HEIDENHAIN

CNC PILOT 640

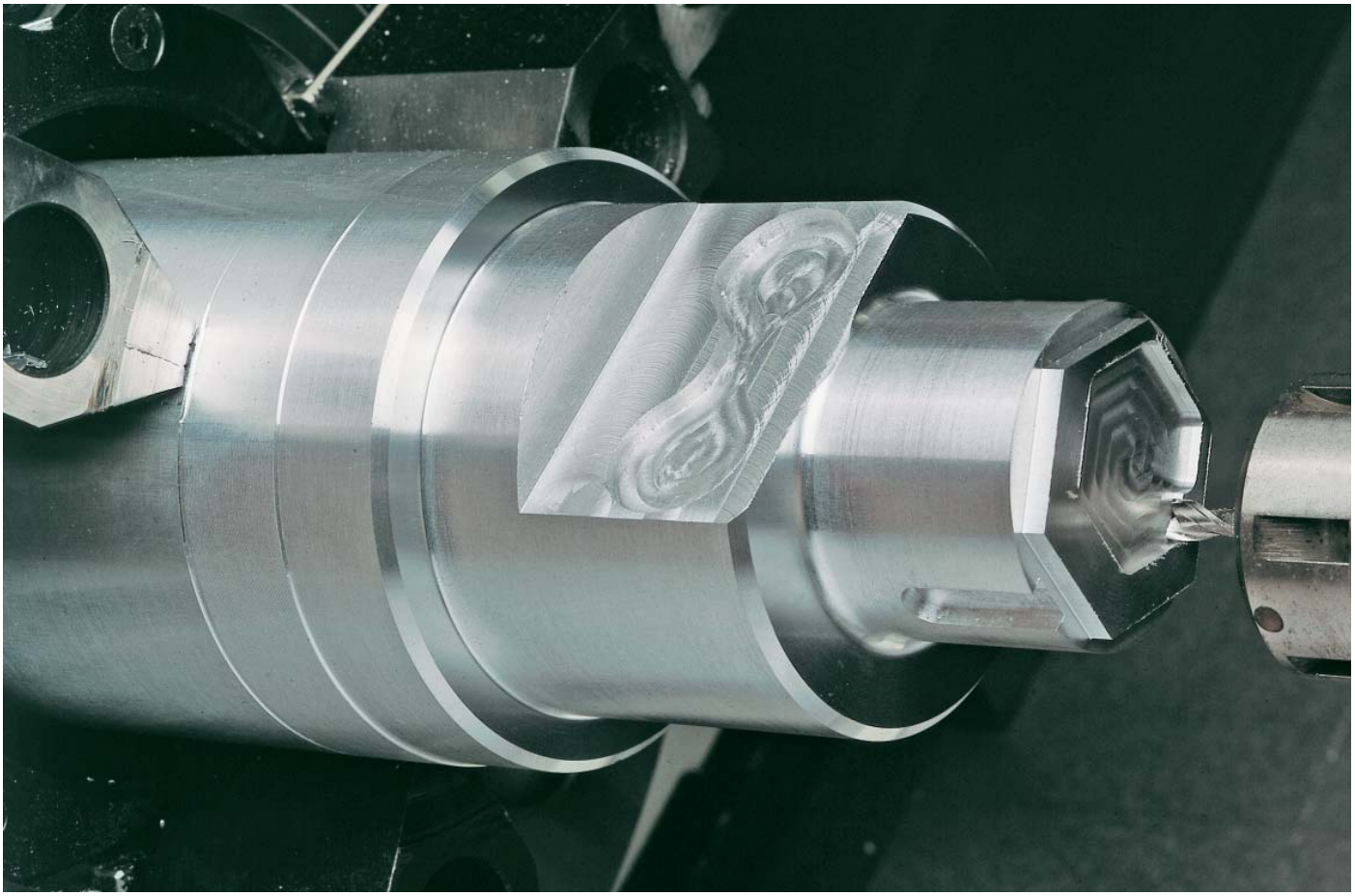
The Contouring Control for Lathes and Turning-Milling Machines

HSCI

September 2015

Start smart

For years, lathe controls from HEIDENHAIN have been proving themselves on compact, but also complex lathes. They distinguish themselves particularly with their convenient and simple programming as well as their high-quality, ergonomic operating components.



This brochure describes the functions and specifications of the CNC PILOT 640 with NC software 68894x-03.

Contents

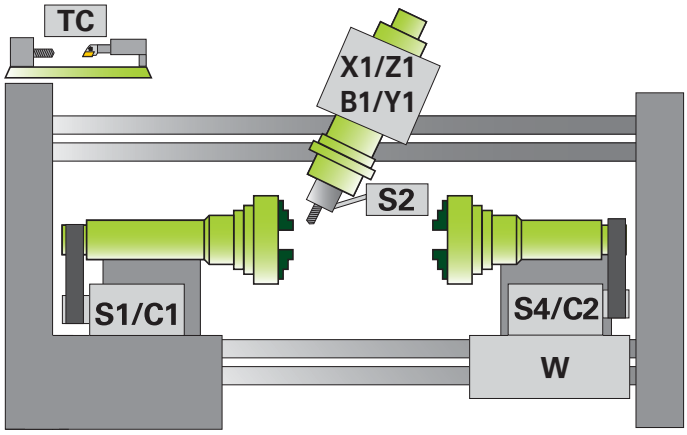
The CNC PILOT 640...

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

Thanks to its flexible design and numerous programming features, the CNC PILOT 640 always gives you optimum support. Regardless of whether you are manufacturing single parts or batches, simple or complex workpieces, the CNC PILOT 640 is characterized by its simple operation and programming. It is quickly learned and requires minimum training time.

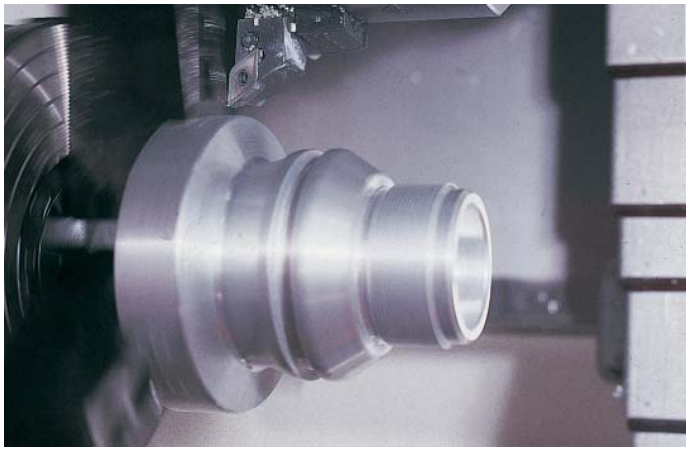
The CNC PILOT 640 was conceived for CNC lathes. It is suitable for horizontal and vertical lathes.

The CNC PILOT 640 supports lathes with main and opposing spindle, one slide (X and Z axis), C axis or positionable spindle, driven tools, and machines with Y and B axes.

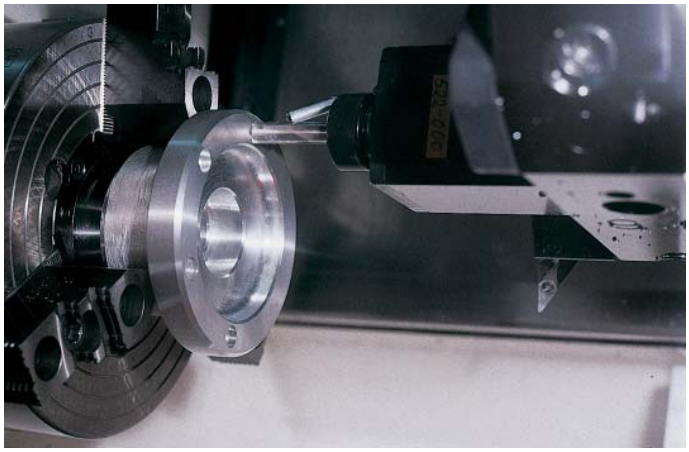


Regardless of whether you are turning simple parts or complex workpieces, the CNC PILOT 640 provides you with the benefits of graphical contour input and convenient programming with smart.Turn.

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.



From simple jobs on a compact machine ...



... to complex tasks

... to large batch production

Well designed and user friendly
– The CNC PILOT 640 in dialog with the user

The screen
The 19-inch TFT color flat-panel display shows a clear overview of all relevant information for programming, operating and inspecting the machine tool and control such as program blocks, comments and error messages.

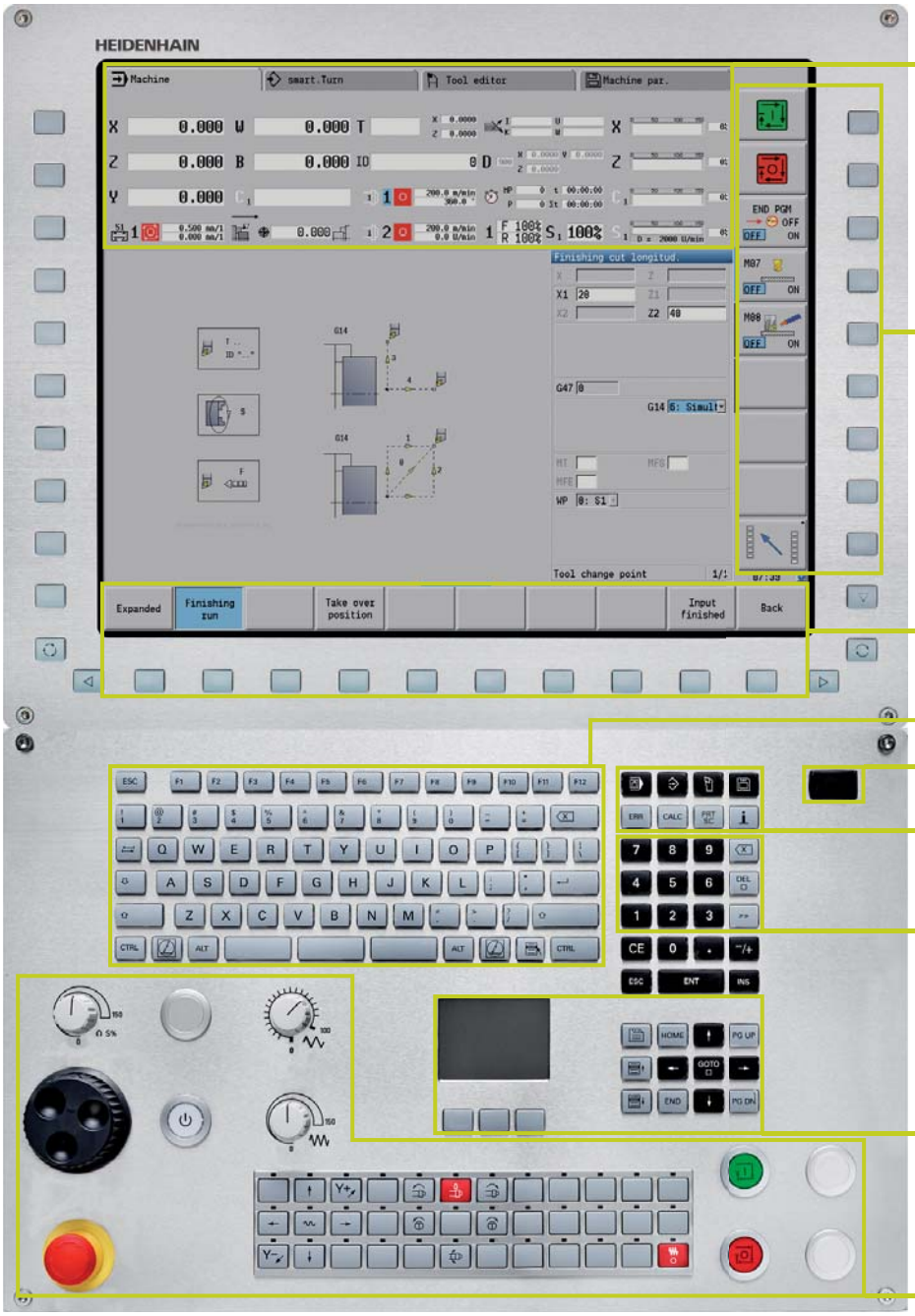
During program input the required parameters are illustrated in help graphics, and during Test Run the CNC PILOT 640 simulates the cutting process in full detail on the screen. During program run the screen displays information on the tool position, the rotational speed, the feed rate and the utilization of the drives as well as further information on the machine status.

The positions of the tool are shown in large characters. The respective distance-to-go, the feed rate, the spindle speed and the ID number of the current tool are also clearly visible. A moving-bar diagram shows the current utilization of the spindle and the axis drives.

The keyboard
The CNC PILOT 640 needs very few keys. Easily understood symbols clearly indicate the functions.

The keys on the numeric keypad are used both for data input and for selecting the functions. The menu window displays the available functions graphically. The function keys below the screen are used to modify the selected functions, assume position and technology values, and control the data input.

As an alternative, the CNC PILOT 640 is also available with a 15-inch screen and matching control panel. The PLC soft-key row on the left side of the monitor is missing in this version.



Display of operating modes and machine status (configurable)
You can choose a suitable function for each of the 16 fields, and save different display assignments for the automatic and manual mode.

PLC function keys for machine functions

Self-explanatory function keys for NC programming

Alphanumeric keyboard for comments

USB interface

Keys for operating modes and functions

Keypad for numerical input and fast, direct menu selection

Keys and touchpad for navigation

Machine operating panel with override potentiometer and electronic handwheel

Keys on the monitor

Switch the help graphics between outside/inside machining (cycle programming)

Soft keys for selecting functions on screen

Shift between soft-key rows

Keys on the control panel

Operating mode keys

Machine Operating Modes

Programming Operating Modes

Tables for tool data and technology data

Parameters, file management, transfer, diagnostics

Navigation keys

PG UP PG DN Screen/page up/down

HOME END Go to beginning of program/list or to end of program/list

smart.Turn: switches to the next detail input form

smart.Turn: switches to the previous/next group

Special keys

CALC Calculator

ERR Call up messages and errors

Info key

GOTO Display block or activate special functions, such as input options or text input

PRT SC Save screen contents as graphic file

>> Expanded input capabilities

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

Thanks to its digital design, the CNC PILOT 640 has control over the machine's entire drive system. Not only does the field-proven digital drive technology from HEIDENHAIN make high contour accuracy and rapid machining at high speeds possible, but also all control components of the CNC PILOT 640 are connected via digital interfaces.

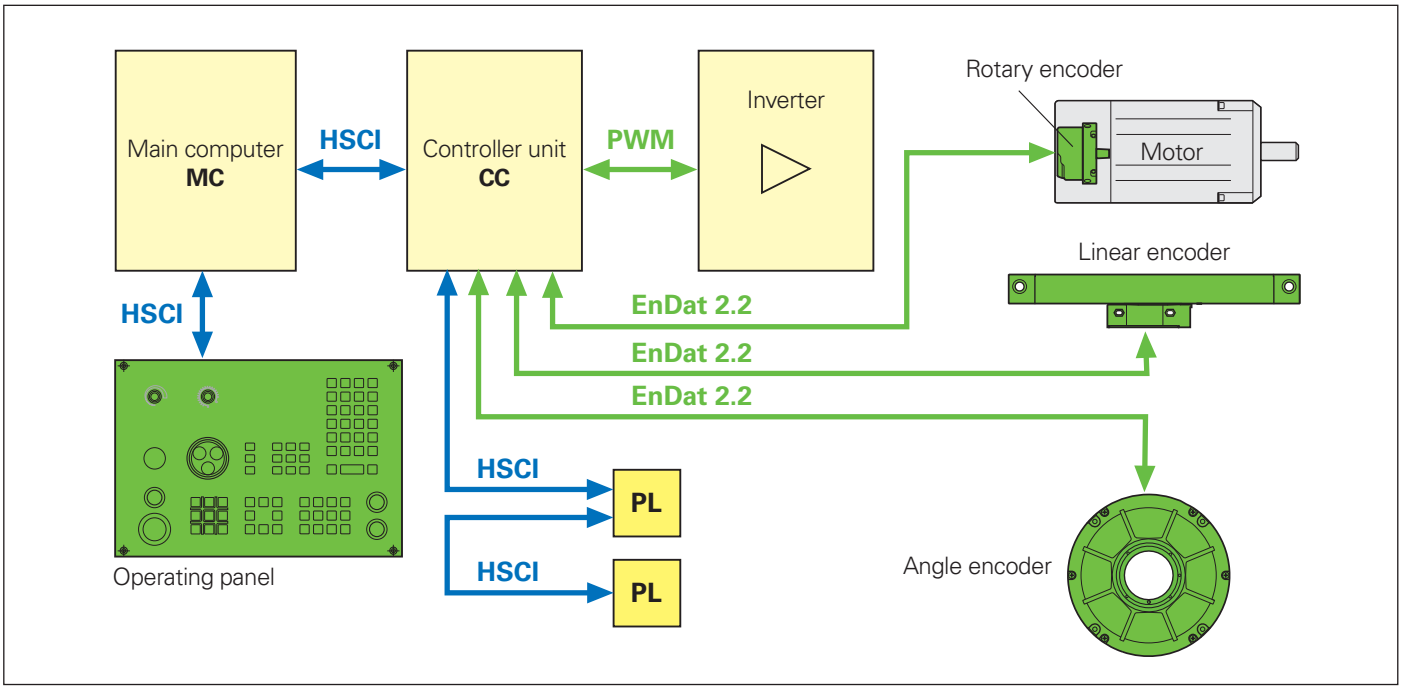
Digital drive technology
The position controller, speed controller and, if required, the current controller are integrated in the CNC PILOT 640. The digital motor control makes it possible to attain very high feed rates.

High contour accuracy
The CNC PILOT 640 dynamically calculates the contour in advance. This enables it to adapt the axis velocities early enough to the contour transitions. It controls the axes with special algorithms that ensure path control with the required limits to velocity and acceleration.

High availability
In the uniformly digital control concept of the CNC PILOT 640, all components are connected to each other via purely digital interfaces: The control components are connected via HSCI (HEIDENHAIN Serial Controller Interface), the real-time protocol from HEIDENHAIN for Fast Ethernet, and the encoders are connected via EnDat2.2, the bidirectional interface from HEIDENHAIN.

This achieves a high degree of availability for the entire system. It can be diagnosed and is immune to noise—for everything from the main computer to the encoder.

The uniformly digital design from HEIDENHAIN guarantees not just very high accuracy and surface quality, but high traverse speeds as well.

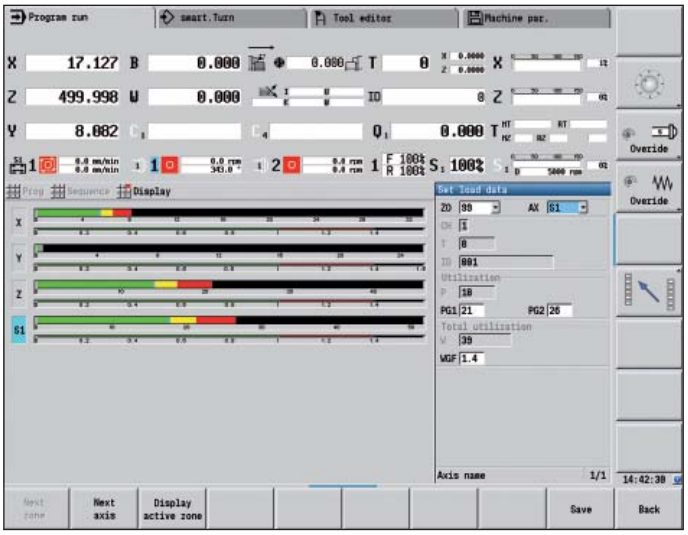


– Intelligent supervision with load monitoring (option)

Load monitoring—detecting tool wear and breakage during machining*
The load monitor observes the machine's spindle and motor load while comparing them with the utilization values of a reference operation. The CNC PILOT 640 can graphically display the utilization rates 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 predefined 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, in particular during unmanned shifts.

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



Graphic display of the load values

Effective, clearly organized and flexible
– Simple programming with smart.Turn (option)

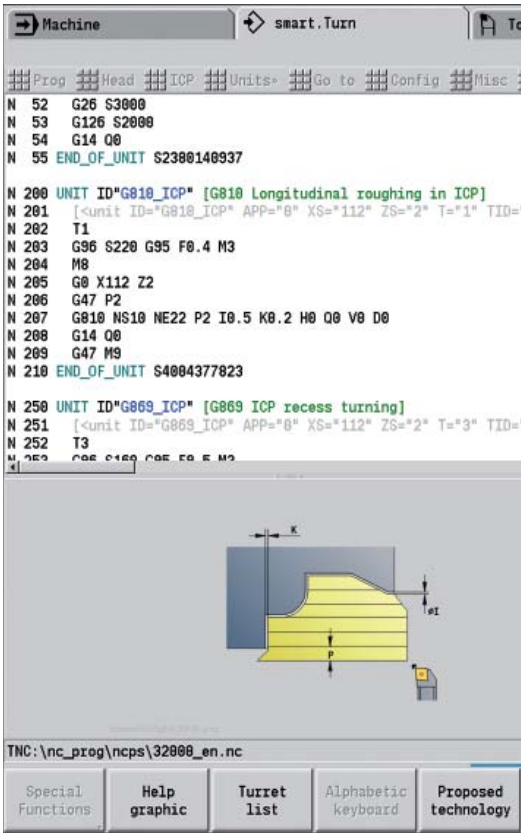
Has the safety clearance been correctly entered, is the speed limit taken into account, how are oversizes defined? All this needs to be considered not only by the beginner, but also by the experienced NC programmer when creating conventional DIN programs.

The smart.Turn principle
The working block—called a unit—plays the central role in smart.Turn programs. A unit describes a machining step completely and unambiguously. The unit includes the tool call, the technology data, the cycle call, the approach and departure strategies as well as global data, such as safety clearance, etc. All these parameters are summarized in one, clearly structured dialog box.

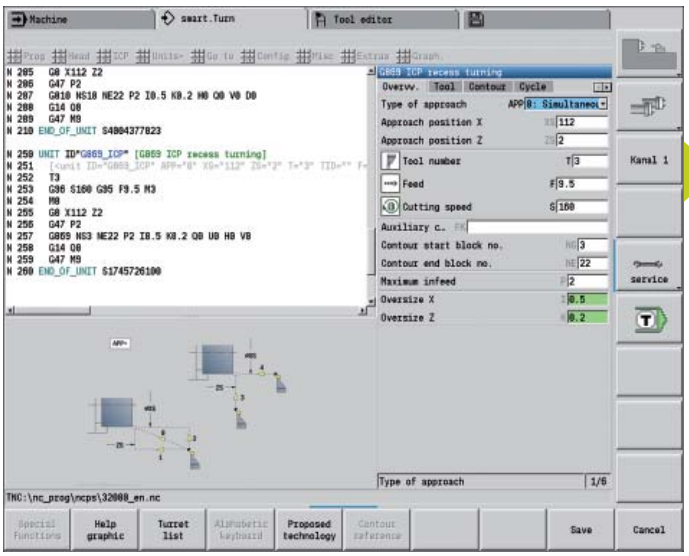
The smart.Turn principle gives you the reassurance that the working block is defined correctly and completely. In the NC program, smart.Turn lists the DIN PLUS commands of the unit. This gives you an overview of all working-block details at any time.

The unit
All parameters of a smart.Turn unit are united in simple and well designed fillable forms. The overview form shows you a summary of the selected unit, and subforms provide information on the details of a working block. Clearly arranged help graphics illustrate all required input. 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 for programming with smart.Turn. You can create and test the smart.Turn program while the program is running.



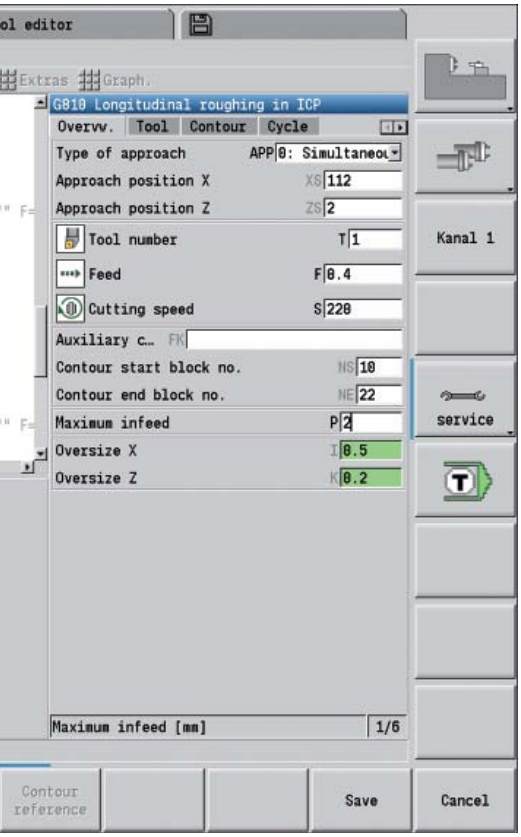
Straightforward dialogs;
help graphics illustrate the parameters.



Input form in smart.Turn

```
N 250 UNIT ID"G869_ICP" [G869 ICP recess turning]
N 251   [<unit ID="G869_ICP" APP="0" XS="112" ZS="2"
N 252   T3
N 253   G96 S160 G95 F9.5 M3
N 254   M8
N 255   G0 X112 Z2
N 256   G47 P2
N 257   G869 NS3 NE22 P2 I0.5 K0.2 Q0 U0 H0 V0
N 258   G14 Q0
N 259   G47 M9
N 260 END_OF_UNIT S1745726100
```

smart.Turn unit in the NC program



Structured and easy-to-read
Clearly structured and easy-to-read—these are the characteristics of smart.Turn programs. smart.Turn uses section codes that clearly distinguish between the program head with setup information, the turret assignment, the workpiece description and the actual machining operation.

Under dialog guidance, you enter in the following order:

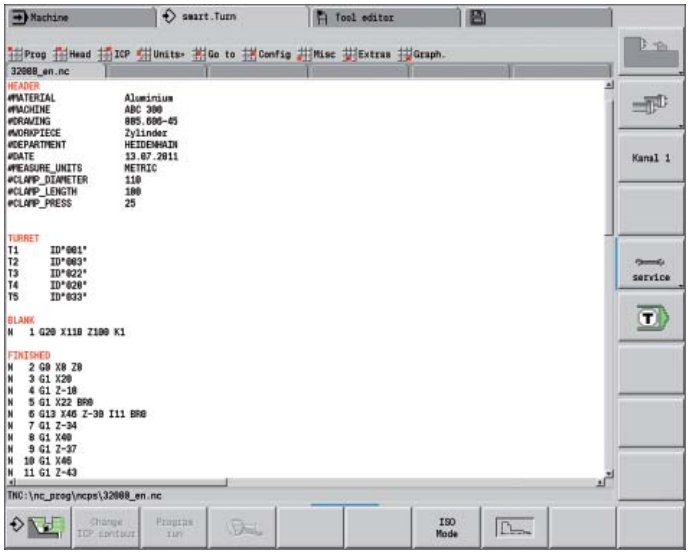
- Program head
- Tool assignment in the turret
- Workpiece-blank definition
- Description of machined part
- Individual machining steps

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

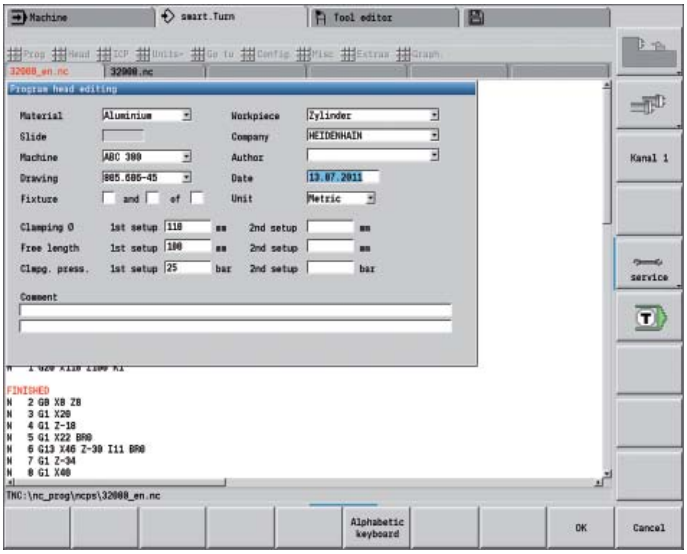
Production data at a glance
The program head includes all important information on the workpiece, e.g. drawing number, date, programmer, material, fixture, etc.

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

Programming in more than one window
Up to six NC programs can be opened simultaneously in the DIN PLUS editor. The part program to be displayed is selected using the smart keys. This enables you to transfer program blocks from one part program to another and allows you to quickly get an overview of complex part programs including subprograms.



smart.Turn program with section codes



Input form for program head

Effective, clearly organized and flexible
– Simple 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.

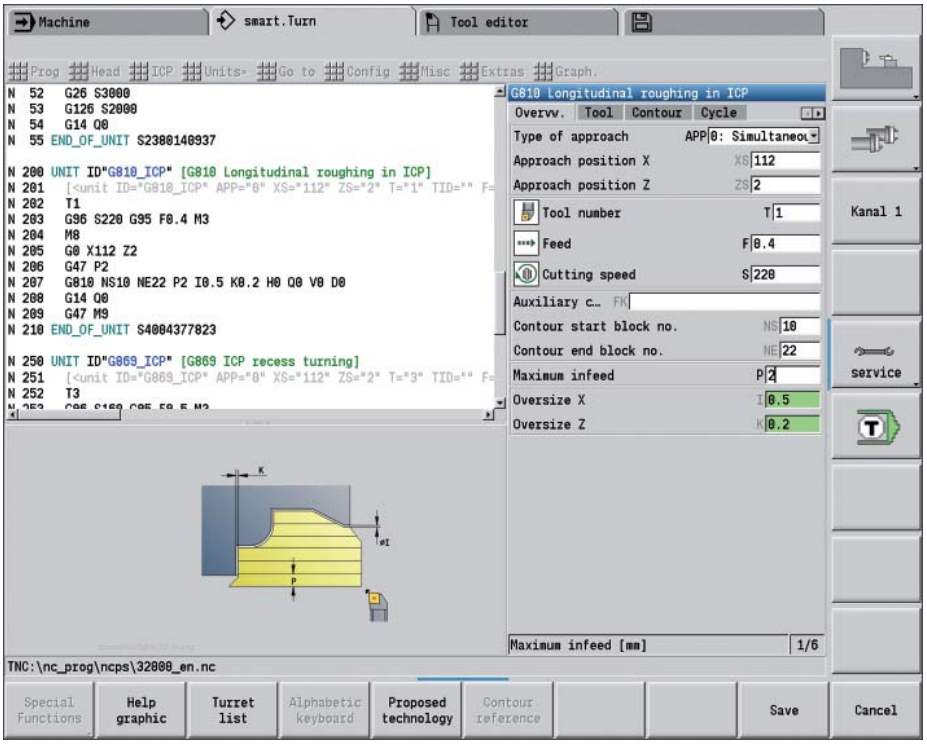
In the NC program, smart.Turn lists the DIN PLUS commands of the unit. This not only gives you an overview of all working-block details, but you also have a clearly legible and well-structured NC program.

smart.Turn supports units for roughing, finishing, recessing, recess turning, thread cutting, boring, 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
smart.Turn enables you to work simply and flexibly. Simple contours can be defined with just a few entries in the cycle. Complex contours are described with ICP graphic interactive programming. Workpiece descriptions that are available in DXF format can be easily imported. Contours are saved in the NC program in a consistently legible and editable form. 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 the 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 new blank for each new cut. The machining cycles always adapt to the current workpiece blank. The contour follow-up helps you to avoid air cuts and optimize approach paths, even if the workpiece material has been previously removed.

Technology data as default values
The CNC PILOT 640 saves your cutting data according to the criteria of workpiece material, tool material and machining mode. As you have already entered the cutting material in the tool definition, you need only enter the material of your workpiece. This provides smart.Turn with all data for setting default values for the cutting data.



smart.Turn dialogs with help graphic

– Powerful NC programs with DIN PLUS

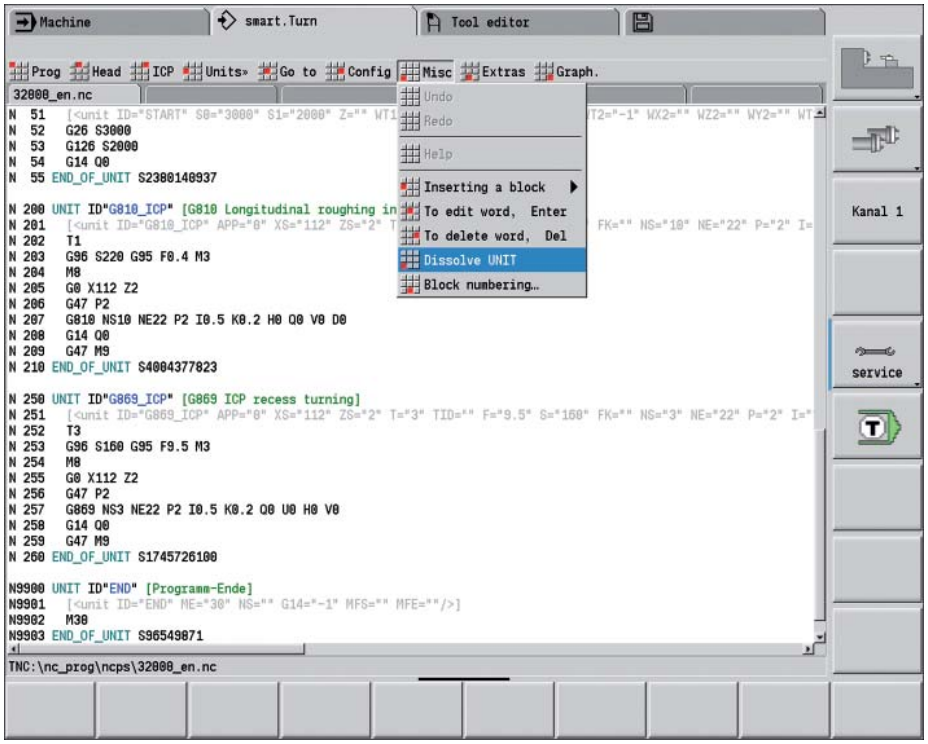
Programming in DIN PLUS
smart.Turn offers units for all machining tasks as well as units for special functions. If you want to control special machine components, or use the variable programming function or other complex functions that are not provided by smart. Turn, DIN PLUS will support you. It provides powerful machining cycles, program branches 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 break up a unit into blocks at any time 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 test each work step immediately in the simulation. You select the respective commands from a menu or enter them directly with G codes. The screen displays a dialog box in which you enter the related parameters. All input is explained on screen in plain language and with graphic illustrations.

Thanks to the powerful fixed cycles and the assignment of cycles to machining sections, with DIN PLUS you dramatically improve effectiveness and flexibility compared to conventional part programming.



smart.Turn program with display of menu item for breaking up a smart.Turn unit

Effective, clearly organized and flexible
– NC program at the push of a button with TURN PLUS (option)

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

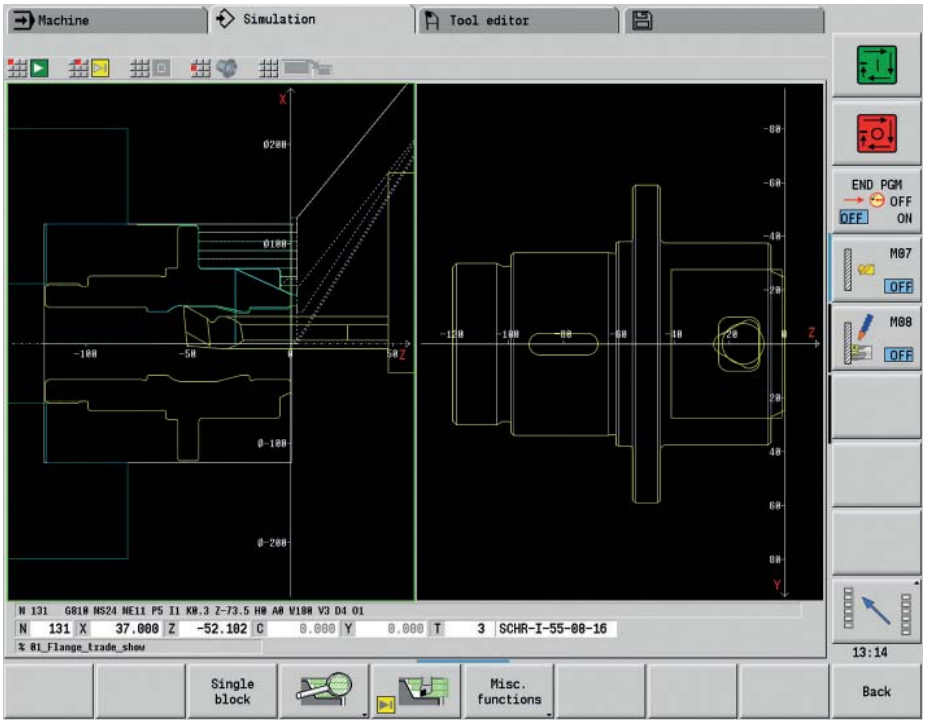
Your result is a comprehensively commented smart.Turn program with working blocks (units). That gives you the assurance you need for optimization and safety when you're breaking in the part program.

TURN PLUS can do all that for milling, drilling and boring operations with the C or Y axis on face and cylindrical surfaces as well as on rear-face surfaces in machines with opposing spindles.

The part program at a keystroke
If short programming times are important to you, you can generate all machining steps by pressing a single key. On the basis of the contour entered and the information from the technological database, TURN PLUS independently prepares the working plan and chooses suitable machining strategies, tools and cutting data. The whole operation takes only a few seconds. You can monitor each individual step in the control graphics. 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 yourself to suit various tasks. In this way the CNC PILOT 640 can profit from your company's machining know-how even during the automatic working plan generation.

Automatic program generation for full-surface machining
The CNC PILOT 640 automatically generates the part program even for complex workpieces that need to be machined on the front face, back face, and lateral surfaces. After defining the geometry, this can save you about 90 % of the time otherwise needed to create a program.

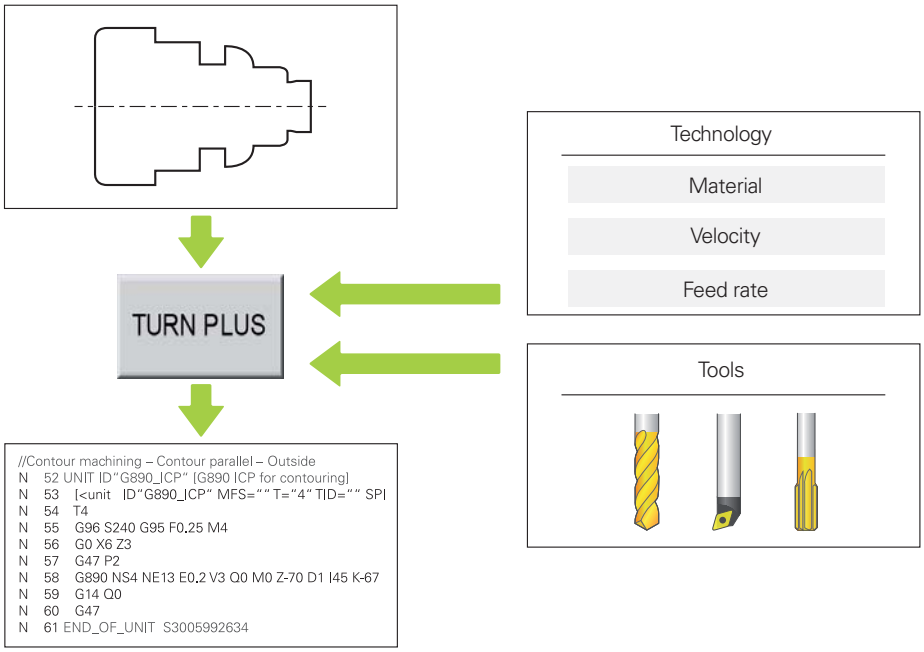
Automatic program generation for the second setup
TURN PLUS knows the contour of the clamping devices when it generates the working plan. It automatically limits the tool path to a safe distance from the clamping device. When the program for the first setup is completed, you can "rechuck" using interactive graphics. The control then automatically generates the program for the second setup using the workpiece geometry that has already been entered.



Inclined contours are no problem
The control is presented with special requirements when it has to generate the part program for inclined contours. Often, the angle the contour falls off at is steeper than the tool tip angle. In that case the control automatically chooses another tool and machines the contour in the opposite direction or as a recess. In any case the result is an executable part program.

Defining the machining sequence
In the TURN PLUS dialog you can define a standard machining sequence.
You can save various machining sequences, e.g for chuck parts or shaft machining.

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



Go with TURN PLUS in the shortest possible time from programming to the first cut

Describing and importing contours
– ICP interactive contour programming

For jobs that cannot be machined with the standard cycles because of the complexity of the workpiece or the lack of certain dimensions in the workpiece drawing, you need ICP, the interactive contour programming. 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 already specify the direction of the line or the direction of rotation of the circular arc. This way the CNC PILOT 640 needs very little information about the contour element.

When entering the data, you decide whether the coordinates are absolute or incremental, and whether you enter the end point or the length of the line or the center point or the radius of a circular arc. You also specify whether the path to the next contour element should be tangential or non-tangential.

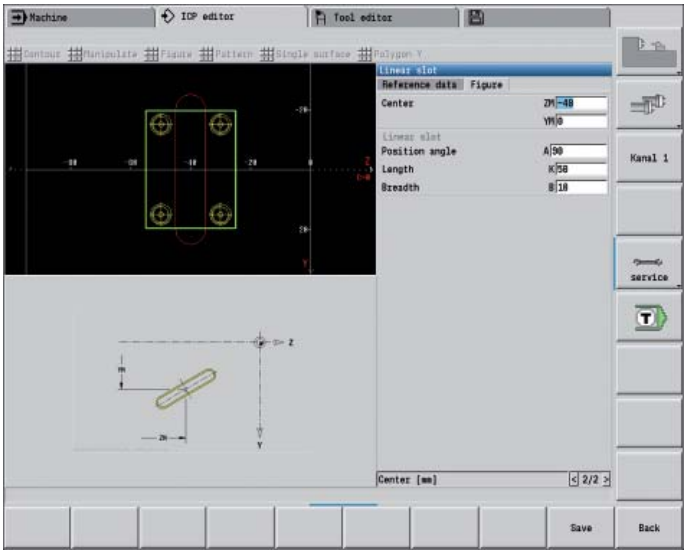
As long as they are mathematically defined, the CNC PILOT 640 calculates missing coordinates, intersections, center points, etc. If the entered data permit several mathematically possible solutions, you can view the individual solutions and then select the proposal that matches the drawing. You can modify or change 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 in the course of the sequential contour definition. However, it is often easier to first define the "rough" contour, and then to 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.

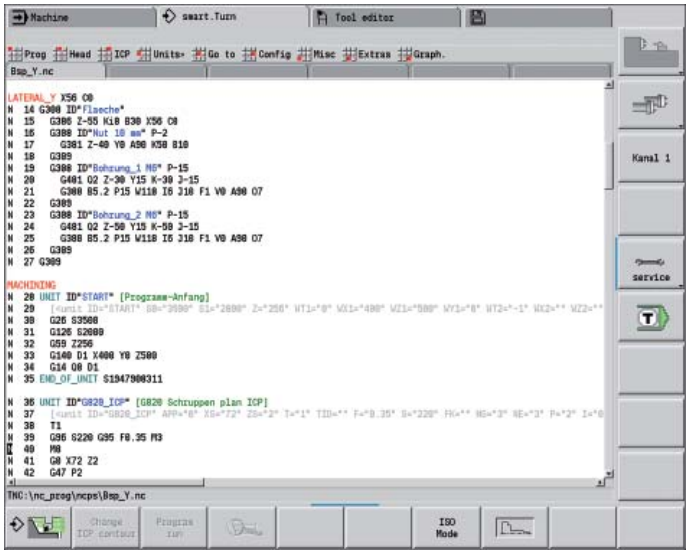
ICP contours for smart.Turn and DIN PLUS
In smart.Turn you have various possibilities for describing the contour to be machined. You can describe simple contours right in the unit and use ICP for complex turning or milling contours as well as linear or circular drilling and milling patterns. The contour defined with ICP is transferred to 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 legible 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 also describe the turning and milling contours, linear and circular patterns with ICP. In the contour-based cycles you enter a reference to the contour section to be machined.

The ICP editor can be called immediately from within smart.Turn.



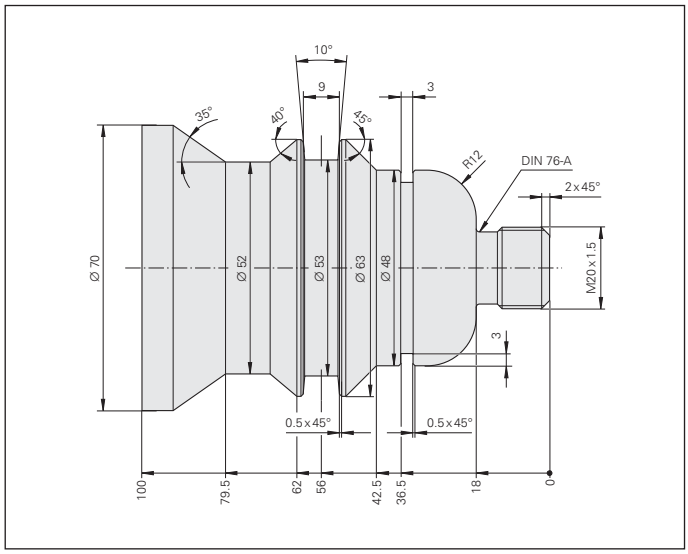
Define the contour element in ICP.



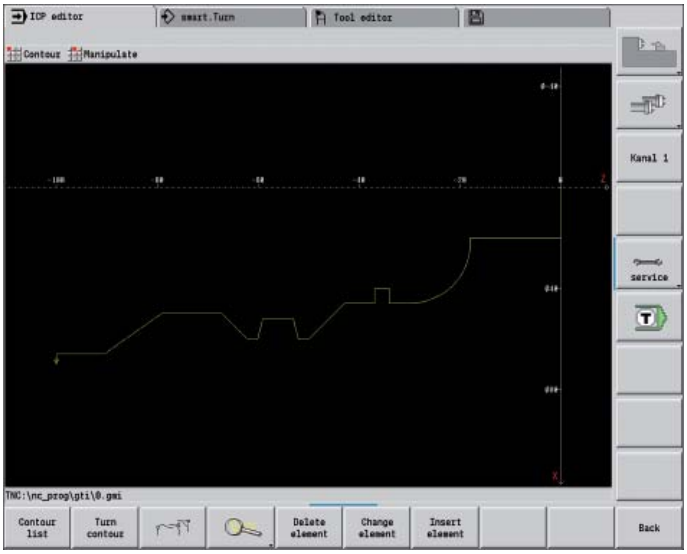
ICP contour description in the NC program

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 does this save time otherwise spent on programming and testing, but you can also be sure that the finished contour is exactly according to the designer's specifications. DXF contours can describe workpiece blanks, finished parts, contour trains and milling contours. They must exist as two-dimensional elements in a separate layer, i.e. without dimension lines, wrap-around edges, etc.

First, you download the DXF file onto the CNC PILOT 640 over the network or use a USB stick. Since the DXF format is fundamentally different from the ICP format, the contour is converted from DXF to 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

Realistic testing before machining
– Graphic simulation

Timely detection of errors is very important, particularly for NC programming. With its graphic simulation feature, the CNC PILOT 640 supports you in checking the program for errors—exactly and with the real dimensions of the contour and cutting edge, because the simulation operates with the geometry values from the tool database.

Graphic simulation
Before actual machining, you use the graphic simulation to inspect the
- approach and departure behavior,
- the machining sequence,
- the proportioning of cuts,
- and the finished contour.

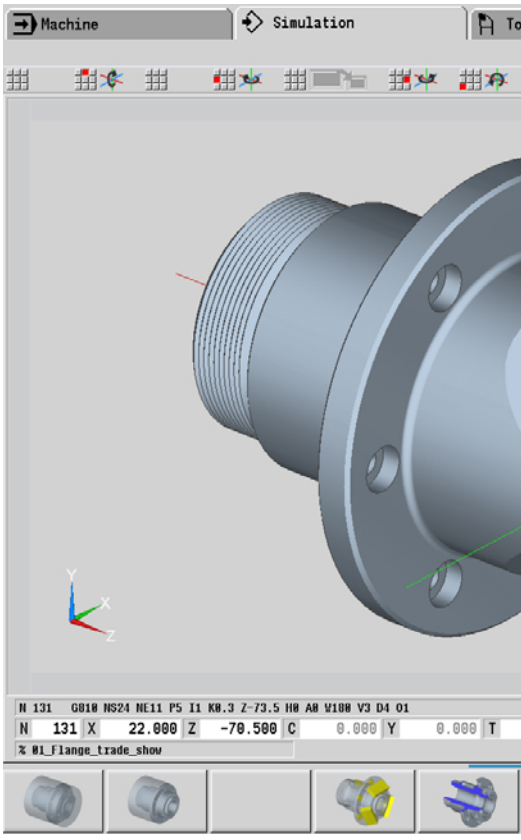
In the graphic simulation you can display the tool cutting edge. You see the cutting-edge radius, the cutting-edge width and the cutting-edge position with their actual dimensions. This helps to recognize machining details or collision risks in time.

Wire-frame or cutting-path graphics, 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 only need a quick overview of the approaching and departing movements and the proportioning of cuts. The wire-frame graphics illustrate the path of the theoretical cutting point.

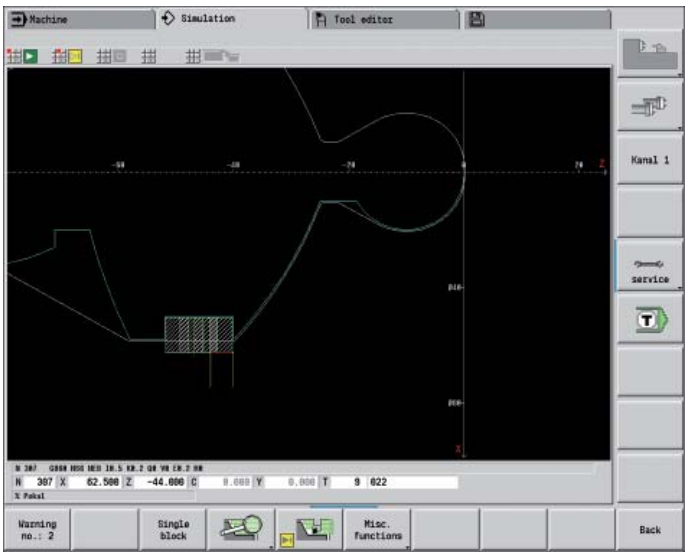
A more accurate contour verification is provided by the **cutting-path graphics**. The cutting-path graphics account for the exact geometry of the tool tip. You immediately see if material was left behind, the contour is damaged or the overlaps are too large. The cutting-path graphics is especially useful for recessing, drilling and milling operations where the tool shape has an essential influence on the accuracy of the resulting workpiece.

The **machining simulation** (material removal graphic) displays the workpiece blank from which material is removed. The blank is displayed as a white surface. The CNC PILOT 640 simulates every tool movement at the programmed cutting speed and removes the machined material.

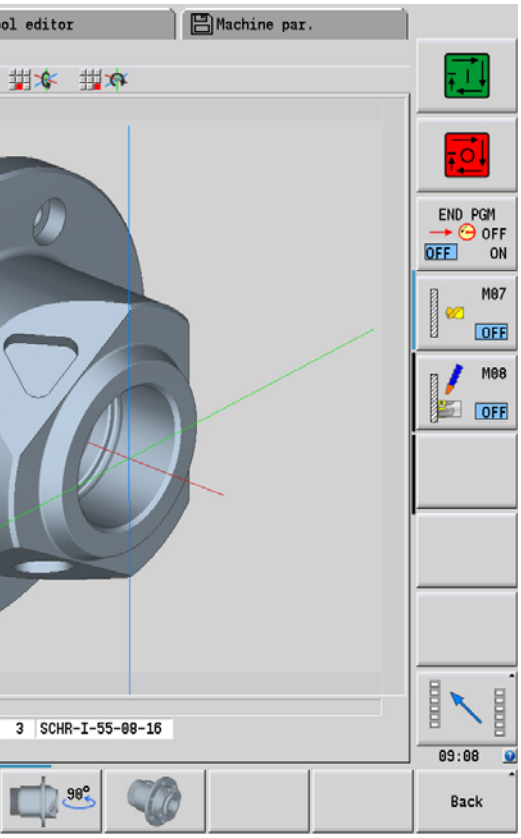


Finished part in a 3-D graphic

Wire frame graphics



Cutting path graphics



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

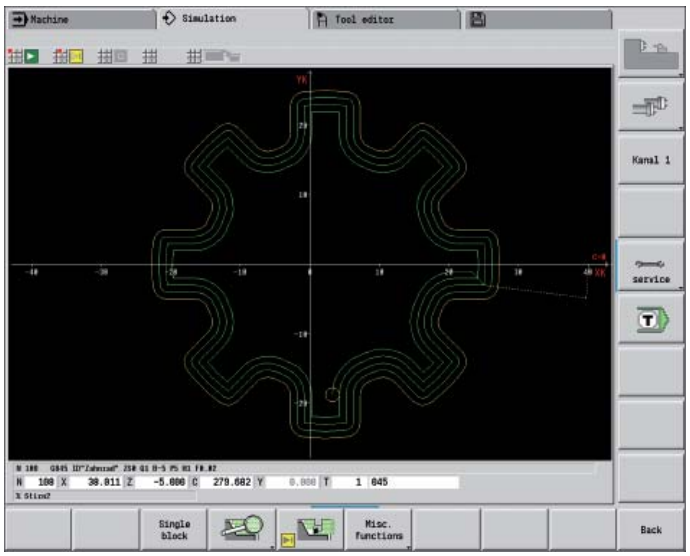
The CNC PILOT 640 depicts C-axis machining of the cylindrical surface as an "unrolled" plane surface.

3-D simulation graphics
With the high-resolution, finely detailed 3-D graphic simulation, you can exactly evaluate the result of drilling, turning or milling processes even before actual machining.

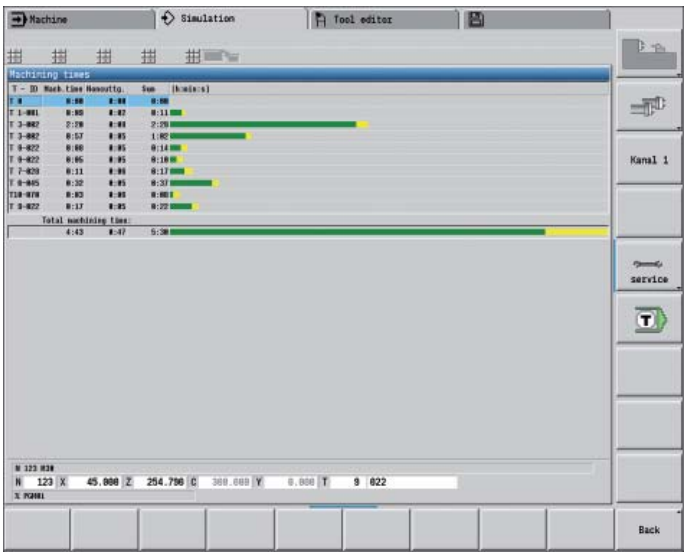
The freely rotational view about the axes permits visual inspection of the blank and finished part from all angles. With its intuitive mouse and keyboard operation, you can navigate and zoom into every programmed detail—of course even with C-axis contours on the cylindrical surface or face, and with Y-axis contours in the tilted plane. In this way the 3-D simulation graphics enable you to detect even the smallest error already before machining.

Calculating the machining time
If your customer needs an offer in a hurry, and you need exact information in a very short time, the CNC PILOT 640 is a valuable aid with 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.

Along with the total time, the table displays the machining time and idle time of each cycle or each tool insert. This assists you not only in your calculations, but you can also tell at a glance whether there are more possibilities for optimization during the machining process.



Machining the front face



Calculation of machining time

Expandable for complex tasks
– Turning, drilling and milling in one setup (option)

You can use the CNC PILOT 640* to drill and mill your workpiece on the front face and lateral surface in one setup. In addition, the control offers you numerous functions and well-proven cycles

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

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

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

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—the milling cycles automatically take care of the rest.

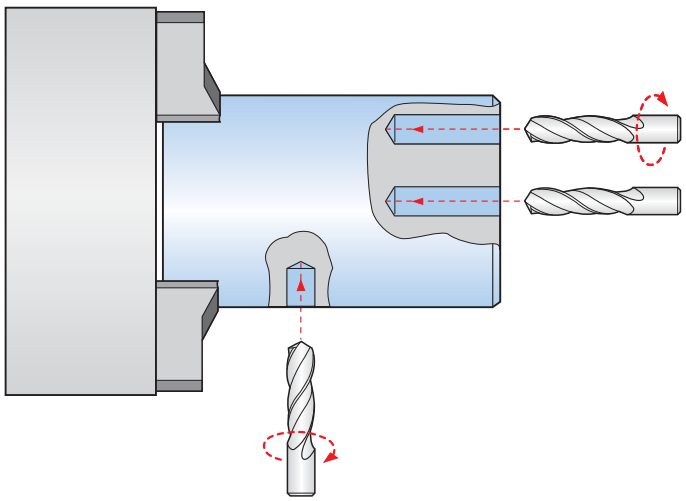
Even for simple contours such as circles, rectangles and equilateral polygons, just a few keystrokes are necessary to determine the figure and position.

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 the important details, such as 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 good surface quality.

In smart.Turn and DIN programming, 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 important parameters such as pitch, cutting in multiple infeeds, etc.

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

On the workpiece face you can arrange the characters on a line or an arc. On the lateral surface, and when engraving with the Y axis, you define the angle at which the characters are to be arranged.

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.



Slot milling



Figure milling (circles, rectangles, regular polygons)

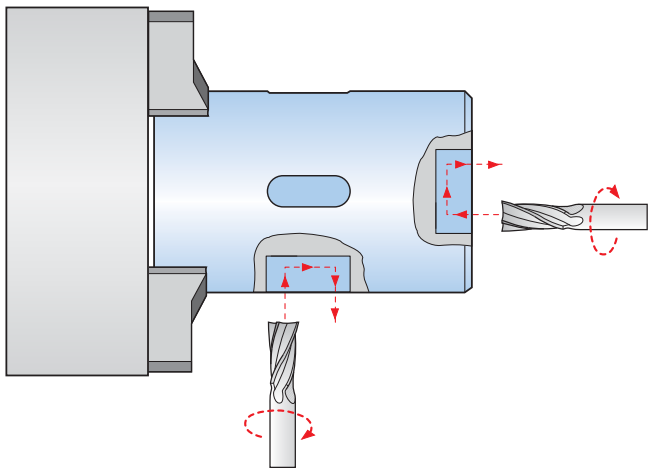


ICP contour milling


Face milling (single surfaces, flattening, polygon)


Helical slot milling

smart.Turn units and DIN PLUS cycles for milling



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 any machine configuration: it performs complex machining tasks with a C or Y axis. It also controls full-surface cutting on dual-spindle machines.

And 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 spindle is at rest. The C axis or positionable spindle permit milling, drilling, and boring on the face and lateral surface of the workpiece. These elements can be displayed for programming and verification in face view and in the unrolled lateral surface view.

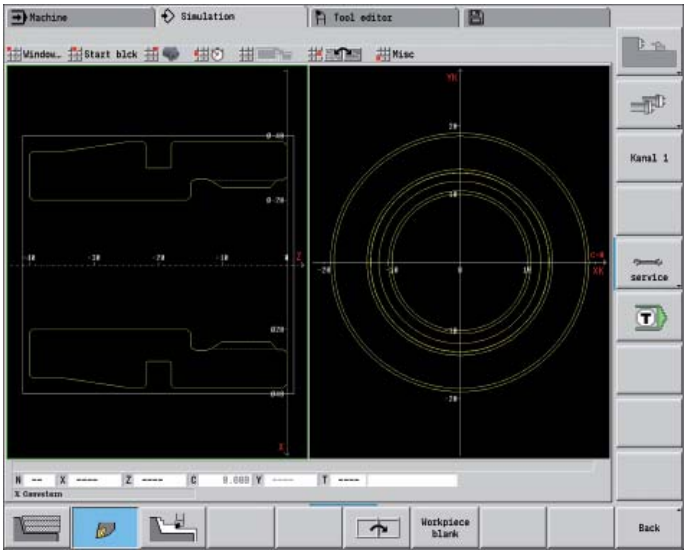
Y axis*
With the Y-axis option of the CNC PILOT 640 you can machine slots or pockets with plane bottoms and perpendicular slot angles. By defining the spindle angle, you can determine the position of the milling contours on the workpiece. For programming and verification of these machining sections, the workpiece is shown in side and face view. 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)

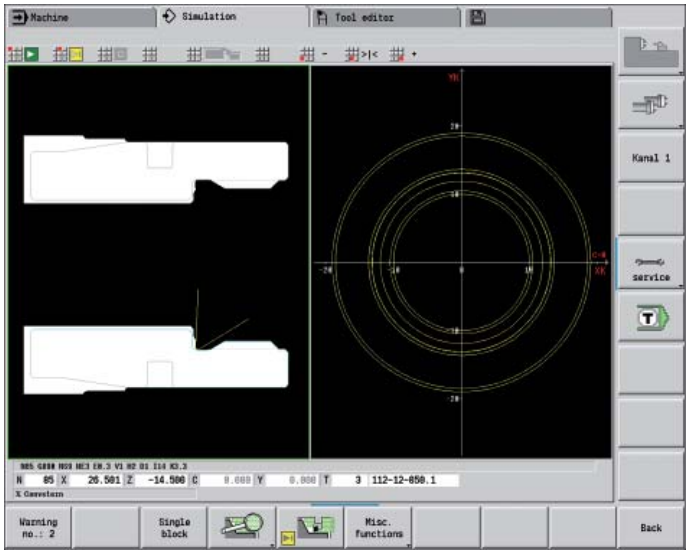
These features are complemented by additional functions such as coordinate transformation, spindle synchronization and traversing to a stop surface.

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

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



Graphic contour programming for C-axis machining (milling, drilling and boring)



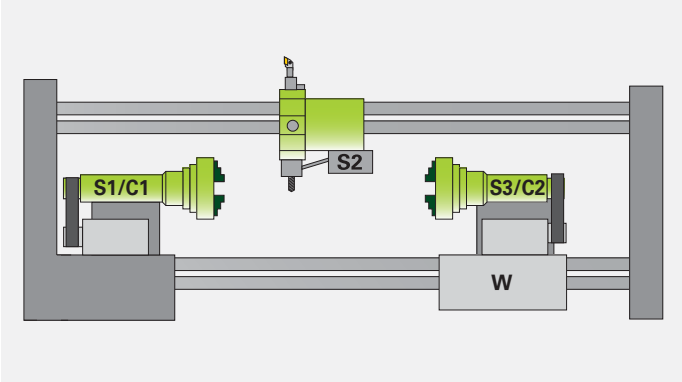
First fixture

Spindle synchronization
Opposing 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 the time otherwise spent braking and starting the spindles. The CNC PILOT 640 detects any angular offset and compensates it during subsequent milling on the opposing spindle.

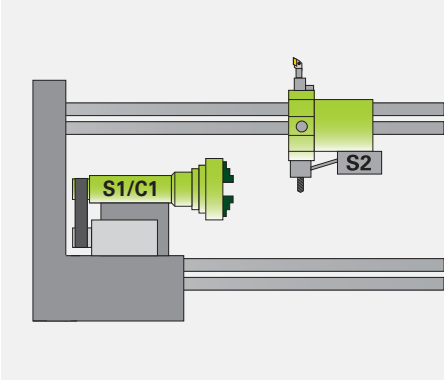
Traversing to a fixed stop
To ensure that the workpiece is firmly pressed to the opposite spindle surface, the control monitors the nominal and actual positions while the longitudinal axis is moving and thereby detects the fixed stop. The CNC PILOT 640 monitors the motor torque and uses it to reach the programmed contact force.

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—in addition to the actual contour machining—traverse movements of the X and Y axes are superimposed. The manufacture of cams and non-circular parts is possible without any additional machine elements.

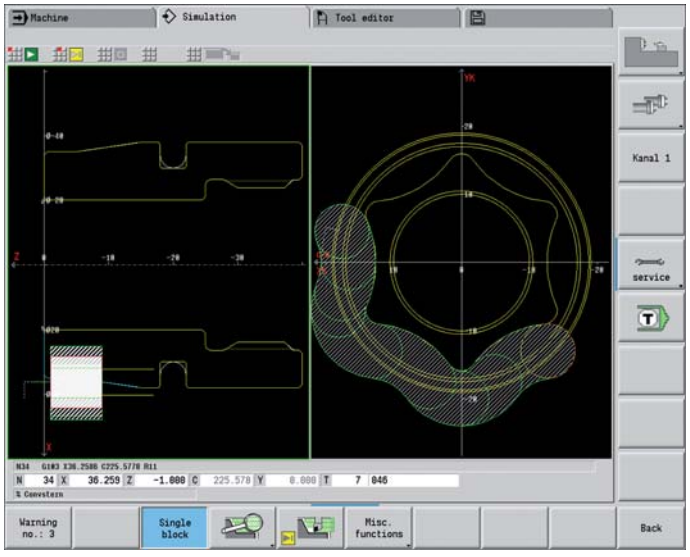
* These functions must be implemented in the machine and control by the machine tool builder.



Full-surface machining: opposing spindle (S3) with C axis (C2) on secondary axis (W) and driven tool (S2)



Spindle 1 (S1) with C axis (C1) and driven tool (S2)



Machining of the rear face on the opposing spindle after automatic workpiece transfer

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

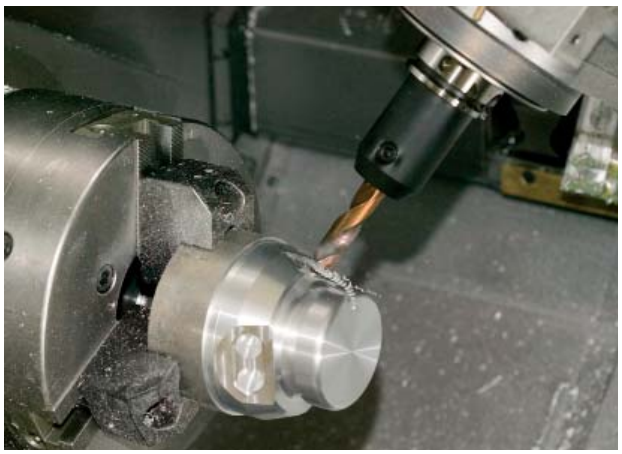
The B axis makes it possible to drill, bore and mill in oblique planes. At first glance, programming for such operations seems very complex and compute-intensive. But with the CNC PILOT 640 you simply tilt the coordinate system to the required position and program machining as usual in the working plane.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

The B axis also provides benefits for 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 machine in the longitudinal and transverse directions on the main and opposing spindles. That allows you to reduce the number of tools needed as well as do without certain tool changes.

Programming
The usual separation of contour description and machining on the CNC PILOT 640 also applies to milling, drilling and boring 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. Here you can use the hole pattern and figure definitions of the CNC PILOT 640. This means that, for linear or circular patterns and simple figures (circles, rectangles, regular polygons, etc.), you only need a few more entries 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 perpendicular to the tilted plane—without distortion. This ensures simple verification of programmed hole patterns and milling contours. You also verify the tool movements in the Side View window. If you want to check the machining in the tilted plane with respect to the rotated contour or the face, 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 in the B axis. And do you want to see 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

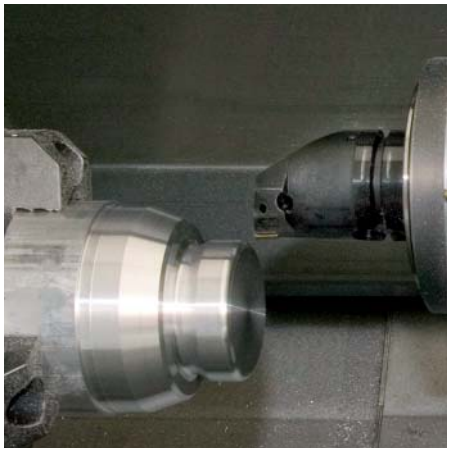
Flexible use of tools*
If your machine is equipped with a B axis, you can use your tools much more efficiently than before. On conventional lathes you need four different tools for longitudinal and transverse turning on opposing spindles. With a B axis, you can do it with a single tool.

You simply tilt the B axis and rotate the tool to the normal position or for machining from behind the workpiece—whichever is required for longitudinal or transverse turning on the main or opposing 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.

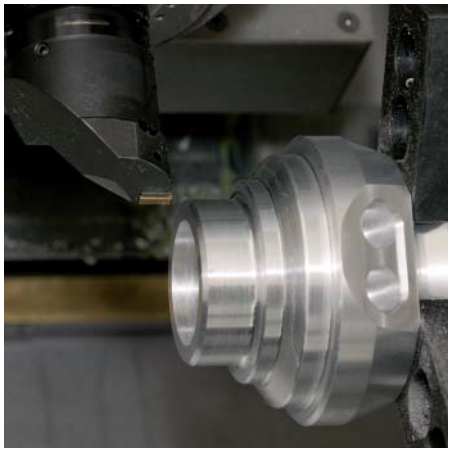
Tool-use flexibility is increased significantly when several tools are mounted in one holder. For example, with a roughing, finishing and recessing tool you can perform considerable parts of turning and recessing operations on a main and opposing spindle—without changing the tool. And programming is very easy. You simply indicate which tooth of the tool to use and then define the tilting angle and the tool position. And no more, because the CNC PILOT 640 already has the rest position and the data of each tool tooth in its database.

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

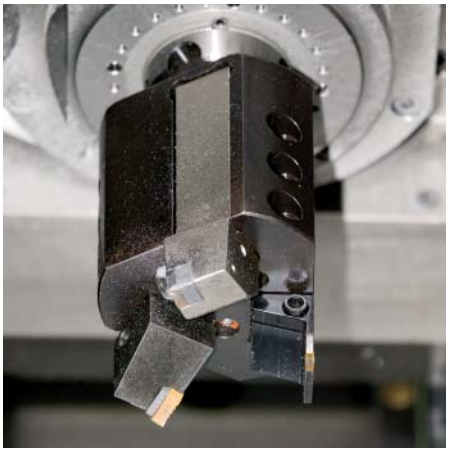
* The machine and CNC PILOT 640 must be adapted for this function.



Facing and ...



... longitudinal turning with the same tool ...



... and for several tools on one holder.

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

Workpiece machining in Teach-in mode
For simple, non-recurring tasks, reworking, 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.

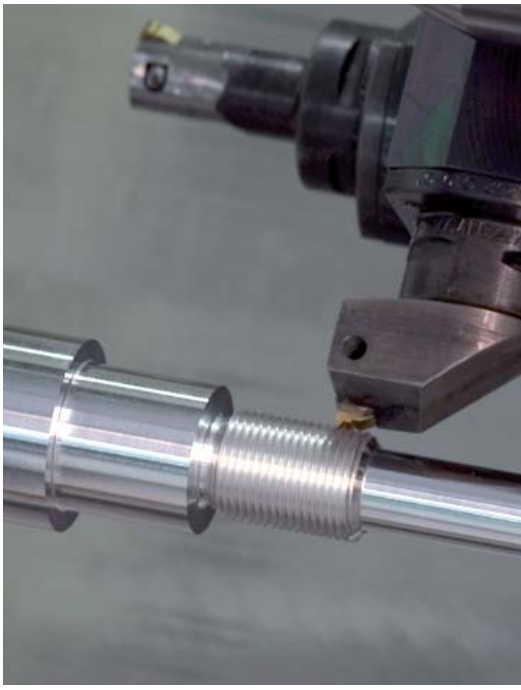
Thread-recut cycles
Even if the workpiece was unclamped, you can very easily recut a thread with the CNC PILOT 640.

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

Fewer calculations
In Teach-in mode, too, the CNC PILOT 640 automatically calculates the number of cuts for roughing, recessing, recess turning or thread cutting, and for pecking it determines the required number of infeeds. When turning a taper, you can enter either the starting point and end point, or the starting point and the taper angle—whichever is shown on your drawing.

Constant availability of tool data
The CNC PILOT 640 uses a tool database. Tool data, such as cutting radius, tool angle and point angle only have to be entered once to find the setting dimensions. 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. You can immediately work to dimension.

When turning a contour, the CNC PILOT 640 automatically compensates the deviations due to the cutting-edge radius. This increases the precision of your workpiece.



Single- or multi-start longitudinal, tapered or API thread

Undercuts as per DIN 76, DIN 509 E or DIN 509 F

Undercut form H, form K or form U

Parting

Threads, undercuts, parting

Longitudinal/transverse cutting for simple contours

Longitudinal/transverse cutting with plunging

Longitudinal/transverse ICP cutting for any contours

Longitudinal/transverse ICP contour parallel cutting

Area clearance—cutting and finishing

Technology data as default values
The CNC PILOT 640 saves the cutting data according to the criteria of workpiece material, tool material, and machining mode. As you have already entered the cutting material in the tool definition, you need only enter the material of your workpiece. This provides the cycle with all data required for setting default values for the cutting data.

Datums
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. Then a simple cycle call suffices to return to the tool change point.

Protective zone for the spindle
For every tool movement in the negative Z direction, the CNC PILOT 640 checks whether the programmed protective zone would 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 every Teach-in cycle knows the current workpiece blank, so that air cuts are avoided. This is possible with any kind of turning operation.

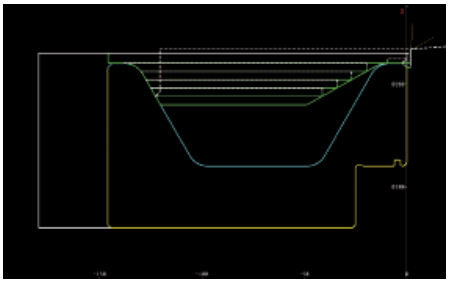
Radial/axial recessing for simple contours

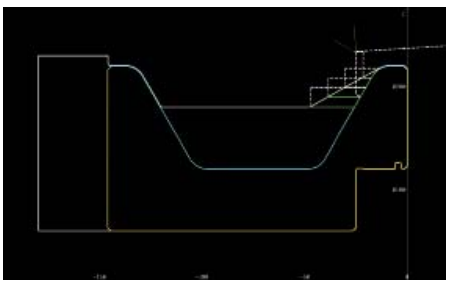
Longitudinal/transverse recess turning for simple contours

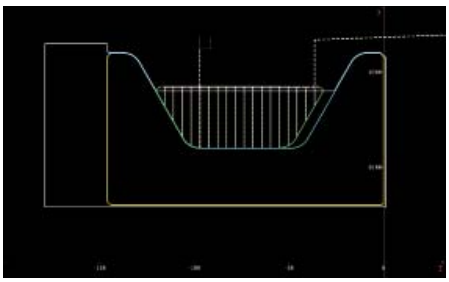
Radial/axial ICP recessing for any contours

Longitudinal/transverse ICP recess turning for any contours

Recessing and recess turning—cutting and finishing







Fast availability of tool data and cutting data
– The tool database and technology database of the CNC PILOT 640

Tool database
The CNC PILOT 640 can store 250 tools in the standard tool database. The tool database can be expanded to 999 tools (option).

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

The tool data are entered through prompts in which you enter parameters such as cutting-edge radius, tool angle and point angle, cutting material and the tool description. The input parameters are illustrated in context-sensitive help graphics.

Tool list
The CNC PILOT 640 shows all tools in a clearly laid out 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 when you're creating NC programs.

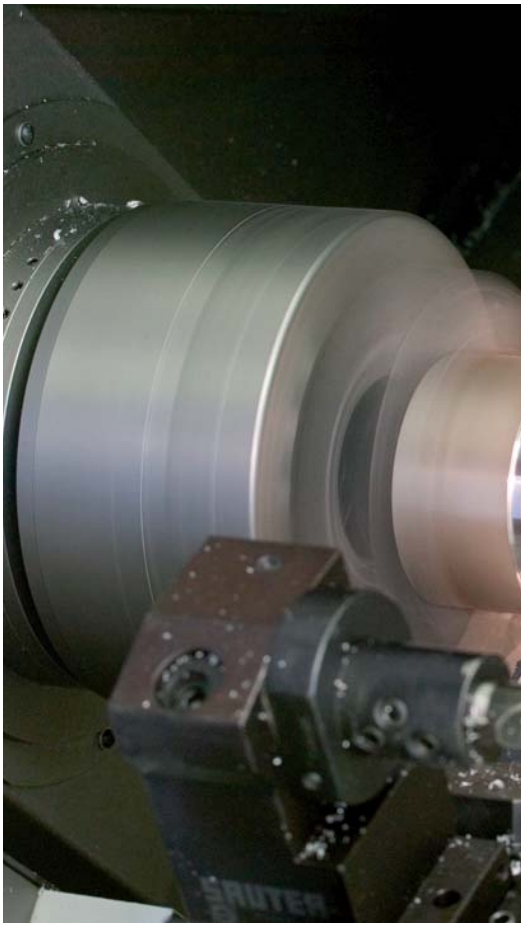
Wear compensation
The CNC PILOT 640 offers a simple and straightforward function for compensating tool wear in both the X and the Z axes. You can enter the compensation values at any time, even during machining or after machining the workpiece.

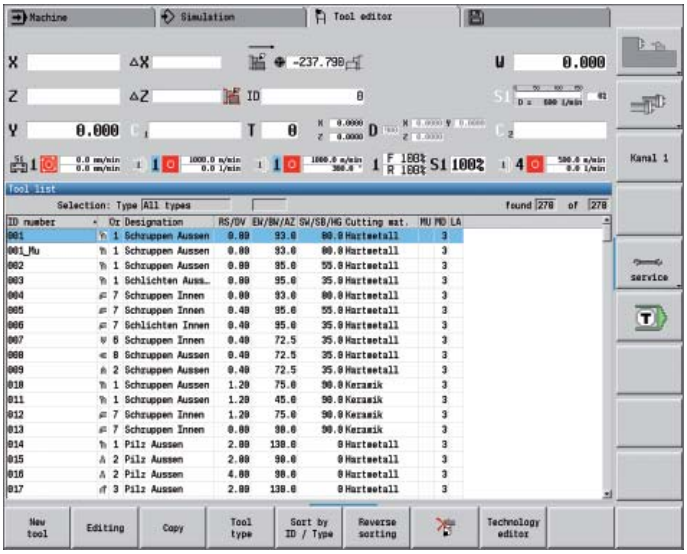
Tool measurement
The CNC PILOT 640 offers various possibilities for the measurement of tools directly on the machine:

- By touching the workpiece
- By means of an optical gauge* (option): the tool is manually traversed to the cross hairs of the measuring optics, and the value is saved with a keystroke.
- Through a tool touch probe* (option): the tool moves in measuring direction. 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.

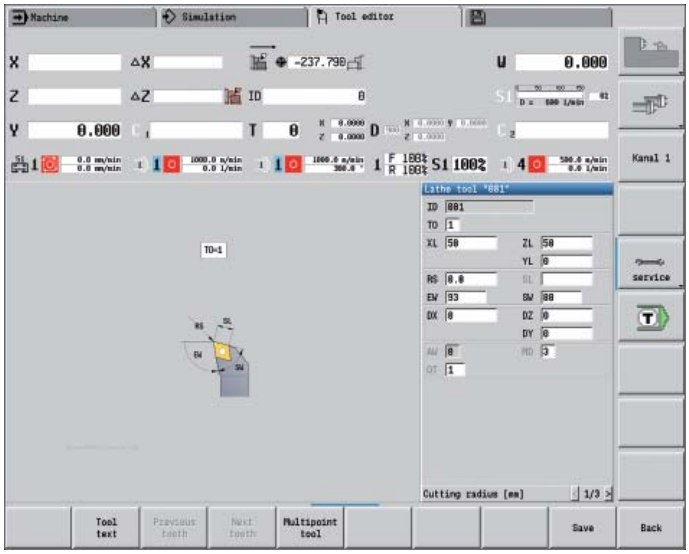
You can determine the tool data particularly easily, reliably and precisely during tool measurement with an optical gauge or tool touch probe.

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

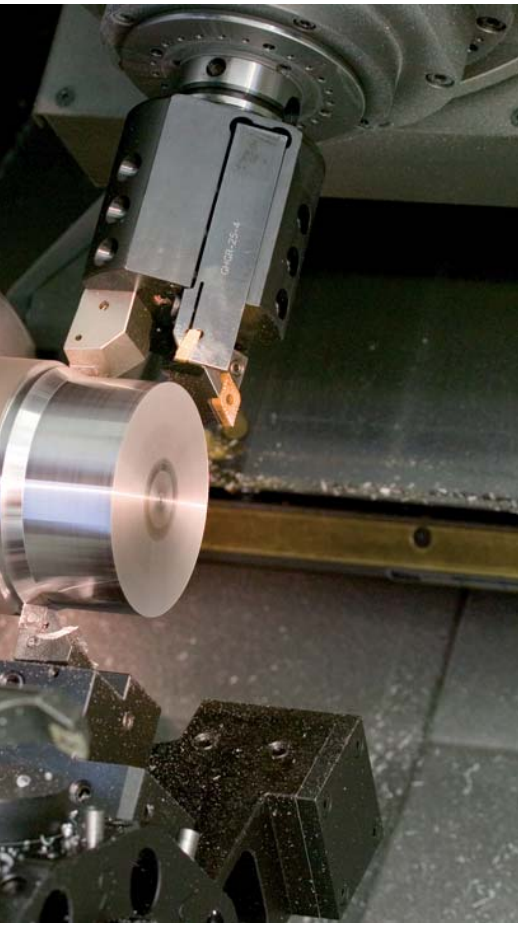




Tool management in the tool list



Tool editor



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

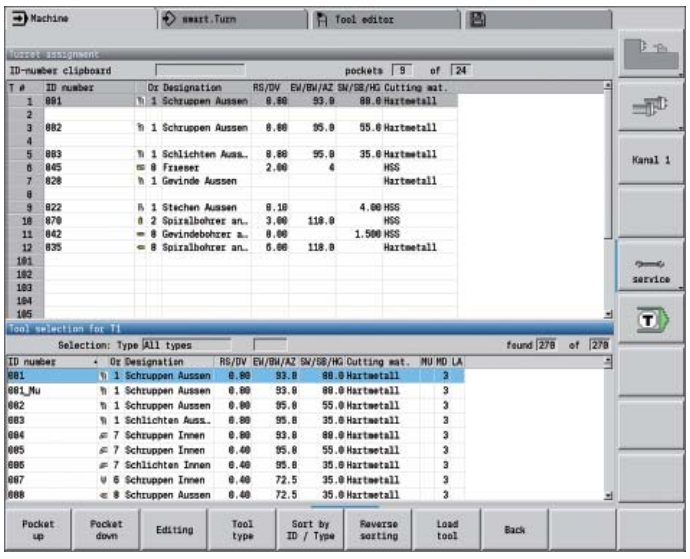
If you want to change the tool assignment or the tools in the turret, you can additionally display the entries of the tool database in the lower window. Now you need only select the desired turret pocket and choose the correct tool from the database. You can transfer the tool data to the turret assignment entry with a simple keystroke.

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 sister tool as soon as the active tool is used up. When the last tool of the replacement chain is used up, the CNC PILOT 640 stops program execution.

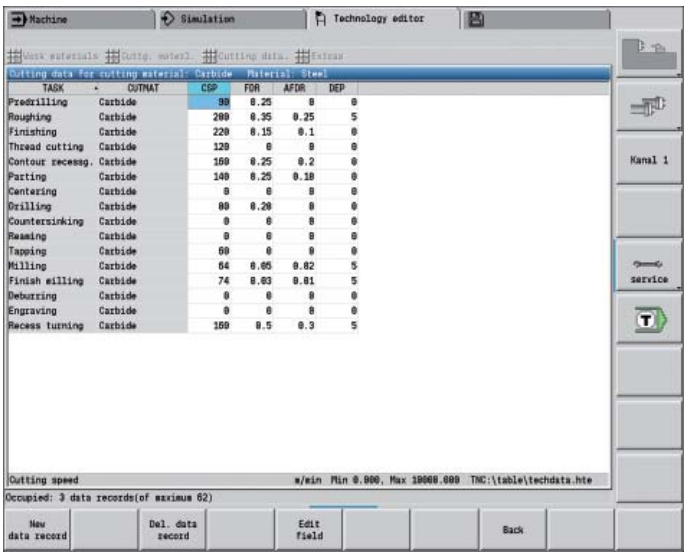
Technology data (option)
With the CNC PILOT 640 you need enter the cutting data only once. The control saves the cutting data according to the criteria of workpiece material, cutting material, and machining mode. Thanks to this three-dimensional table, the control always knows the correct feed rate and 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 during the tool description. 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 propose the correct values for your machining operation. You can use the suggested cutting parameters or adjust them if required.

In its standard version, you can store the cutting data for 9 workpiece-material/tool-material combinations in the technology database of the CNC PILOT 640. It can be expanded to 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
– Fast data transfer with the CNC PILOT 640

The networked CNC PILOT 640
The CNC PILOT 640 can be integrated into networks and connected with PCs, programming stations and other data storage devices. Even in its standard version, the CNC PILOT 640 features a latest generation Gigabit Ethernet interface. The CNC PILOT 640 communicates with NFS servers and Windows networks in TCP/IP protocol without needing additional software. The fast data transfer at rates of up to 1000 Mbit/s guarantee very short transfer times.

USB interface
The CNC PILOT 640 supports standard memory media with USB interface. Using USB memory media (such as memory sticks), you can quickly and easily exchange DXF contours, ICP contour descriptions, NC programs, tool parameters, etc., between systems that are not connected to each other.

All programs at a glance
After entering the path of the partner terminal, your own CNC PILOT 640 programs will be listed on the left side of the screen, and your partner's programs are on the right side. Now select the programs that you want to transfer and press the send or receive button. The data is transferred reliably and almost instantaneously.

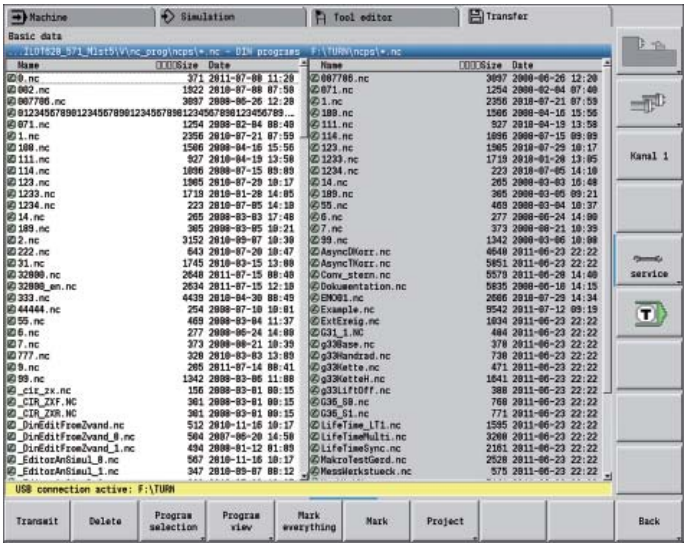
Transferring programs
An especially easy and convenient method of transferring data is to integrate the systems into your company network.

When transferring NC programs, the CNC PILOT 640 even considers the files associated with the cycle program, smart. Turn program or DIN PLUS program, such as contour descriptions, DIN macros or subprograms.

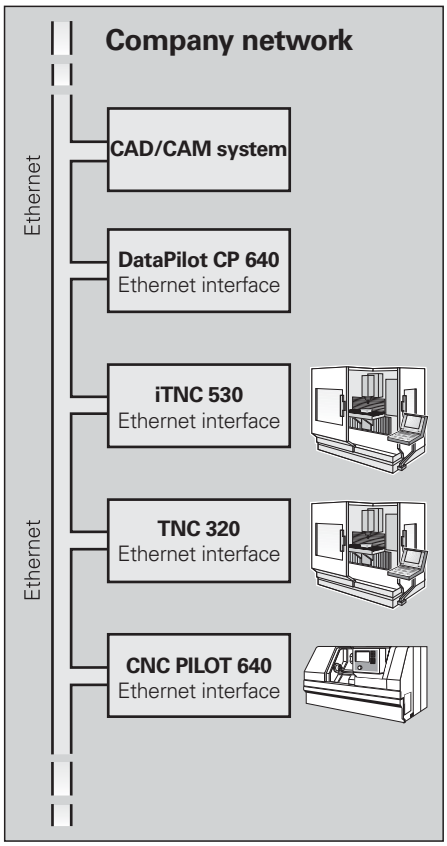
Exchanging tool data
Once you have acquired tool data, you may also transfer them. That is not only important for data backup: it also helps you when using the PC programming station DataPilot. The benefits: no redundant data acquisition; your files are always up to date.

Programs for data transfer
With the aid of the free PC software **TNCremo** from HEIDENHAIN and an Ethernet or other data interface you can
- transfer remotely stored part programs and tool tables in both directions and
- make backups.

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



Data transfer on the control



– Display various file formats on the control screen

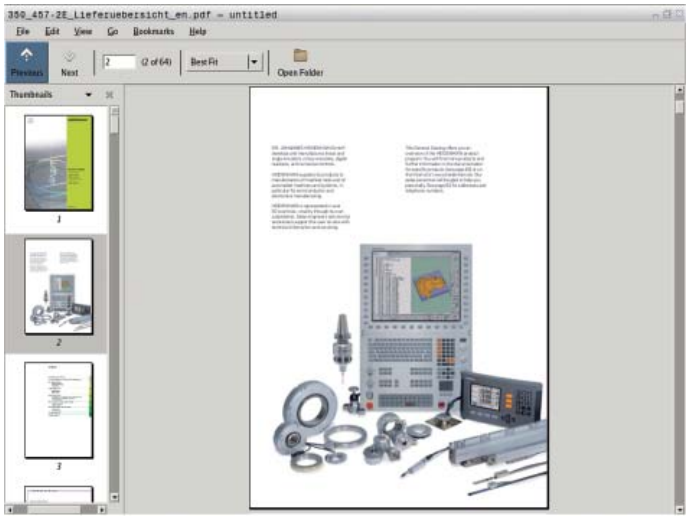
The new integrated PDF viewer enables the user to open PDF files directly on the control. The PDF format is a widely used data format that can be generated out of a great variety of applications. This enables you to easily view work instructions, drawings or other information in the CNC PILOT 640.

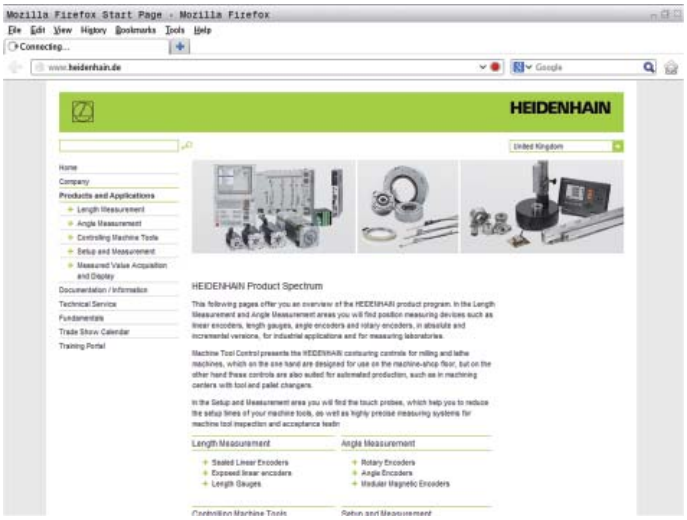
The integrated browser now lets you connect the CNC PILOT 640 to the Internet and access it directly from the control.

The following further file formats can also be opened directly on the CNC PILOT 640 with corresponding editors, and sometimes edited.

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

An operating panel with an integrated touchpad or an external USB pointing device is required for operation.





Open for communication
– The DataPilot CP 640 programming station

DataPilot CP 640 is the PC programming station for the CNC PILOT 640 and the organizing system for the workshop and design office.

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

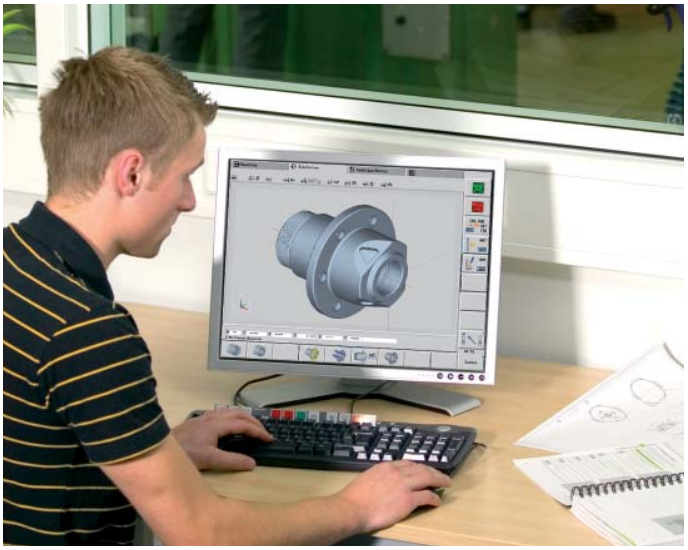
Creating programs
Programming, testing and optimizing smart.Turn and DIN PLUS programs with DataPilot on your PC substantially reduce idle machine times. You do not need to adjust your way of thinking, since you program and test with DataPilot in exactly the same way as on the lathe. DataPilot has the same software as the control. This ensures that a program created with DataPilot can be run on the machine immediately.

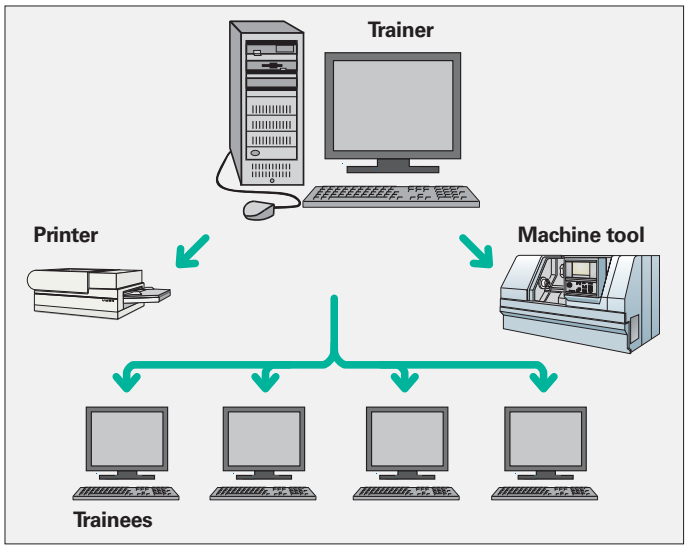
Archiving programs
Even though the CNC PILOT 640 has a large memory capacity, you should also back up your programs on an external system. The CNC PILOT 640 features a USB and an Ethernet interface. This enables you to integrate the CNC PILOT 640 into your existing network or to connect the DataPilot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

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

Training with DataPilot CP 640
Because DataPilot CP 640 is based on the same software as the CNC PILOT 640, it is ideally suited for apprentice and advanced training. Programming and program testing on the DataPilot PC function exactly the same as they do on the machine. DataPilot even simulates setup functions such as defining the workpiece datum, measuring tools or running individual cycles or smart.Turn or DIN PLUS programs. This gives the trainee the experience needed to enable him to safely operate the machine later.

System requirements
DataPilot runs on PCs with the Windows XP, Windows Vista, Windows 7 or Windows 8 operating systems.





– Fast availability of all information

Do you have questions on 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 show the user documentation in a separate window.

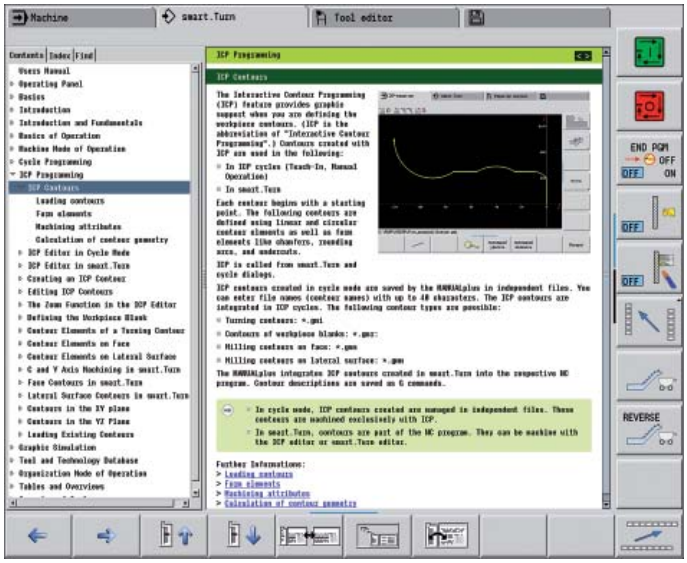
You activate TURNguide simply by pressing the Info key on the keyboard.

TURNguide usually displays the information in the immediate context of the element in question (context-sensitive help). This means that you immediately receive the relevant information. The function is particularly helpful for the programming of cycles. The respective operating method is explained in detail in an open dialog window when you press the Info key.

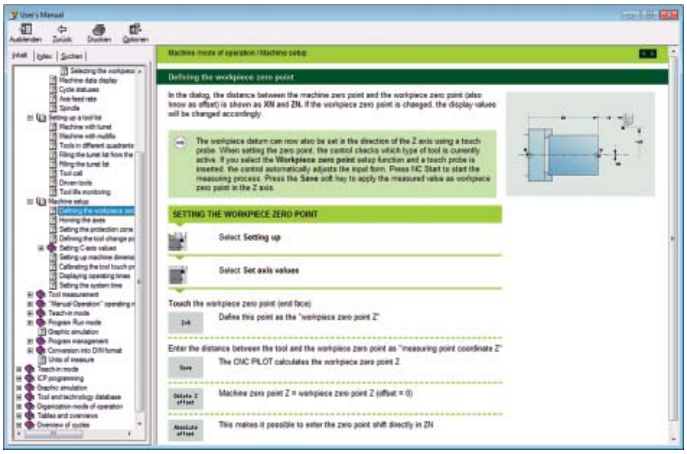
You can download the documentation in the desired language from the HEIDENHAIN homepage into the corresponding language directory on your control.

The following manuals are available in the help system:

- CNC PILOT 640 User's Manual
- User's Manual for smart.Turn and DIN programming
- User's Manual for the CP 640 DataPilot (only included in the programming station)



TURNguide integrated in the control, e.g. on the CNC PILOT 640 ...



... or at the programming station.

Workpiece measurement
– Setup, presetting and measuring with touch trigger probes

Inspecting workpieces for proper 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 in place of a tool:

- Check whether all machining operations were conducted correctly
- Determine infeeds for finishing
- Detect and compensate tool wear
- Check workpiece geometry and sort parts
- Log measured data
- Ascertain the machining error trend

Workpiece touch probes from HEIDENHAIN help you to reduce costs in the workshop and in series production: Together with the CNC PILOT 640, touch probes can automatically perform setup, measuring and inspection functions.

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

HEIDENHAIN touch probes* for workpiece measurement are available in different versions. The ruby ball tips are available in several diameters, and the styli in different lengths.

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

Touch probes with **cable connection for signal transmission** for machines with manual tool change and for grinding machines and lathes:

TS 260 – New generation, axial or radial cable

Touch probe with **radio or infrared signal transmission** for machines with automatic tool change:

TS 460 – New generation standard touch probe for radio and infrared transmission, with compact dimensions
TS 444 – Battery-free voltage supply through integrated air turbine generator from compressed air, for infrared transmission, with compact dimensions
TS 740 – High probing accuracy and repeatability, low probing force, with infrared transmission

More information about workpiece touch probes is available on the Internet at *www.heidenhain.de* or in the Product Overview *Touch Probes – New Generation.*

TS 460

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

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

TT 160 – New generation, signal transmission to the NC over connecting cable
TT 460– New generation, signal transmission over radio and infrared beam to transmitter/receiver unit

The SE 660 is a common transmitter/receiver unit 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 physical probing of a tool. At that moment the TT generates a trigger signal that is transmitted to the control, where it is processed further. The trigger signal is generated through a wear-free optical switch that ensures high reliability.

With their rugged design and high degree of protection, these tool touch probes can be installed directly within the machine tool's work envelope and make it possible to calibrate the tool right in the machine. This way you can determine the tool dimensions quickly, easily and, above all, very precisely. The TT tool touch probes are the ideal supplement to improve the efficiency and quality of your production.

SE 660

TT 160

More information about tool touch probes is available on the Internet at *www.heidenhain.de* or in the Product Overview *Touch Probes – New Generation*.

Overview
– User functions

User functions	Default	Option	
Configuration	•		Basic version: X and Z axis, 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)
		132+0-6	Opposing spindle
	•		Digital current and speed control
Operating modes			
Manual Operation	•		Manual slide movement through axis-direction keys, intermediate switch or electronic handwheels
	•		Graphic support for entering and running cycles without saving the machining steps in alternation with manual machine operation
		11	Thread reworking (thread repair in a second workpiece setup)
Teach-In mode		8	Sequential linking of fixed cycles, where each cycle is run immediately after input, or is graphically simulated and subsequently saved
Program Run			Both in single-block and full-sequence modes
	•		DIN PLUS programs
		9	smart.Turn programs
		8	Cycle programs
Setup Functions	•		Workpiece datum setting
	•		Defining the tool-change point
	•		Definition of protection zone
	•		Defining 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, threads with variable pitch
		8+55	Cycles for axial and radial drilling, pecking 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 polygons as well as for complex contours defined 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, boring and milling operations with the C axis
		8	Context-sensitive help graphics
		8	Transfer of cutting values from technology database
		8	Use of DIN macros in cycle programs
		8+9	Conversion of cycle programs to smart.Turn programs

User functions	Default	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	Graphic 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 by superimposition later
		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 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 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: Import of contours for lathe 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 fillable forms and 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 technology database
		9	Units for all turning and recessing operations for simple contours and ICP contours
		9+55/70	Units for boring, drilling and milling operations with the C or Y axis for simple holes, milling contours and drilling and milling patterns as well as those programmed with ICP
		9+55	Special units for activating/deactivating the C axis, subroutines and section repeats
		9+55/70	Verification graphics for blank and finished part and 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 working plan generation with: • Automatic tool selection • Automatic turret assignment • Automatic calculation of cutting data • Automatic generation of the machining sequence in all working planes, also for C-axis machining (with option 55) and Y-axis machining (with option 70) • Automatic cutting limitation through chucking equipment • Automatic generation of work blocks for rechucking during full-surface machining • Automatic generation of work blocks for rear-face machining (with option 132)

Overview
– User functions (continued)

User functions	Default	Option	
DIN PLUS programming	•		Programming in DIN 66025 format
	•		Extended 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 boring, drilling and milling with the C axis
		70	Powerful fixed cycles for boring, drilling and milling with the Y axis
	•		Subprograms
			Technology functions for full-surface machining:
	•		– Moving to a fixed stop
	•		– Parting control
		131/132	– Spindle synchronization
		132	– Converting and mirroring
	•		– 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
Program verification graphics	•		Graphic simulation of the cycle process, or of the cycle, smart.Turn or DIN PLUS program
	•		Display of the tool paths as pencil-trace or cutting-path graphics, special identification of the rapid traverse paths
	•		Machining simulation (2-D material-removal graphic)
		55	Side or face view, or 2-D view of cylindrical surface for verification of C-axis machining
	•		Display of programmed contours
		54	View of the tilted plane (B-axis machining)
	•		View of face and YZ plane for verification of Y-axis machining
	•		Three-dimensional display of the workpiece blank and finished part
		132	Simulation of mirrored contours for rear-face machining
	•		Shifting and magnifying functions
	•		Block scan in the simulation
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 traverse movements of the X and Y axes synchronous 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—detecting tool wear and breakage during machining

User functions	Default	Option	
Tool database	•		For 250 tools
		10	For 999 tools
	•		Tool description can be entered for every tool
	•		Automatic inspection of tool-tip position with respect to the contour
	•		Compensation of tool-tip position in the X/Y/Z plane
	•		High-precision correction via handwheel, capturing compensation values in the tool table
	•		Automatic tool-tip and cutter radius compensation
	•		Tool monitoring for lifetime of the insert (tool tip) or the number of workpieces produced
		10	Tool monitoring with automatic tool change after expiration tool life
	•		Management of multipoint tools (multiple inserts or reference points)
	•		Support of quick-change systems
Technology database		8/9	Access to cutting data after definition of workpiece material, cutting material and 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 default values.
		8/9	9 workpiece-material/tool-material combinations (144 entries)
		10	62 workpiece-material/tool-material combinations (992 entries)
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	Remark
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	**Additional control loops 1 to 8**
8	Software option 1 Teach-in	01	632226-01	**Cycle programming** • Contour description with ICP • Cycle programming • Technology database with 9 workpiece-material/tool-material combinations
9	Software option 2 smart.Turn	01	632227-01	**smart.Turn** • Contour description with ICP • Programming with smart.Turn • Technology database with 9 workpiece-material/tool-material combinations
10	Software option 3 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	Software option 4 Thread recutting	01	632229-01	**Thread** • Thread recutting • Handwheel superimposition 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**

– Accessories

Option number	Option	As of NC software 688946-688947-	ID	Remark
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	Opposing spindle	01	806275-01	**Opposing spindle** (spindle synchronism, rear-face machining)
135	Synchronizing functions	03	1085731-01	Expanded synchronization of axes and spindles
143	Load Adapt. Control Control	01	800545-01	LAC: Dynamic adaptation of the control parameters
151	Load monitoring	03	1111843-01	Monitoring of the tool load

Accessories	
Electronic handwheels	One portable **HR 510** 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 444:** Touch trigger probe with infrared transmission or • **TS 740:** Touch trigger probe with infrared transmission
DataPilot CP 640, MP 620	Control software for PCs 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 with TFT color flat panel display (19-inch or 15-inch), or MC main computer with integrated TFT color flat panel display (15-inch) CC, UEC or UMC controller units TE operating panel (19-inch or 15 inch) with – Integrated machine operating panel – Rapid-traverse override, feed-rate override and spindle speed override potentiometer – Electronic handwheel
Operating system	HEROS 5 real-time operating system for machine control
NC program memory	1.8 GB (on compact flash memory card CFR)
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 2 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 pairs × ball screw pitch) at f_{PWM} = 5000 Hz
Spindle	Maximum 60 000 rpm (with 2 pole pairs)
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: minimum 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 5 x USB (1 x front USB 2.0; 4 x rear 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

Vollständige und weitere Adressen siehe www.heidenhain.de
For complete and further addresses see 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 31-1345

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 3026, Australia
E-mail: vicsales@fcrmotion.com

BE **HEIDENHAIN NV/SA**
1760 Roosdaal, Belgium
www.heidenhain.be

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

BR **DIADUR Indústria e Comércio 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.**
Tel Aviv 61570, 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

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**
01133 Kiev, Ukraine
www.heidenhain.ua

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

VE **Maquinaria Diekmann S.A.**
Caracas, 1040-A, Venezuela
E-mail: purchase@diekmann.com.ve

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

Zum Abheften hier falzen! / *Fold here for filing!*

895949-25 · 5 · 9/2015 · F&W · Printed in Germany