-recut cycles
Even if the workpiece was unclamped, you can very easily recut a thread with the CNC PILOT 640.

Simply clamp the workpiece, and position the threading tool in the middle of a thread. The CNC PILOT 640 remembers this position as well as the spindle angle. When you position your threading tool in front of the workpiece and enter the other thread parameters, the CNC PILOT 640 has all of the information it needs to recut the thread.

Fewer calculations
In Teach-in mode as well, the CNC PILOT 640 automatically calculates the proportioning of cuts for roughing, recessing, recess turning, or thread cutting, as well as the number of required strokes for deep hole drilling. When turning a taper, you can enter either the starting point and the end point or the starting point and the taper angle—whichever is shown on your drawing.

Continuous availability of tool data
The CNC PILOT 640 makes use of a tool database. Tool data, such as the cutting radius, tool angle, and point angle need be entered only once for the setting dimensions to be determined. The CNC PILOT 640 saves the data. The next time you use the tool, you simply call the tool number. The CNC PILOT 640 automatically adjusts for the correct tool size, and you can immediately work to dimension.

When turning a contour, the CNC PILOT 640 automatically compensates for the deviations that arise from the radius of the tool cutting edge. This increases the precision of your workpiece.
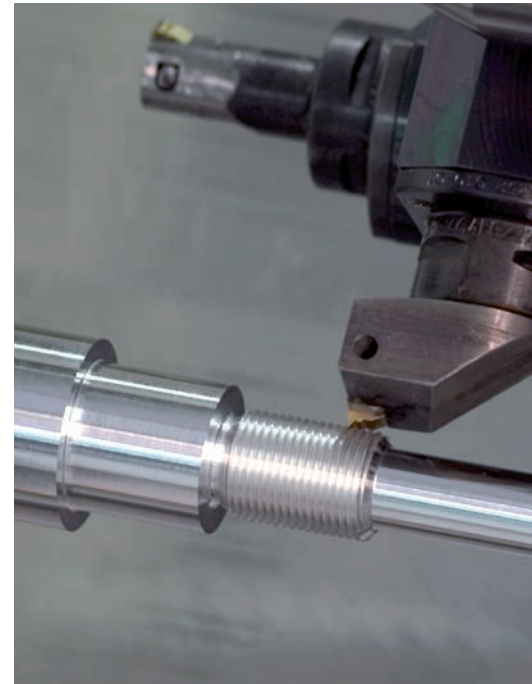
Single- or multi-start longitudinal, taper, or API threads	Longitudinal/transverse cutting for simple contours
Undercuts as per DIN 76, DIN 509 E or DIN 509 F	Longitudinal/transverse cutting with plunging
Undercut form H, form K, or form U	Longitudinal/transverse ICP cutting for any contours
Parting	Longitudinal/transverse ICP cutting for parallel contours

Threads, undercuts, parting

Turning: cutting and finishing

Technology data as suggested values

The CNC PILOT 640 saves your cutting data according to the criteria of workpiece material, tool material, and machining mode. Because you have already entered the cutting material into the tool definition, you need only enter the material of your workpiece. This provides the cycle with all of the data required for suggesting the cutting data.

Reference points

You can define the **workpiece datum** by touching the workpiece with the tool or by entering the datum coordinates.

Approach the **tool-change point** once and store this position. Afterwards, a simple cycle call suffices in order to return to the tool change point.

Protective zone for the spindle

For every traversing movement in the negative Z direction, the CNC PILOT 640 checks whether the programmed protective zone will be violated. If so, it stops the movement and responds with an error message.

Teach-in with contour follow-up

You can activate "contour follow-up" by defining a workpiece blank. This ensures that the current workpiece blank is known to every Teach-in cycle so that air cuts are avoided. This is possible with any kind of turning operation.

Radial/axial recessing for simple contours

Longitudinal/transverse recess turning for simple contours

Radial/axial ICP recessing for any contours

Longitudinal/transverse ICP recess turning for any contours

Recessing and recess turning: cutting and finishing
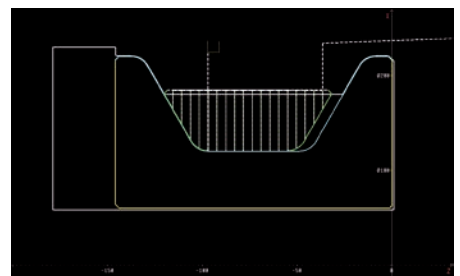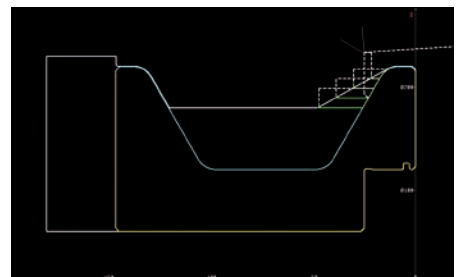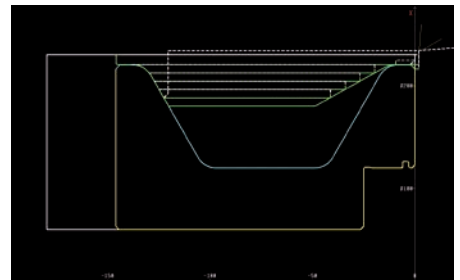
Multi-channel machining with more than one slide
– Convenient programming of complex tasks (option)

Regardless of whether multiple slides are machining a single workpiece or multiple workpieces are being machined simultaneously in the workspace, the challenge for the NC programmer is to optimally divide the machining operation between the available slides and spindles.

Programming
Workpiece machining for complex machines is already included in the command set of the CNC PILOT 640. Specialized four-axis cycles and synchronization commands facilitate the creation of NC programs for machines with multiple slides. The assignment of program sections to slides or spindles also gives the NC program a clear structure.

By mirroring and/or moving the contour, and by converting special NC commands, it is also easy to program turning operations that use the counter spindle, as well as rear-side drilling and milling operations—for nearly any machine design.

Simulation
The graphical simulation shows all of the workpieces and the tool movements of all slides. The CNC PILOT 640 takes the entire work envelope into consideration. The tools and chucking equipment are shown to scale.

Furthermore, the graphical simulation is characterized by a high degree of flexibility. You define what is displayed in the simulation window. You can specify whether the window for lathe, front-face or lateral machining is shown, or a combination of these windows. You can also specify whether only the NC blocks of a certain slide are displayed, or the NC blocks of all slides.

With this support, you can effectively and thoroughly check complex, multi-slide programs before making the first cut.
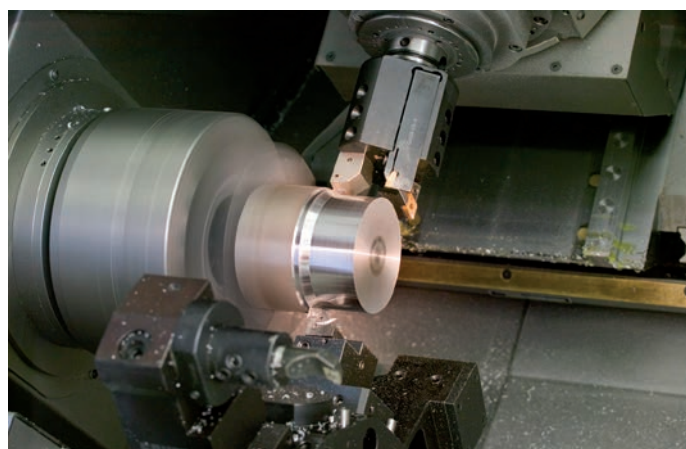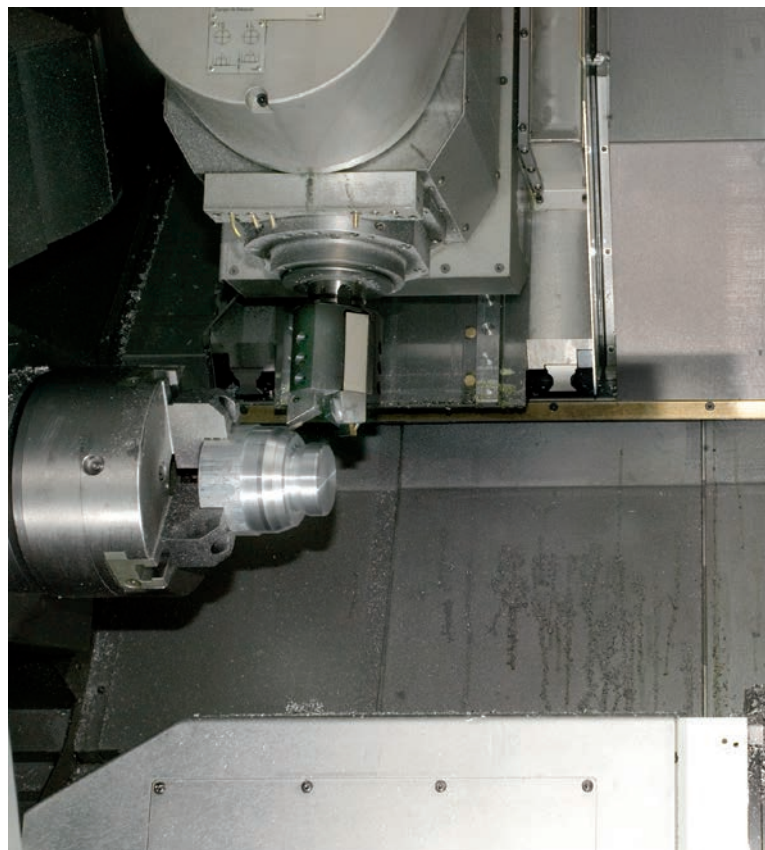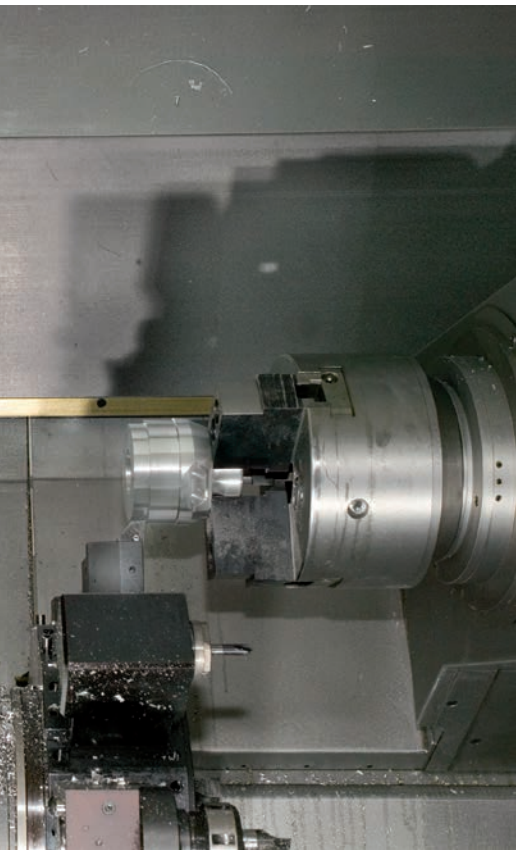
Synchronous point analysis

Synchronous point analysis shows the chronological sequence of the workpiece machining operation and the interdependency of the slides.

During the simulation, the CNC PILOT calculates the productive, non-productive, and idle times, as well as all tool changes and synchronization points. Synchronous point analysis graphically depicts the times, tool changes, and synchronous points for each slide.

The sequence of the workpiece machining steps thereby becomes clear. This is a good basis upon which the NC programmer can analyze and optimize the machining operation.
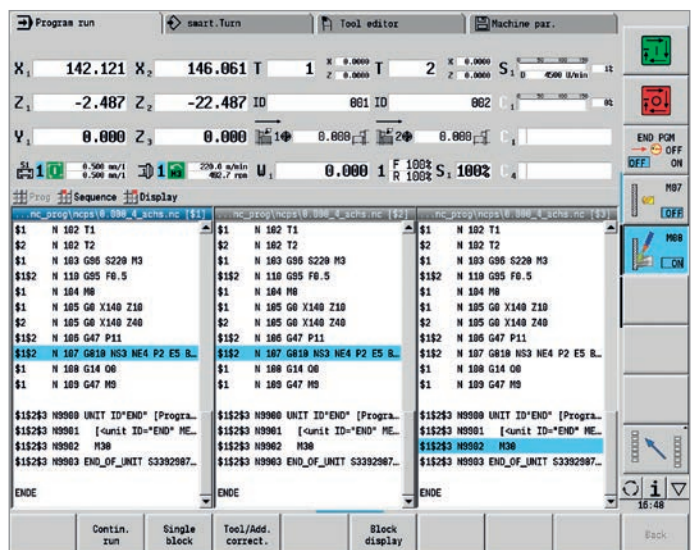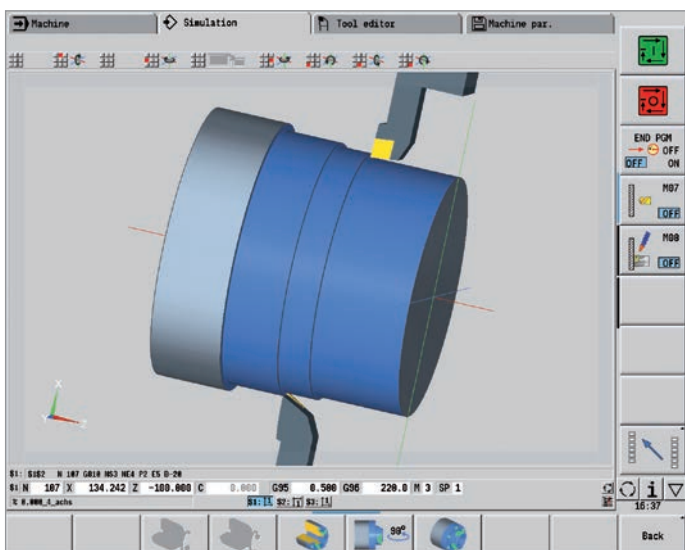
Rapid availability of tool data and cutting data
– The tool database and technology database of the CNC PILOT 640

Tool database

The standard tool database of the CNC PILOT 640 stores 250 tools. The tool database can be expanded to contain 999 tools (option).

The CNC PILOT 640 differentiates between various types of turning, drilling, and milling tools. The data that are input vary depending on the tool type. In this way, you can be sure that all of the important parameters are specified despite reduced data input.

The tool data are entered with the help of dialog prompts in which you enter parameters such as the cutting-edge radius, tool angle, point angle, cutting material, and tool description. The data to be input is explained by context-sensitive help graphics.

Tool list

The CNC PILOT 640 shows all of the tools in a straightforward tool list. Various sorting criteria help you to quickly find the desired tool. This list not only gives you a good overview of your tools—it is also the basis for transferring tool data during manual machining and during the creation of NC programs.

Wear compensation

The CNC PILOT 640 offers a simple way of compensating for tool wear in both the X and the Z axes. You can enter the compensation values at any time, even while the workpiece is being machined.

Tool measurement

The CNC PILOT 640 offers various possibilities for the measurement of tools directly on the machine:
- By "touching off" the workpiece
- With the aid of an optical gauge*: the tool is manually traversed to the crosshairs of the optical measuring device, and the value is saved with a keystroke.
- Through a tool touch probe* (option): the tool moves in the direction of measurement. The tool setting dimension is ascertained and adopted when the tool touch probe releases a trigger signal (e.g., the TT 160 touch trigger probe with cuboid probe contact).

You can determine the tool data very conveniently, reliability, and accurately by using an optical gauge or tool touch probe for tool measurement.

* The machine and the CNC PILOT 640 must be adapted to this function by the machine manufacturer.
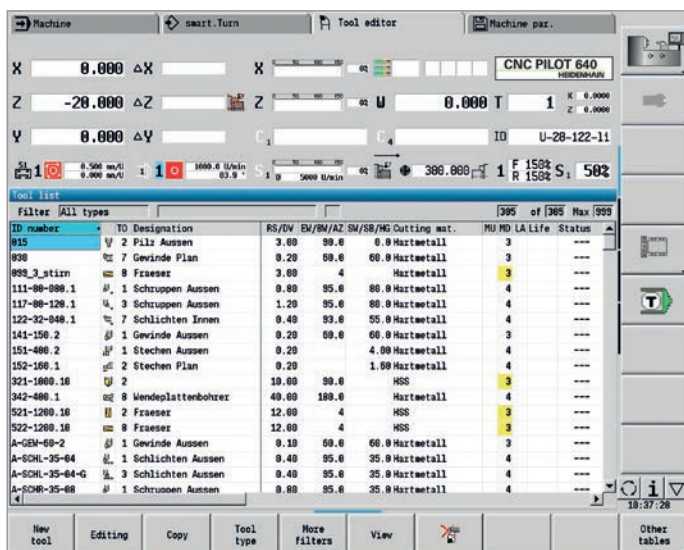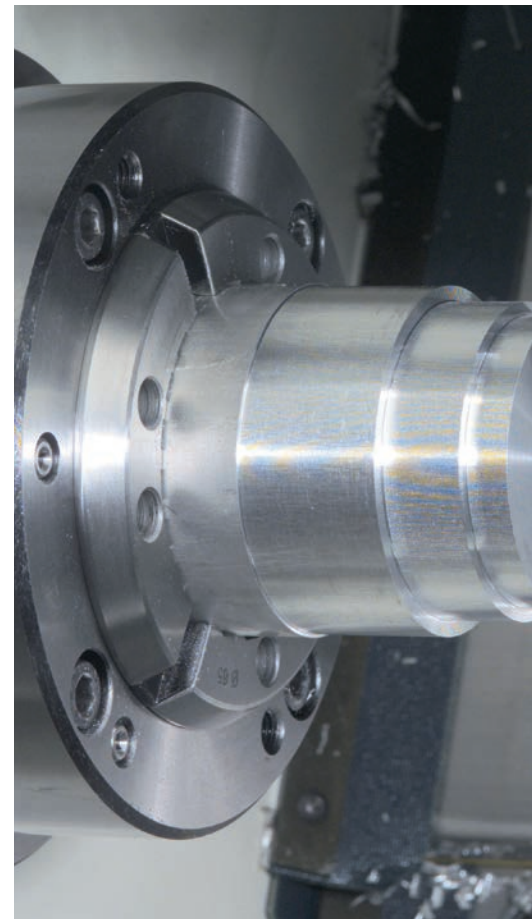
Tool management in the tool list
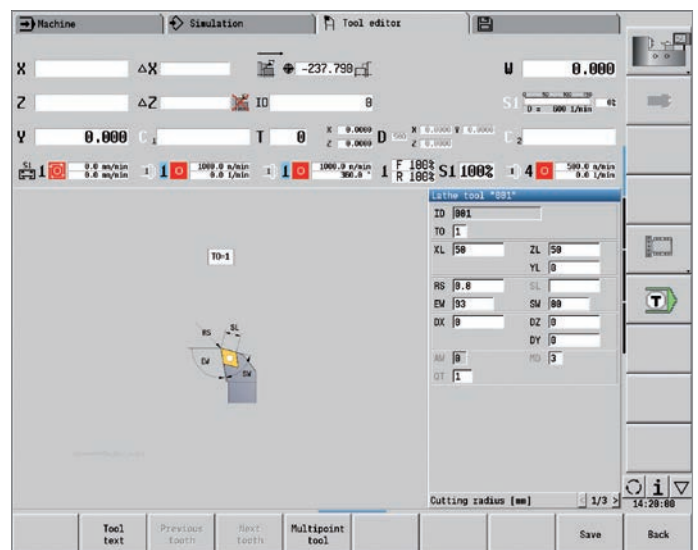
Tool editor
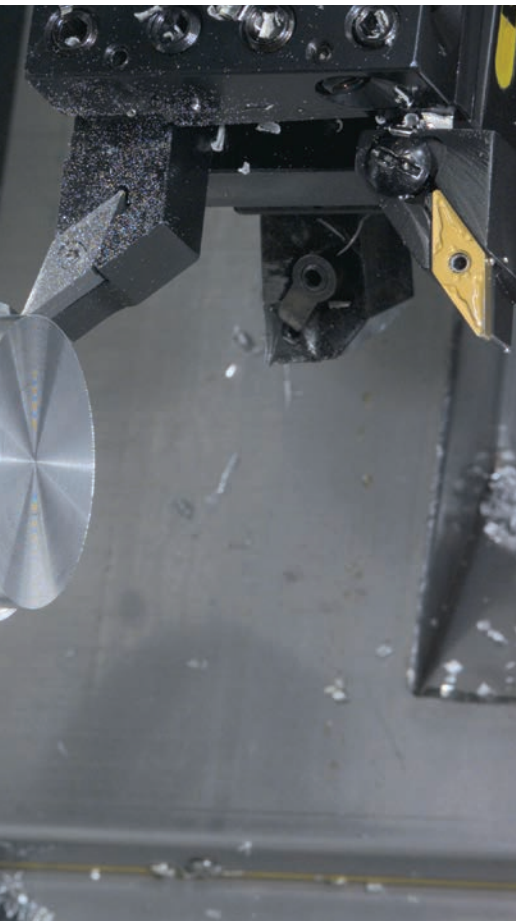
Turret assignment

You can view your machine's programmed turret assignment with all of the important tool parameters at any time.

If you want to change the tool assignment or the tools in the turret, you can additionally display the tool database entries in the lower window. Now you need only select the desired turret pocket and the proper tool from the database. You can apply the tool data to the turret assignment entry at the stroke of a key.

Tool life monitoring (option)

With smart.Turn and DIN PLUS programs, in addition to the simple tool life monitoring feature, you can also use the "Tool Life Monitoring with Sister Tool" option. The CNC PILOT 640 then automatically inserts a replacement tool as soon as the active tool is worn. When the final tool in the replacement chain is worn, the CNC PILOT 640 aborts program execution.

Technology Data (option)

With the CNC PILOT 640, you need to enter the cutting data only once. The control saves the data according to the criteria of the workpiece material, cutting material, and machining mode. Thanks to this three-dimensional table, the control always knows the correct feed rate and the correct cutting speed.

The CNC PILOT 640 determines the machining mode from the Teach-in cycle or the unit. The cutting material is entered for the tool description. Now you need only define the workpiece material at the beginning of the cycle program or the smart.Turn program, and the CNC PILOT 640 will provide the correct suggested values for your machining operation. You can use the cutting values or adjust them as required.

In its standard version, you can store the cutting data for nine workpiece-material / tool-material combinations in the technology database of the CNC PILOT 640. It can be expanded to a total of 62 combinations (option). Each workpiece-material / tool-material combination includes the cutting speed, the main and secondary feed rates, and the infeed for 16 machining modes.
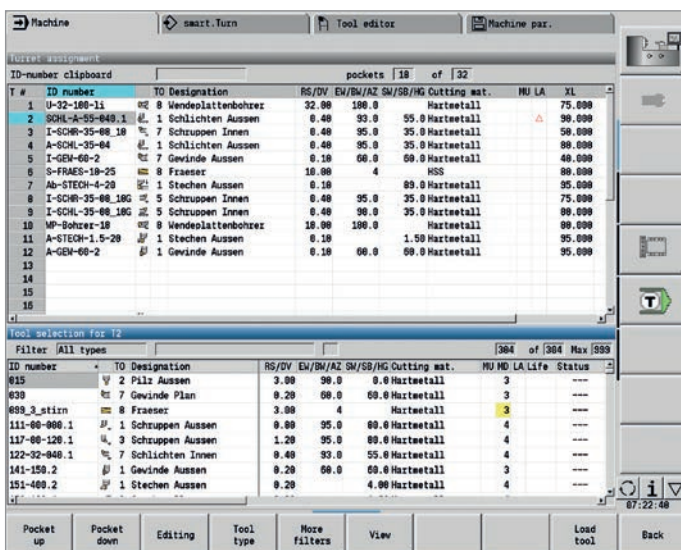
Tool selection for turret assignment
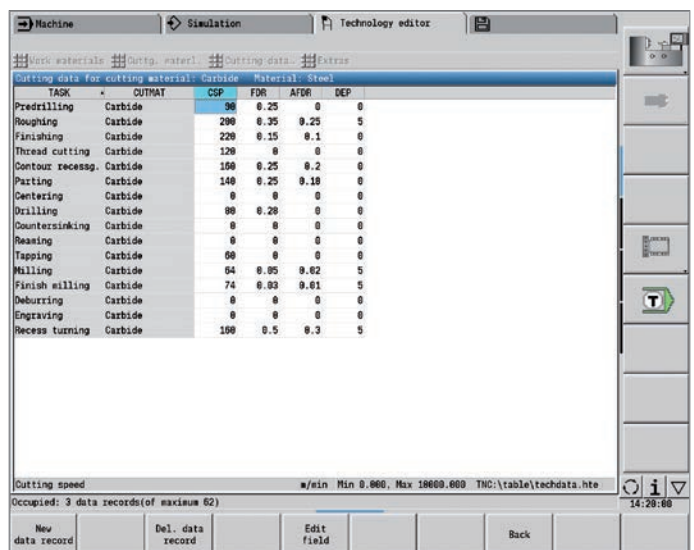
Input of cutting values in the technology editor

Open for communication

– Uniformly digital job management with Connected Machining

connected + machining

The effective transfer of knowledge contributes decisively to the success of any company. To transfer knowledge quickly and without loss, effective communication via e-mail is just as much a matter of course as the universal availability of electronic production documents or the transfer of data to inventory management systems and production activity control systems. The tools and raw materials in stock, tool data, fixture setups, CAD data, NC programs, and inspection instructions must be available to machine operators in all shifts. Economical manufacturing therefore demands an efficiently working process chain and a networked control.

The CNC PILOT 640, with its **Connected Machining** package of functions, integrates itself flexibly into your process chain and helps you optimize the transfer of knowledge within your company. So let the shop, as well, use all of the information available within the company. **Connected Machining** makes uniformly digital job management possible within a networked manufacturing environment. You thus benefit from the following:
• Easy data usage
• Time-saving procedures
• Transparent processes

The networked CNC PILOT 640
You can integrate the CNC PILOT 640 with the functions of **Connected Machining** into your corporate network and, via the control, connect the shop to PCs, programming stations, and other data storage devices in the following departments:
• Design
• Programming
• Simulation
• Production planning
• Production

Even in its standard version, the CNC PILOT 640 features a Gigabit Ethernet interface of the latest generation. The CNC PILOT 640 communicates with NFS servers and Windows networks in TCP/IP protocol without requiring additional software. The fast data transfer at rates of up to 1000 Mbit/s ensures very short transfer times. The CNC PILOT 640 therefore meets the ideal technological requirements for **Connected Machining,** the networking of the control in the shop with all product-related areas in your company.
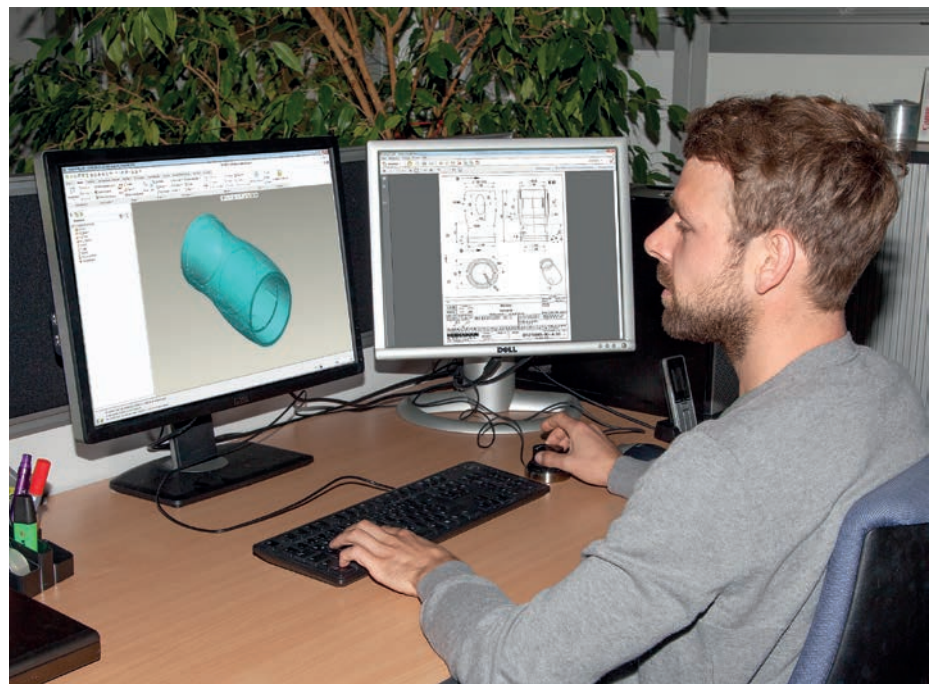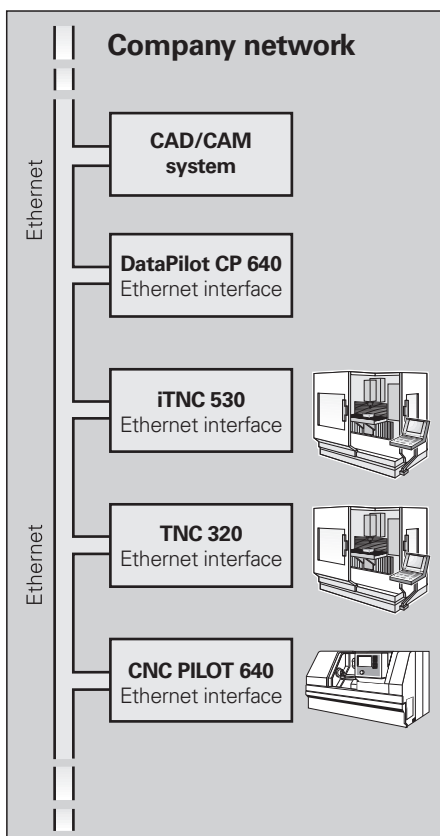
Standard range of functions

In order for you to be able to use the data that you have transferred to the CNC PILOT 640 over the standard network connection, the CNC PILOT 640 likewise offers attractive applications, even as part of its standard range of functions. A CAD viewer, PDF viewer, or the web browser Mozilla Firefox makes the simplest form of **Connected Machining** possible: access to manufacturing process data right on the control. Thus, using web-based documentation software or ERP systems is as equally possible as accessing your e-mail inbox. The following file formats can also be opened directly on the CNC PILOT 640:

• Text files ending with .txt or .ini
• Graphic files ending with .gif, .bmp, .jpg, or .png
• Table files ending with .xls or .csv
• HTML files

Data transfer with Connected Machining

An additional solution for uniformly digital job management within the framework of **Connected Machining** is the free **TNCremo** PC software. With it, you can

• transfer remotely stored part programs and tool or pallet tables bidirectionally, and
• start the machine.

With the powerful **TNCremoPlus** PC software, you can also transfer the screen contents of the control to your PC using the live-screen function.

Use job-related data on the control

With **option 133 "Remote Desktop Manager"** you can operate a Windows PC from the CNC PILOT 640. On the control, you gain direct access to the IT systems of the process chain, and you can benefit from considerably more efficient setup processes by eliminating the burdensome legwork between the machine and the front office. Technical drawings, CAD data, NC programs, tool data, work instructions, parts lists, and warehouse information are digitally available at the machine. E-mails can be sent and received with ease. With a simple keystroke on the machine operating panel, you can switch between the control screen and the screen of the Windows PC. This PC can be a computer in the local network or an industrial PC (IPC) in the machine's electrical cabinet.

Detailed data for the optimal organization of the production process

HEIDENHAIN DNC* enables, among other things, the connection of CNC PILOT controls to inventory management systems and production activity control systems. For example, this interface can be used for the configuration of automatic feedback messages about active production processes.

* The machine must be prepared by the machine manufacturer for this function.

With the **StateMonitor*** PC software, Connected Machining enables access to the status of the current machining operation. You can use the software with any device that has a web browser. This allows you to use StateMonitor not only on your control or PC but also on your smartphone or tablet. With the well-arranged display you can quickly gain an overview of the current machine status or see whether any machine messages are pending. This allows you to react immediately and take appropriate action. You can also easily configure StateMonitor to send you an e-mail for specific events, such as at the end of a program, for a machine stop, or for a servicing message.

* Option 18 required

Open for communication
– The DataPilot CP 640 programming station

DataPilot CP 640 is the PC programming station for the CNC PILOT 640 and the organizational system for the shop and supervisor's office.

This is why DataPilot CP 640 is the ideal supplement to the CNC PILOT 640 for program creation and archiving, and for basic and advanced training.

Creating programs

Programming, testing, and optimizing smart.Turn or DIN PLUS programs with the programming station significantly reduces machine idle times. This does not require an adjustment in thinking, because, with DataPilot, programming and testing are performed in the same manner as on the lathe. DataPilot is based on the same software that is on the control. This also gives you the assurance that a program created with DataPilot can be run on the machine immediately.

Archiving programs

Even though the CNC PILOT 640 features a large memory capacity, you should also back up your programs on an external system. The CNC PILOT 640 features a USB port and an Ethernet interface. This gives you everything you need in order to integrate the CNC PILOT 640 into your existing network or to connect the DataPilot PC directly to the control.

Convenient program transfer functions support both programming and archiving on the DataPilot PC.

Training with DataPilot CP 640

Since DataPilot CP 640 is based on the same software as that of the CNC PILOT 640, it is optimally suited for training purposes. Programming and program testing on the DataPilot PC are performed in exactly the same way as on the machine. DataPilot even simulates setup functions such as defining the workpiece datum, tool measurement, or the execution of individual cycles, smart.Turn programs, or DIN PLUS programs. This gives trainees the confidence they will need for later work on the machine.

System requirements

The DataPilot software can run on PCs with the Windows 7, 8, 8.1, or Windows 10 (64-bit) operating systems.
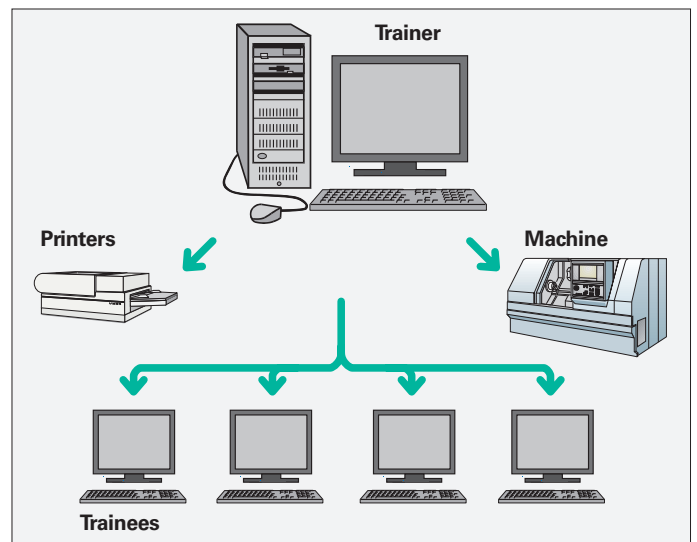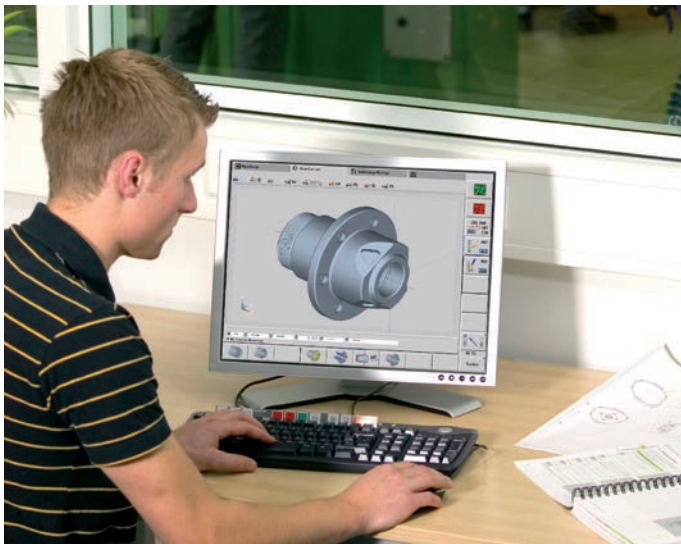
– Rapid availability of all information

Do you have questions about a programming step, but your User's Manual is not at hand? No problem: the CNC PILOT 640 numerical control and DataPilot CP 640 programming station feature TURNguide, a convenient help system that can display the user documentation in a separate window.

You activate TURNguide by simply pressing the Info key on the keyboard.

In most cases, TNCguide directly displays the information in the proper context (context-sensitive help). This means that you immediately receive the information that you need. This function is particularly helpful for the programming of cycles. The respective operating method is explained as soon as you press the Info key in the opened dialog.

You can download the documentation in the desired language from the HEIDENHAIN homepage to the corresponding language directory on your control.

The following manuals are available in the help system:
- CNC PILOT 640 user's manual
- smart.Turn and DIN programming user's manual
- DataPilot CP 640 user's manual (installed only with a programming station)
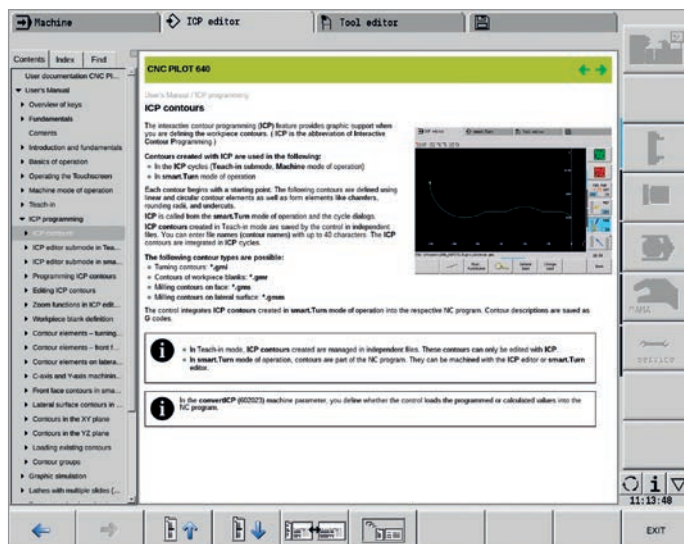
TURNguide integrated into the control (e.g., on the CNC PILOT 640 ...
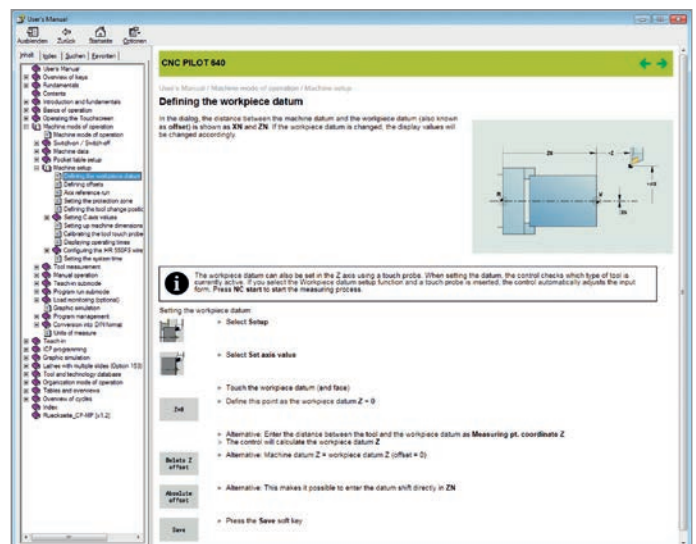
… or at the programming station).

Workpiece measurement
– Setup, preset setting, and measuring with touch trigger probes

Inspecting workpieces for completed machining and dimensional accuracy
The CNC PILOT 640 features measuring cycles for checking the geometry of the machined workpieces. For the measuring cycles, you simply insert a 3-D touch probe from HEIDENHAIN into the turret instead of a tool:
- Check whether all of the machining operations have been properly performed
- Determine infeeds for finishing
- Detect and compensate for tool wear
- Inspect workpiece geometry and sort parts
- Create measurement reports
- Ascertain the machining error trend

Workpiece touch probes from HEIDENHAIN help you reduce costs in the shop and in serial production: in conjunction with probing functions of the CNC PILOT 640, setup, measurement, and inspection functions can be performed automatically.

The stylus of a TS touch trigger probe is deflected upon contact with a workpiece surface. The TS generates a trigger signal that, depending on the model, is transmitted to the control either by cable or by an infrared or radio beam.

HEIDENHAIN touch probes* for workpiece measurement are available in different versions. The ball tips, which are made of ruby, are available in different diameters with various stylus lengths.

* The touch probes must be interfaced to the CNC PILOT 640 by the machine manufacturer.

Touch probes with a **cable connection for signal transmission** for machines with manual tool changing, as well as for grinding machines and lathes:

TS 260 – new generation, axial or radial cable connection

TS 248 – like the TS 260, but with reduced deflection forces

Touch probes with **radio or infrared signal transmission** for machines with an automatic tool changer:

TS 460 – new generation standard touch probe for radio and infrared transmission, with compact dimensions
TS 740 – high probing accuracy and repeatability, low probing force, with infrared transmission

📖 **Further information:**
Detail descriptions of the workpiece touch probes are available on the Internet at *www.heidenhain.de* or in the *Touch Probes for Machine Tools* brochure.

TS 460

Tool measurement
– Measuring the length, radius, and wear directly in the machine

Exact measurement of the tool dimensions is a decisive factor in ensuring a consistently high level of production quality. The TT tool touch trigger probes from HEIDENHAIN serve this purpose.

TT 160 – new generation, signal transmission to the NC via a cable
TT 460 – new generation, signal transmission via radio and infrared beam to the transceiver

The SE 660 is a common transceiver for tool and workpiece touch probes with radio or infrared transmission.

The TT 160 and TT 460 are 3-D touch trigger probes for tool measurement and inspection. The disk-shaped probe contact of the TT is deflected during the physical probing of a tool. During probing, the TT generates a trigger signal that is transmitted to the control for further processing. The trigger signal is generated by means of a wear-free and highly reliable optical sensor.

With their rugged design and high protection rating, these tool touch probes can be installed directly within the machine tool's work envelope, thereby enabling tool measurement directly within the machine. You can thus determine the tool dimensions quickly, easily and, above all, very precisely. The TT tool touch probes from HEIDENHAIN are the ideal addition for improving the efficiency and quality of your production system.

SE 660

TT 160

📖 **Further information:**

Detailed descriptions of tool touch probes are available on the Internet at *www.heidenhain.de* or in the *Touch Probes for Machine Tools* brochure.

Overview
– User functions

User functions	Standard	Option	
Configuration	•		Basic version: X and Z axis, main spindle
		0-6	Driven tool and auxiliary axes (U, V, W)
		55+0-6	C axis and driven tool
		70+0-6	Y axis
		54+0-6	B axis
		94+0-6	Parallel axes U, V, W (display function and compensation)
		132+0-6	Counter spindle
	•		Digital current and speed control
Operating modes			
Manual operation	•		Manual slide movement through axis-direction keys, an intermediate switch, or electronic handwheels
	•		Graphical support for entering and running cycles without storage of the machining steps, in direct alternation with manual machine operation
		11	Thread reworking (thread repair) in a second workpiece setup
Teach-in mode		8	Sequential arrangement of fixed cycles, with each cycle being run immediately after input or graphically simulated and subsequently saved
Program run	•		In both Single Block mode and Full Sequence mode:
			DIN PLUS programs
		9	smart.Turn programs
		8	Cycle programs
Setup functions	•		Workpiece datum setting
	•		Definition of tool-change position
	•		Definition of protection zone
	•		Definition of machine dimensions
	•		Manual programs
	•		Tool measurement by touching the workpiece
		17	Tool measurement with a TT tool touch probe
		17	Tool measurement with an optical gauge
		17	Workpiece measurement with a TS workpiece touch probe
Programming			
Cycle programming		8	Turning cycles for simple and complex contours, and contours described with ICP
		8	Contour-parallel turning cycles
		8	Recessing cycles for simple or complex contours, as well as contours defined with ICP
		8	Repetitions with recessing cycles
		8	Recess turning cycles for simple and complex contours, and contours described with ICP
		8	Undercut and parting cycles
		8	Engraving cycles
		8	Threading cycles for single or multi-start longitudinal, taper, or API threads, and threads with variable pitch
		8+55	Cycles for axial and radial drilling, deep-hole drilling, and tapping operations with the C axis
		8+55	Thread milling with the C axis
		8+55	Axial and radial milling cycles for slots, figures, single surfaces, and centric polygon surfaces, and for complex contours described with ICP for machining with the C axis
		8+55	Helical slot milling (multi-start) with the C axis
		8+55	Deburring of ICP contours
		8+55	Linear and circular patterns for drilling and milling operations with the C axis
		8	Context-sensitive help graphics
		8	Transfer of cutting values from the technology database
		8	Use of DIN macros in cycle programs
		8+9	Conversion of cycle programs to smart.Turn programs

User functions	Standard	Option	
Interactive Contour Programming (ICP)		8/9	Contour definition with linear and circular contour elements
		8/9	Immediate display of entered contour elements
		8/9	Calculation of missing coordinates, intersections, etc.
		8/9	Graphical display of all solutions for selection by the user if more than one solution is possible
		8/9	Chamfers, rounding arcs, and undercuts available as form elements
		8/9	Input of form elements immediately during contour creation or through later superimposition
		8/9	Changes to existing contours can be programmed
		8/9	Machining attributes available for individual contour elements
		8/9+55	C-axis machining on the front face and lateral surface: • Description of individual holes and hole patterns (only in smart.Turn) • Description of figures and figure patterns for milling (only in smart.Turn) • Creation of freely definable milling contours
		9+70	Y-axis machining on the XY and ZY planes (only in smart.Turn): • Description of individual holes and hole patterns • Description of figures and figure patterns for milling operations • Creation of freely definable milling contours
		8/9+55+ 70+132	Programming of the rear face for full-surface machining with the C and Y axes
		8/9+42	DXF import: importing of contours for turning and milling operations
smart.Turn programming		9	The basis is the "unit," which is the complete description of a machining block (geometry, technology, and cycle data)
		9	Dialog boxes divided into overview and detail forms
		9	Fast navigation between the forms and the input groups via the "smart" keys
		9	Context-sensitive help graphics
		9	Start unit with global settings
		9	Transfer of global values from the start unit
		9	Transfer of cutting values from the technology database
		9	Units for all turning and recessing operations for simple contours and ICP contours
		9+55/70	Units for drilling and milling operations with the C or Y axis for holes, milling contours, and drilling and milling patterns that are simple or that have been programmed with ICP
		9+55/70	Units for trochoidal milling operations for pockets and slots
		9+55+ 135	Unit for hobbing for the machining of gear teeth
		9+55	Special units for activating/deactivating the C axis, as well as for subroutines and section repeats
		9+55/70	Verification graphics for the workpiece blank and finished part, as well as for C- and Y-axis contours
		9	Turret assignment and other setup information in the smart.Turn program
		9	Parallel programming
		9	Parallel simulation
TURN PLUS		63	Automatic smart.Turn program generation with: • Automatic tool selection • Automatic turret assignment • Automatic calculation of the cutting data • Automatic generation of the machining sequence in all working planes, including for C-axis machining (with option 55) and Y-axis machining (with option 70) • Automatic cutting limit through chucking equipment • Automatic generation of rechucking work blocks for turning with full-surface machining • Automatic generation of work blocks for rear-face machining (with option 132)

User functions	Standard	Option	
DIN PLUS programming	•		Programming in DIN 66025 format
	•		Expanded command format (IF ... THEN ... ELSE ...)
	•		Simplified geometry programming (calculation of missing data)
	•		Powerful fixed cycles for area clearance, recessing, recess turning, and thread machining
		55	Powerful fixed cycles for drilling and milling with the C axis
		70	Powerful fixed cycles for drilling and milling with the Y axis
	•		Subprograms
			Technology functions for full-surface machining:
	•		• Traversing to a fixed stop
	•		• Controlled parting
		131/132	• Spindle synchronization
		132	• Mirroring and converting
			• Mechatronic tailstock
	•		Programming with variables
	•	8/9	Contour description with ICP
	•		Program verification graphics for workpiece blank and finished part
	•		Turret assignment and other setup information in the DIN PLUS program
		9	Conversion of smart.Turn units into DIN PLUS command sequences
	•		Parallel programming
	•		Parallel simulation
Simulation	•		Graphical simulation of the cycle process, the cycle program, the smart.Turn program, or the DIN PLUS program
	•		Depiction of the tool paths as pencil-trace or cutting-path graphics; special identification of the rapid traverse paths
	•	55	Machining simulation (2-D material-removal graphic)
			Turning view or front face view, or depiction of the unrolled cylindrical surface for verification of C-axis machining operations
	•	54	Display of programmed contours
			View of the tilted plane (B-axis machining)
	•		View of front face and YZ plane for verification of Y-axis machining
	•	132	Three-dimensional rendering of the workpiece blank and finished part
			Simulation of mirrored contours for rear-face machining
	•		Shifting and magnifying functions
	•		Block scan in the simulation
	•		Depiction of multi-channel machining operations
B-axis machining		54	Machining with the B axis
	•		Tilting the working plane
		54	Rotating the machining position of the tool
Eccentric machining		135	Cycles for eccentric turning and for the manufacture of oval and polygonal contours
		135	Superimpositioning of traversing movements of the X and Y axes synchronously to the rotational motion of the spindle
Machining time analysis	•		Calculation of machining times and idle times
	•		Consideration of switching commands triggered by the CNC
	•		Representation of time per individual cycle or per tool change
Monitoring function		151	Load monitoring: detection of tool wear and breakage during machining

User functions	Standard	Option	
Tool database	•		For 250 tools
		10	For 999 tools
	•		Tool description can be entered for every tool
	•		Automatic inspection of the tool-tip position relative to the machining contour
	•		Compensation of the tool-tip position in the X/Y/Z plane
	•		Fine compensation of tool via handwheel with application of compensation values in the tool table
	•		Automatic tool-tip and cutter radius compensation
	•		Tool monitoring for the service life of the indexable insert or of the number of produced workpieces
		10	Tool monitoring with an automatic tool change after expiration of tool life
	•		Management of multipoint tools (multiple inserts or reference points)
	•		Support of quick-change tool systems
Technology database		8/9	Access to cutting data upon definition of workpiece material, cutting material, and the machining mode. The CNC PILOT 640 distinguishes between 16 machining modes. Each workpiece-material / tool-material combination includes the cutting speed, the main and secondary feed rates, and the infeed for 16 machining modes.
		8/9	Automatic determination of the machining modes from the cycle or the machining unit
		8/9	The cutting data are entered in the cycle or in the unit as suggested values
		8/9	Nine workpiece-material / tool-material combinations (144 entries)
		10	62 workpiece-material / tool-material combinations (992 entries)
User management	•		Configurable linkage of permissions to user roles • Logging on to the control with a user account • User-specific HOME folder for simplified data management • Role-based access to the control and network data
Conversational languages	•		English, German, Czech, French, Italian, Spanish, Portuguese, Dutch, Swedish, Danish, Finnish, Norwegian, Slovenian, Slovak, Polish, Hungarian, Russian (Cyrillic), Romanian, Turkish, Chinese (traditional and simplified), Korean

Overview
– Options

Option number	Option	As of NC software 688946- 688947-	ID	Comment
0 1 2 3 4 5 6 7	Additional Axis	01 01 01 01 01 01 01 03	354540-01 353904-01 353905-01 367867-01 367868-01 370291-01 307292-01 370293-01	**One to eight additional control loops**
8	Teach-in	01	632226-01	**Cycle programming** • Contour description with ICP • Cycle programming • Technology database with nine workpiece-material / tool-material combinations
9	smart.Turn	01	632227-01	**smart.Turn** • Contour description with ICP • Programming with smart.Turn • Technology database with nine workpiece-material / tool-material combinations
10	Tools and Technology	01	632228-01	**Tools and technology** • Tool database expanded to 999 entries • Technology database expanded to 62 workpiece-material / tool-material combinations • Tool life monitoring with exchange tools
11	Thread Recutting	01	632229-01	**Thread** • Thread recutting • Handwheel superimpositioning during thread cutting
17	Touch Probe Functions	01	632230-01	**Tool measurement and workpiece measurement** • Determining tool-setting dimensions with a tool touch probe • Determining tool-setting dimensions with an optical gauge • Automatic workpiece measurement with a workpiece touch probe
18	HEIDENHAIN DNC	01	526451-01	Communication with external PC applications over COM component
24	Gantry Axes	01	634621-01	Gantry axes in master-slave torque control
42	DXF Import	01	632231-01	**DXF import:** import of DXF contours
46	Python OEM Process	01	579650-01	Python application on the CNC PILOT 640
49	Double-Speed Axes	01	632223-01	Short control-loop cycle times for direct drives
54	B-Axis Machining	01	825742-01	**B axis:** tilting the working plane, rotating the machining position of the tool
55	C-Axis Machining	01	633944-01	**C-axis machining**
63	TURN PLUS	01	825743-01	**TURN PLUS:** automatic generation of smart.Turn programs
70	Y-Axis Machining	01	661881-01	**Y-axis machining**
77	4 Additional Axes	03	634613-01	**4 additional control loops**
78	8 Additional Axes	03	634614-01	**8 additional control loops**

Option number	Option	As of NC software 688946- 688947-	ID	Comment
94	Parallel Axes	01	679676-01	**Support of parallel axes (U, V, W)** Combined display of principal axes and secondary axes
131	Spindle Synchronism	01	806270-01	**Synchronization** (of two or more spindles)
132	Counter Spindle	01	806275-01	**Counter spindle** (spindle synchronism, rear-face machining)
133	Remote Desktop Manager	04	894423-01	Display and remote operation of external computer units (e.g., a Windows PC)
135	Synchronizing Functions	03	1085731-01	Expanded synchronization of axes and spindles
137	State Reporting	06	1232242-01	State Reporting Interface (SRI): provision of operating conditions
143	Load Adapt. Control	01	800545-01	LAC: load-dependent adaptation of control parameters
151	Load Monitoring	03	1111843-01	Monitoring of the tool load
153	Multi-channel	05	1217032-01	*Multi-channel capability:* up to three channels for asynchronous multi-slide machining

– Accessories

Accessories	
Electronic handwheels	• **HR 180** panel-mounted handwheels with connection at position inputs, plus • One **HR 130** serial, panel-mounted handwheel or an **HR 510 (FS), HR 520 (FS), or HR 550 FS** portable serial handwheel
Tool measurement	3-D touch trigger probes: • **TT 160** with a cuboid probe contact and cable connection • **TT 460** with a cuboid probe contact and radio or infrared transmission
Workpiece measurement	• **TS 260:** touch trigger probe with cable connection, or • **TS 460:** touch trigger probe with radio or infrared transmission, or • **TS 740:** touch trigger probe with infrared transmission
DataPilot CP 640, MP 620	PC programming station for programming, archiving, and training for the CNC PILOT 640 • Full version with license for single station or multiple stations • Demo version (free of charge)
Software for PCs	• **TeleService:** software for remote diagnostics, monitoring, and operation • **TNCremo:** software for data transfer—free of charge • **TNCremoPlus:** software for data transfer with live-screen function

Overview
– Specifications

Specifications	
Components	MC main computer and 19-inch screen (1280 x 1024 pixels) for multi-touch operation, or MC main computer with 15.6-inch screen (1366 × 768 pixels) or 19-inch screen (1280 × 1024 pixels) for multi-touch operation CC, UEC, or UMC controller unit TE operating panel (19-inch or 15.6-inch) with the following: • Integrated machine operating panel • Feed-rate and spindle override potentiometer • 19-inch version: with or without electronic handwheel
Operating system	HEROS 5 real-time operating system for machine control
NC program memory	1.8 GB (on CFR compact flash memory card)
Input resolution and display step	X axis: 0.5 µm, diameter: 1 µm U, V, W, Y, Z axis: 1 µm B, C1/C2 axis: 0.001°
Interpolation	Straight line: in two principal axes (max. ±100 m), optional in 3 principal axes Circle: in 2 axes (radius max. 999 m), optional additional linear interpolation of the third axis C1/C2 axis: interpolation of X and Z linear axes with the C1/C2 axis (option) B axis: 5-axis interpolation between X, Z, Y, B, and C axes (option)
Feed rate	mm/min or mm/revolution Constant surface speed Max. feed rate (60 000/pole pair number × spindle pitch) with f_{PWM} = 5000 Hz
Main spindle	Maximum of 60 000 rpm (with option 49, max. 120 000 rpm) for motors with one pole pair
Axis feedback control	Integrated digital drive control for synchronous and asynchronous motors Position loop resolution: signal period of the position encoder/1024 Cycle time of position controller: 0.2 ms Cycle time of speed controller: 0.2 ms Cycle time of current controller: at least 0.05 ms
Error compensation	Linear and nonlinear axis error, backlash, reversal peaks during circular movements Static friction
Data interfaces	Gigabit Ethernet interface 1000BASE-T 4 x USB (1 x front USB 2.0; 3 x back panel USB 3.0)
Diagnostics	Fast and simple troubleshooting through integrated diagnostic aids
Ambient temperature	Operation: in electrical cabinet: 5 °C to 40 °C in operating panel: 0 °C to 50 °C Storage: –20 °C to 60 °C

HEIDENHAIN

DR. JOHANNES HEIDENHAIN GmbH
Dr.-Johannes-Heidenhain-Straße 5
83301 Traunreut, Germany
☎ +49 8669 31-0
FAX +49 8669 32-5061
E-mail: info@heidenhain.de

www.heidenhain.de

DE **HEIDENHAIN Vertrieb Deutschland**
83301 Traunreut, Deutschland
☎ 08669 31-3132
FAX 08669 32-3132
E-Mail: hd@heidenhain.de

HEIDENHAIN Technisches Büro Nord
12681 Berlin, Deutschland
☎ 030 54705-240

HEIDENHAIN Technisches Büro Mitte
07751 Jena, Deutschland
☎ 03641 4728-250

HEIDENHAIN Technisches Büro West
44379 Dortmund, Deutschland
☎ 0231 618083-0

HEIDENHAIN Technisches Büro Südwest
70771 Leinfelden-Echterdingen, Deutschland
☎ 0711 993395-0

HEIDENHAIN Technisches Büro Südost
83301 Traunreut, Deutschland
☎ 08669 31-1337

AR **NAKASE SRL.**
B1653AOX Villa Ballester, Argentina
www.heidenhain.com.ar

AT **HEIDENHAIN Techn. Büro Österreich**
83301 Traunreut, Germany
www.heidenhain.de

AU **FCR MOTION TECHNOLOGY PTY LTD**
Laverton North Victoria 3026, Australia
E-mail: sales@fcrmotion.com

BE **HEIDENHAIN N.V.**
1760 Roosdaal, Belgium
www.heidenhain.be

BG **ESD Bulgaria Ltd.**
Sofia 1172, Bulgaria
www.esd.bg

BR **HEIDENHAIN Brasil Ltda.**
04763-070 – São Paulo – SP, Brazil
www.heidenhain.com.br

BY **GERTNER Service GmbH**
220026 Minsk, Belarus
www.heidenhain.by

CA **HEIDENHAIN CORPORATION**
Mississauga, OntarioL5T2N2, Canada
www.heidenhain.com

CH **HEIDENHAIN (SCHWEIZ) AG**
8603 Schwerzenbach, Switzerland
www.heidenhain.ch

CN **DR. JOHANNES HEIDENHAIN (CHINA) Co., Ltd.**
Beijing 101312, China
www.heidenhain.com.cn

CZ **HEIDENHAIN s.r.o.**
102 00 Praha 10, Czech Republic
www.heidenhain.cz

DK **TP TEKNIK A/S**
2670 Greve, Denmark
www.tp-gruppen.dk

ES **FARRESA ELECTRONICA S.A.**
08028 Barcelona, Spain
www.farresa.es

FI **HEIDENHAIN Scandinavia AB**
01740 Vantaa, Finland
www.heidenhain.fi

FR **HEIDENHAIN FRANCE sarl**
92310 Sèvres, France
www.heidenhain.fr

GB **HEIDENHAIN (G.B.) Limited**
Burgess Hill RH15 9RD, United Kingdom
www.heidenhain.co.uk

GR **MB Milionis Vassilis**
17341 Athens, Greece
www.heidenhain.gr

HK **HEIDENHAIN LTD**
Kowloon, Hong Kong
E-mail: sales@heidenhain.com.hk

HR Croatia → **SL**

HU **HEIDENHAIN Kereskedelmi Képviselet**
1239 Budapest, Hungary
www.heidenhain.hu

ID **PT Servitama Era Toolsindo**
Jakarta 13930, Indonesia
E-mail: ptset@group.gts.co.id

IL **NEUMO VARGUS MARKETING LTD.**
Holon, 58859, Israel
E-mail: neumo@neumo-vargus.co.il

IN **HEIDENHAIN Optics & Electronics India Private Limited**
Chetpet, Chennai 600 031, India
www.heidenhain.in

IT **HEIDENHAIN ITALIANA S.r.l.**
20128 Milano, Italy
www.heidenhain.it

JP **HEIDENHAIN K.K.**
Tokyo 102-0083, Japan
www.heidenhain.co.jp

KR **HEIDENHAIN Korea LTD.**
Gasan-Dong, Seoul, Korea 153-782
www.heidenhain.co.kr

MX **HEIDENHAIN CORPORATION MEXICO**
20290 Aguascalientes, AGS., Mexico
E-mail: info@heidenhain.com

MY **ISOSERVE SDN. BHD.**
43200 Balakong, Selangor
E-mail: sales@isoserve.com.my

NL **HEIDENHAIN NEDERLAND B.V.**
6716 BM Ede, Netherlands
www.heidenhain.nl

NO **HEIDENHAIN Scandinavia AB**
7300 Orkanger, Norway
www.heidenhain.no

NZ **Llama ENGINEERING Ltd**
5012 Wellington, New Zealand
E-mail: info@llamaengineering.co.nz

PH **MACHINEBANKS' CORPORATION**
Quezon City, Philippines 1113
E-mail: info@machinebanks.com

PL **APS**
02-384 Warszawa, Poland
www.heidenhain.pl

PT **FARRESA ELECTRÓNICA, LDA.**
4470 - 177 Maia, Portugal
www.farresa.pt

RO **HEIDENHAIN Reprezentanţă Romania**
Braşov, 500407, Romania
www.heidenhain.ro

RS Serbia → **BG**

RU **OOO HEIDENHAIN**
115172 Moscow, Russia
www.heidenhain.ru

SE **HEIDENHAIN Scandinavia AB**
12739 Skärholmen, Sweden
www.heidenhain.se

SG **HEIDENHAIN PACIFIC PTE LTD**
Singapore 408593
www.heidenhain.com.sg

SK **KOPRETINA TN s.r.o.**
91101 Trencin, Slovakia
www.kopretina.sk

SL **NAVO d.o.o.**
2000 Maribor, Slovenia
www.heidenhain.si

TH **HEIDENHAIN (THAILAND) LTD**
Bangkok 10250, Thailand
www.heidenhain.co.th

TR **T&M Mühendislik San. ve Tic. LTD. ŞTİ.**
34775 Y. Dudullu –
Ümraniye-Istanbul, Turkey
www.heidenhain.com.tr

TW **HEIDENHAIN Co., Ltd.**
Taichung 40768, Taiwan R.O.C.
www.heidenhain.com.tw

UA **Gertner Service GmbH Büro Kiev**
02094 Kiev, Ukraine
www.heidenhain.ua

US **HEIDENHAIN CORPORATION**
Schaumburg, IL 60173-5337, USA
www.heidenhain.com

VN **AMS Co. Ltd**
HCM City, Vietnam
E-mail: davidgoh@amsvn.com

ZA **MAFEMA SALES SERVICES C.C.**
Midrand 1685, South Africa
www.heidenhain.co.za