HEIDENHAIN

Votre agent :

araxe

72, rue Yves le Coz
78000 - VERSAILLES

tél : 01 30 21 48 49
contact@araxe.com

http://www.araxe.com

CNC PILOT 620

The Contouring Control for CNC Lathes

August 2010

Start smart

For many years now, the CNC PILOT has been proving itself in daily use on lathes and has earned a reputation for convenient NC programming.

With the introduction of smart.Turn, HEIDENHAIN has made yet another step forward toward greater ease of use. Easily understandable program entry in forms, default setting for global values, numerous selections and straightforward graphic support ensure fast and easy operation.

The new smartTurn interface is based on the proven HEIDENHAIN-DIN PLUS. Because smart.Turn produces DIN PLUS programs. It provides both the NC programmer and the machine operator with all relevant information during program run.

Contents

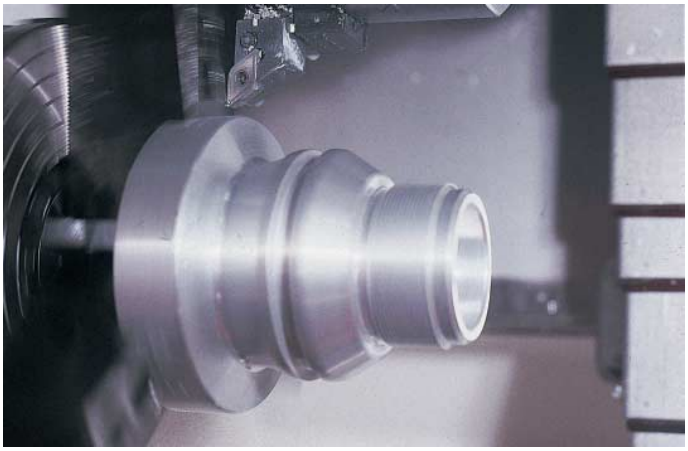
The CNC PILOT 620...

Compact and Versatile
– CNC PILOT 620, the Control for CNC Lathes

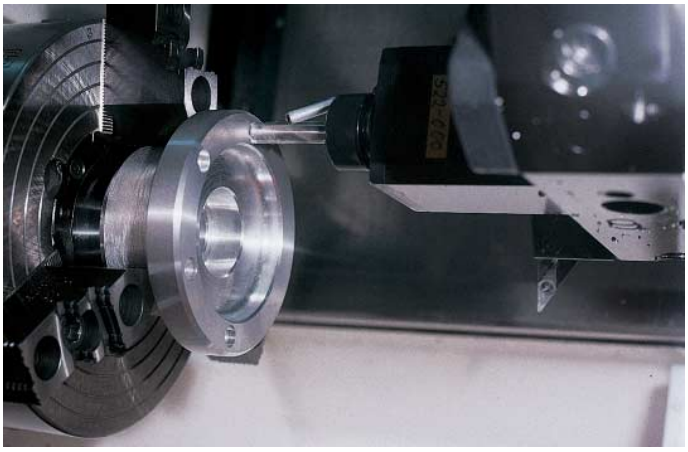
Thanks to its flexible design and numerous programming features, CNC PILOT always gives you optimum support. Regardless of whether you are manufacturing single parts or batches, simple or complex workpieces, the CNC PILOT always provides the right support. The CNC PILOT is characterized by its simple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

The CNC PILOT 620 was conceived for CNC lathes. It is suitable for horizontal and vertical lathes.

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



From simple jobs on a compact machine...



... to complex tasks

... to large batch production

Regardless of whether you are turning simple parts or complex workpieces, the CNC PILOT 620 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.

Well Designed and User Friendly
– The CNC PILOT 620 in Dialog with the User

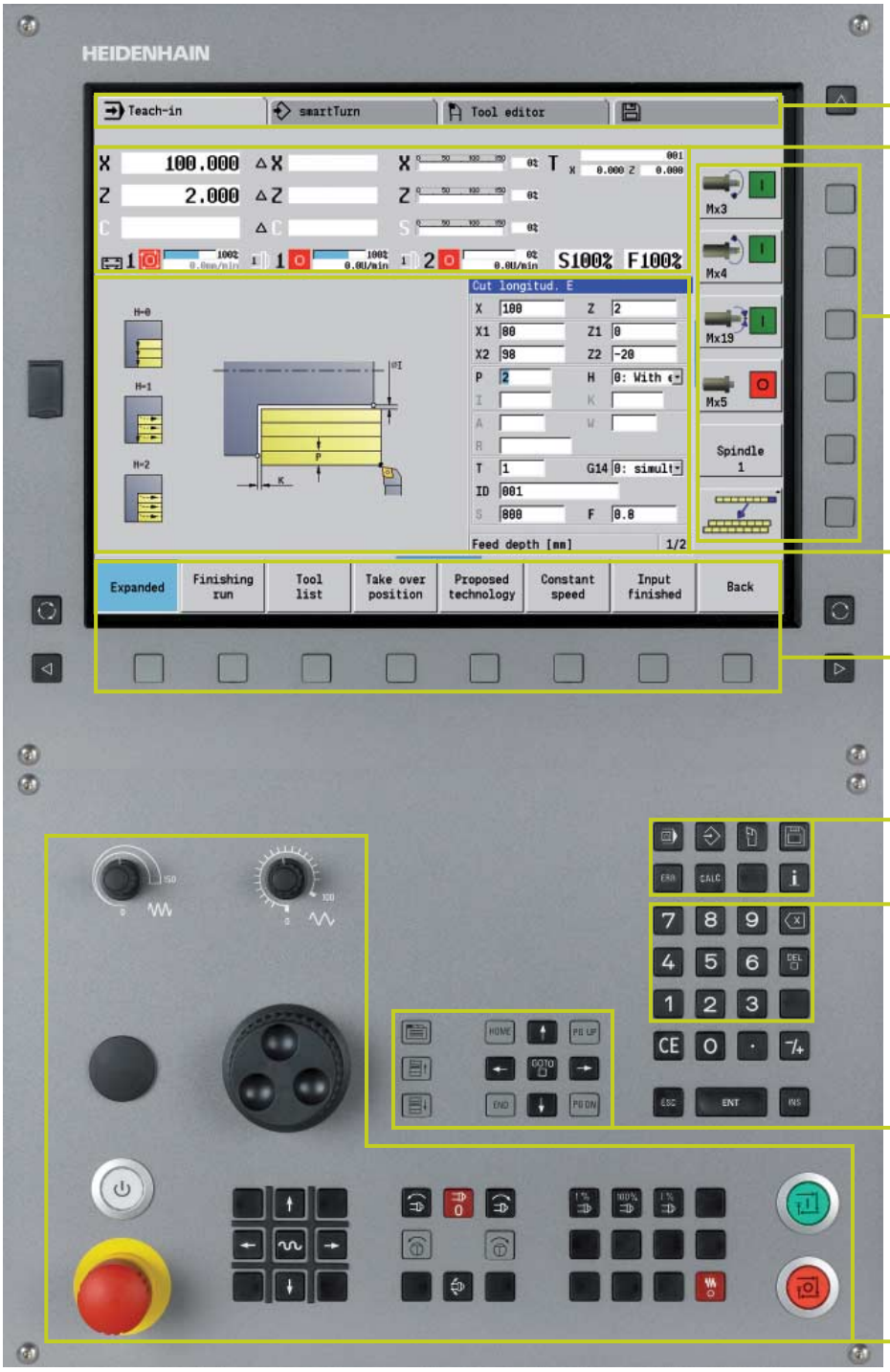
The screen
The TFT 15 inch color flat-panel display, shows a clear overview of all relevant information for programming, operating and monitoring the machine tool and control: program blocks, messages, error messages, etc.

During program input the required parameters are illustrated in help graphics, and during test run the CNC PILOT simulates the cutting process 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 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.



Keys on the monitor

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

Soft keys for selecting functions on screen

Shifts between soft-key rows

Display of operating modes

Display of the 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 components

Keys on the control panel

Operating mode keys

Machine operating modes

Programming modes

Tables for tool data and technology data

Parameters, file management, transfer, diagnostics

Straightforward input forms for cycle programming, smart.Turn programming or DIN PLUS programming. The input parameters are illustrated in help graphics during NC programming.

Unambiguous function keys for NC programming

Navigation keys

Keys for operating modes and functions

PG UP PG DN Screen/page up/down

Keypad for numerical input and fast, direct menu selection

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

Navigation keys

CALC Calculator

ERR Call up messages and errors

Info key

Machine operating panel with override potentiometer and electronic handwheel

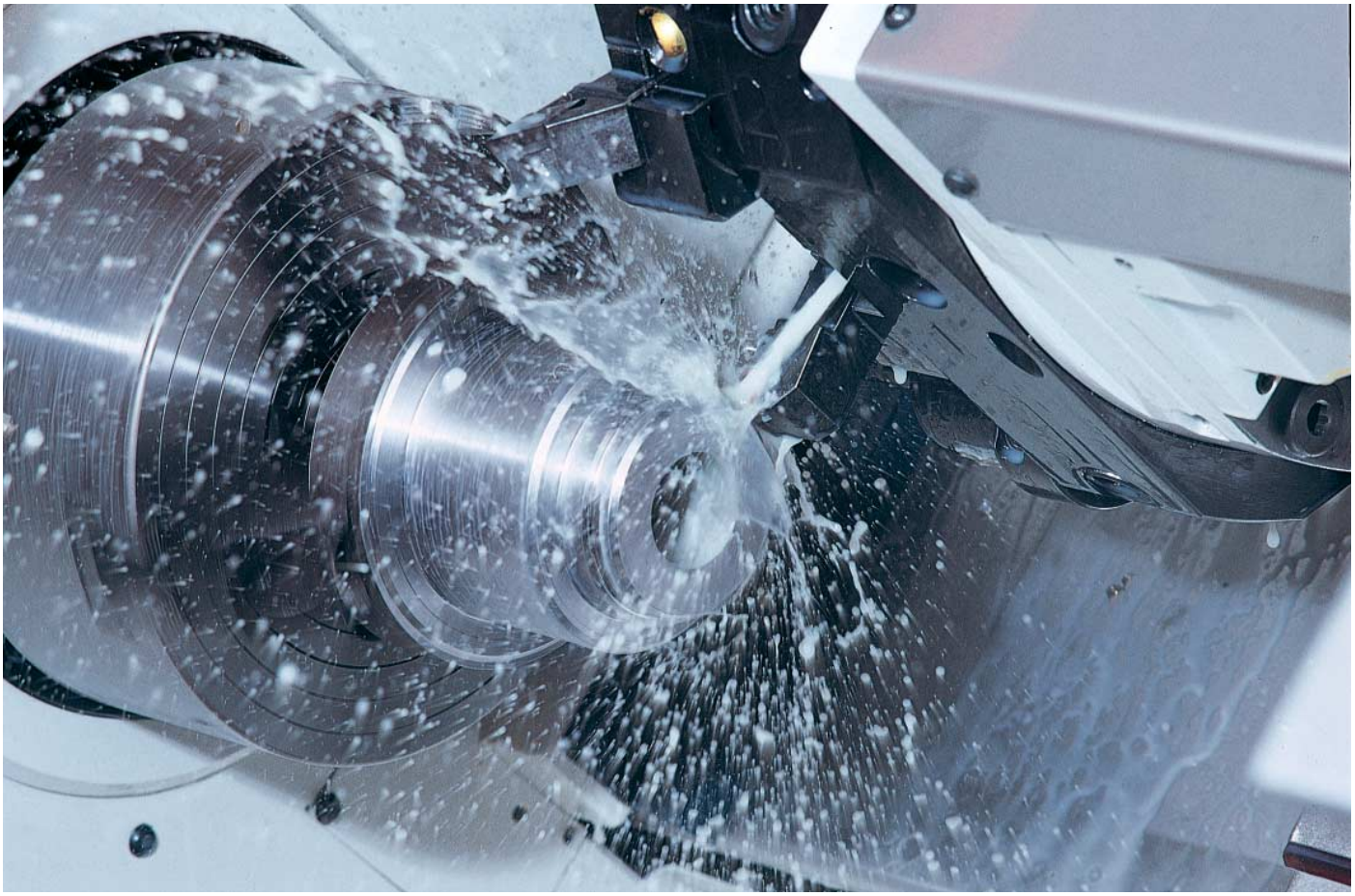
GOTO Display block or activate special functions, such as input options or text input

Quick and Reliable Machining with High Contour Fidelity
– Uniformly Digital Control Design

Thanks to its digital design, the CNC PILOT 620 has control over the machine's entire drive system. Not only does the field-proven digital drive technology from HEIDENHAIN make high contour fidelity and rapid machining at high speeds possible, but also all control components of the CNC PILOT 620 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 620. The digital motor control makes it possible to attain very high feed rates.

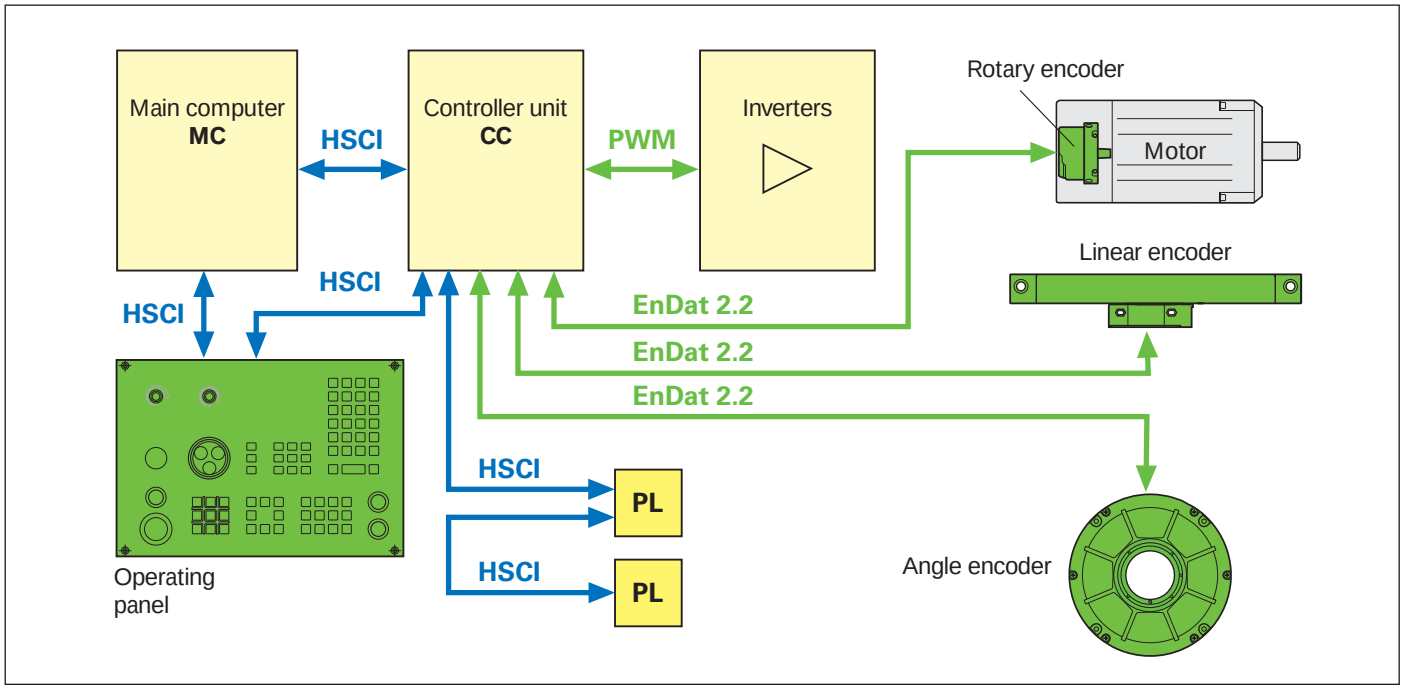
High contour fidelity
The CNC PILOT 620 dynamically calculates the contour in advance. This enables it to adapt the axis velocities to the contour transitions. It controls the axes with special algorithms that ensure path control with the required limits to velocity and acceleration.



High availability
In the uniformly digital control concept of the CNC PILOT 620, 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 EnDat 2.2, 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—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.



Effective, Clearly Organized and Flexible
– Simple Programming with smart.Turn (Option)

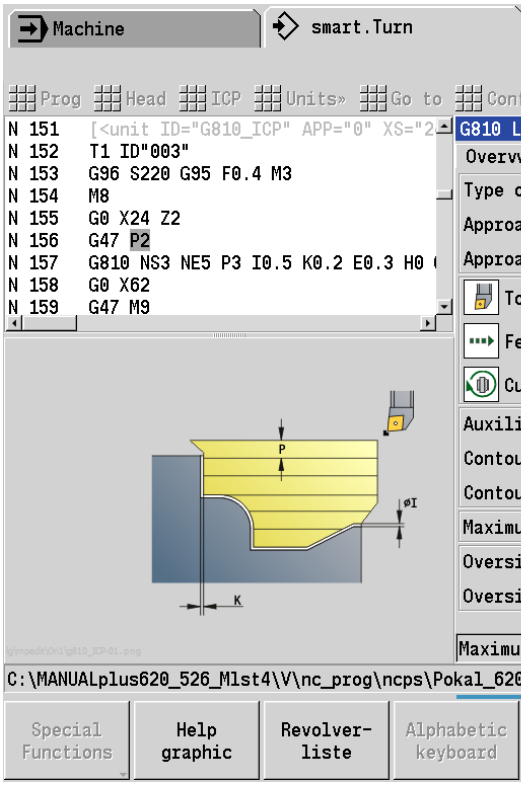
Has the safety clearance been correctly entered, is the speed limit taken into account, how are oversizes defined? All this needs to be considered not only by the beginner, but also by the experienced NC programmer when creating conventional DIN programs.

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.

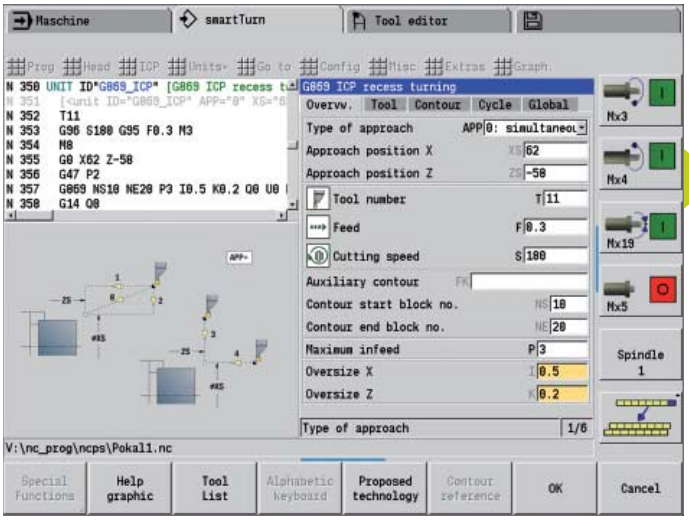
The smart.Turn principle gives you the reassurance that the working block is defined correctly and completely. In the NC program, smart.Turn lists the DIN PLUS commands of the unit. This gives you an overview of all working-block details at any time.

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.

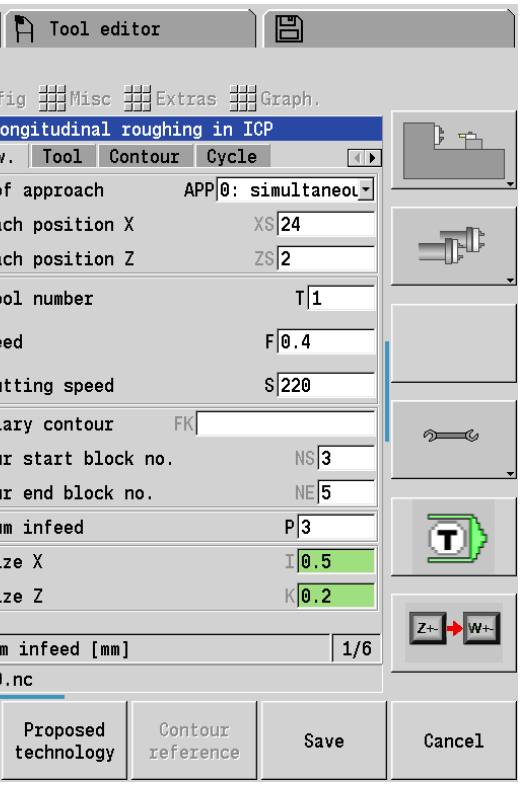
By the way: You do not need to stop the manufacturing process for programming with smart.Turn. You can create and test the smart.Turn program while the program is running.



Straightforward dialogs; help graphic illustrate the parameters.




```
N 400 UNIT ID"G869_ICP" [G869 ICP recess turning]
N 402   T11
N 403   G96 S400 G95 F0.3 M3
N 404   M8
N 405   G0 X75 Z3
N 406   G47 P3
N 407   G869 NS3 NE22 P3 I0.8 K1 Q0 U0 H0 V0
N 408   G14 Q0
N 409   G47 M9
N 410 END_OF_UNIT
```



Structured and easy-to-read
Clearly structured and easy-to-read—these are the characteristics of smart.Turn programs. It 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:

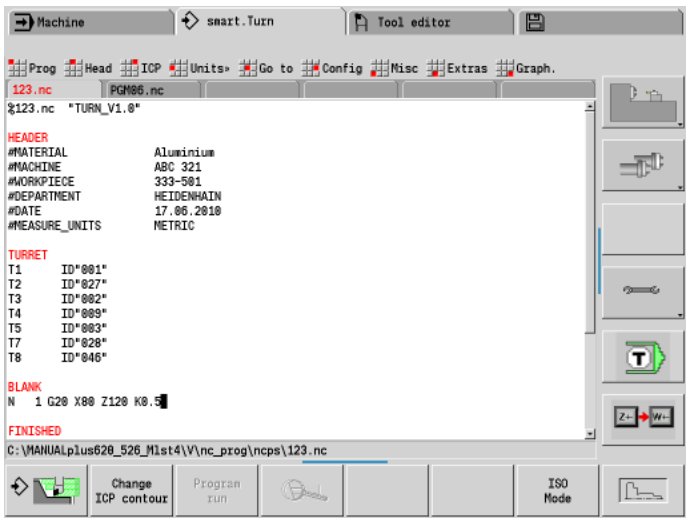
- Program header
- Tool assignment in the turret
- Workpiece-blank definition
- Description of machined part
- Individual machining steps

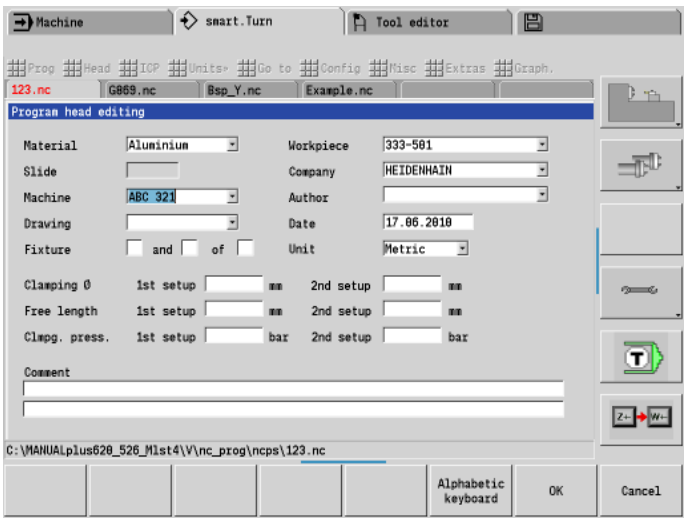
The smart.Turn technique not only ensures that the program is easy to read, but it also makes it possible to save all information required for producing the workpiece in the NC program.

Production data at a glance
The program head includes all important information on the workpiece, e.g. drawing number, date, programmer, material, clamping, 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 6 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.





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.

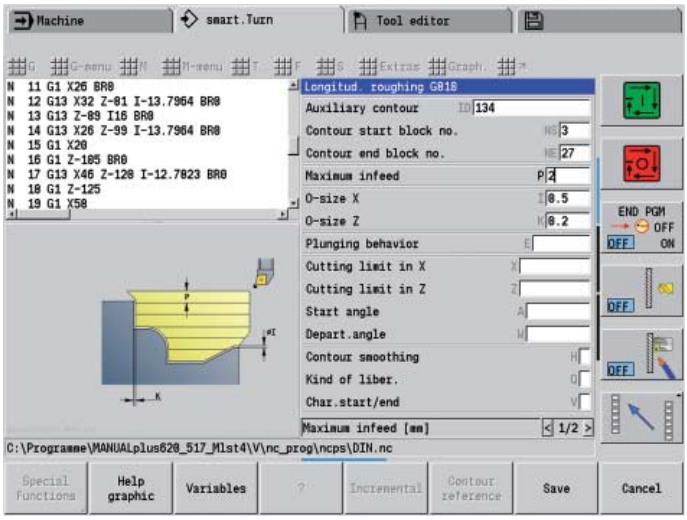
In the NC program, smart.Turn lists the DIN PLUS commands of the unit. This not only gives you an overview of all working-block details, but you also have a clearly legible and well-structured NC program.

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 interactive graphics. 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 620 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 are adapted automatically 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 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 smart.Turn with all data for setting default values for the cutting data.



– 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.

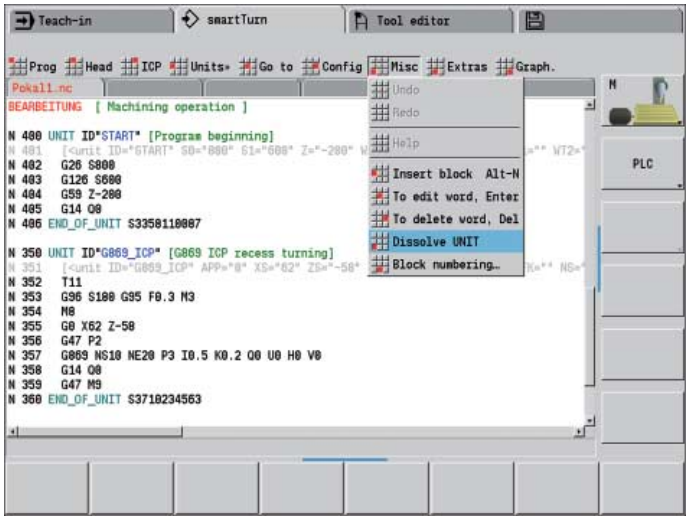
Because the units are based on DIN PLUS, you can break up a unit into blocks at any time to modify and optimize the resulting DIN PLUS program section.

Of course the CNC PILOT 620 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 machine 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 attain a significantly better effectiveness and flexibility than in conventional part programming as per DIN 66025.



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 you simply import the contour directly if the drawing is available in DXF-format.

Contour programming with ICP
You define an ICP contour in the graphic editor by entering the contour elements step-by-step. 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 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.

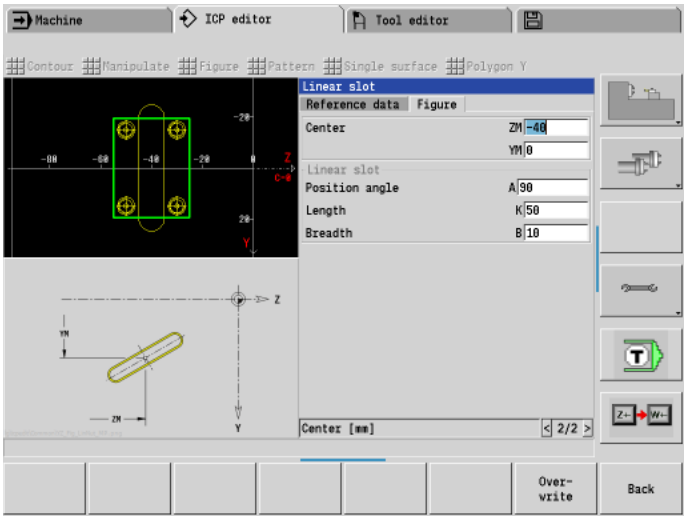
As long as they are mathematically defined, the CNC PILOT calculates missing coordinates, intersections, center points, etc. If there are multiple solutions, you can select the desired solution from the mathematically possible variants displayed. You can modify or change existing contours.

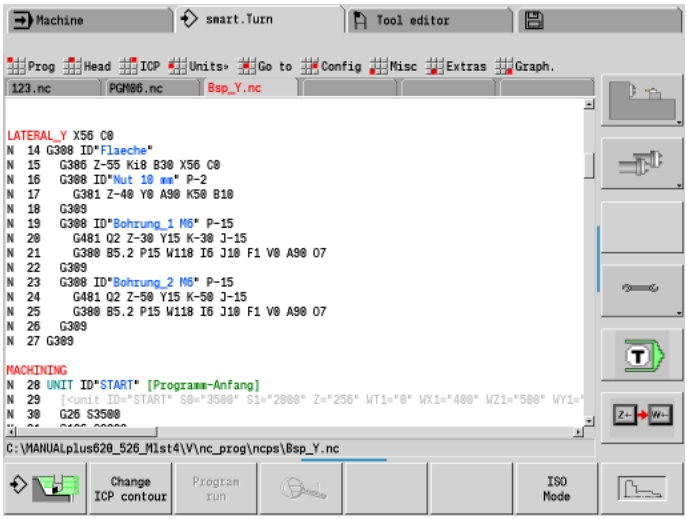
Superimposing form elements
The ICP editor recognizes the chamfer, rounding and undercut (DIN 76, DIN 509 E, DIN 509 F, etc.) form elements. You can enter these form elements in the course of the sequential contour definition. However, it is often easier to first define the “rough” contour, and then to superimpose the form elements. Select the contour corner on which the form element is to be placed, then insert 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 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. In 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.

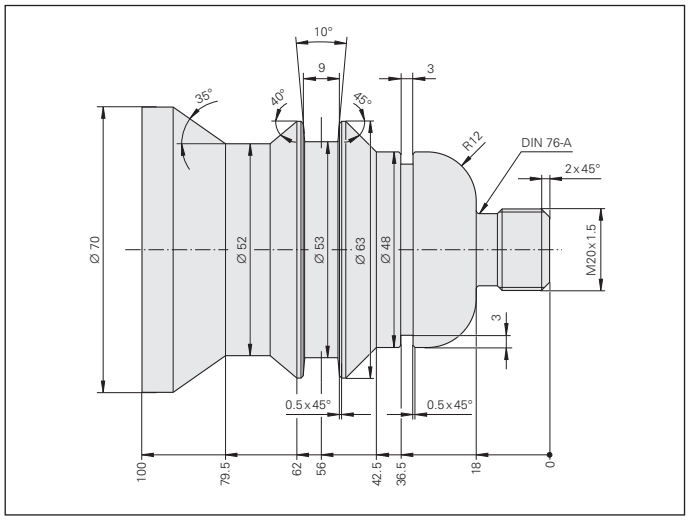
The ICP editor can be called immediately from within smart.Turn.

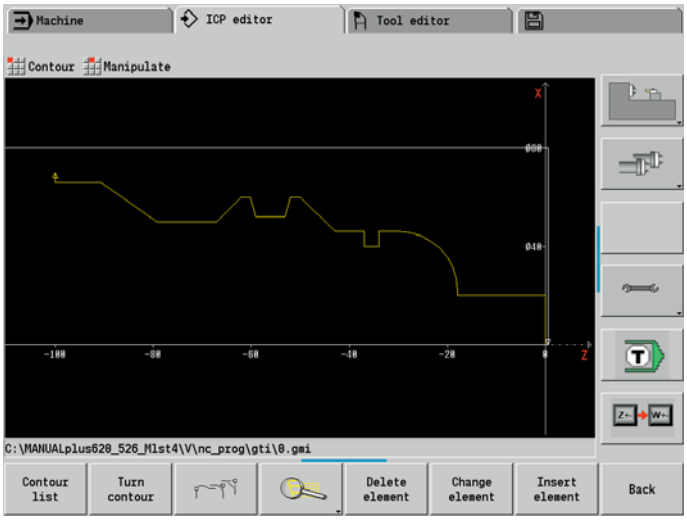




DXF import of contours (option)
Why should you painstakingly enter contour elements if the data already exists in the CAD system? ICP makes it possible to import contours in DXF format directly into the CNC PILOT 620. Not only does this save time otherwise spent on programming and testing, but you can 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 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.





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 620 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
- approach and departure movements,
- machining sequence,
- proportioning of cuts,
- resulting contour.

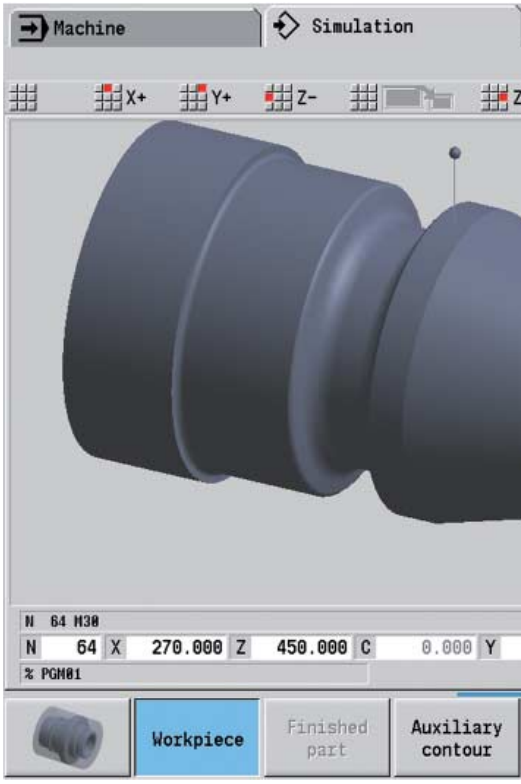
In the graphic simulation you can display the tool cutting edge. You see the cutting-edge radius, the cutting-edge width and the cutting-edge position with their actual dimensions. This helps to recognize machining details or collision risks in time.

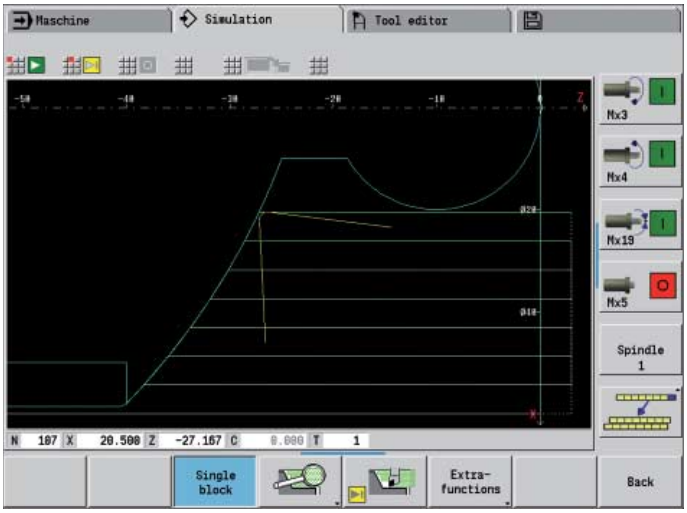
Wire-frame or cutting-path graphics, machining simulation
The CNC PILOT supports various views of the tool paths and the machining process. You can choose the type of verification best suited to the tool or machining process used.

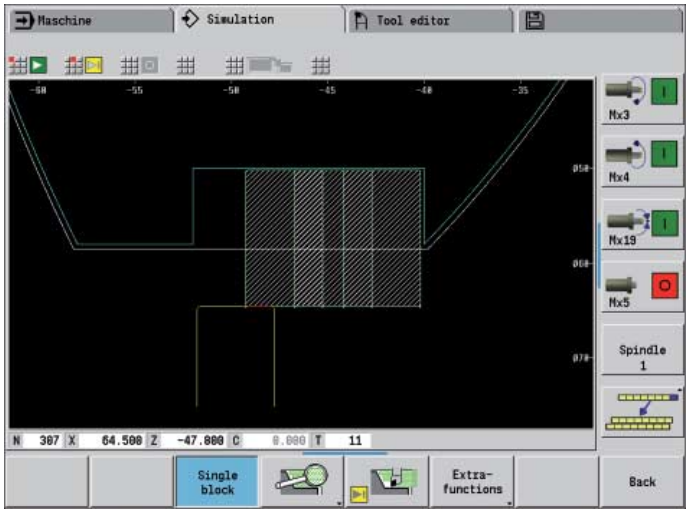
The **wire frame graphics** are particularly convenient if you only need a quick overview of the approaching and departing movements and the proportioning of cuts. The wire frame graphics illustrate the path of the theoretical cutting point.

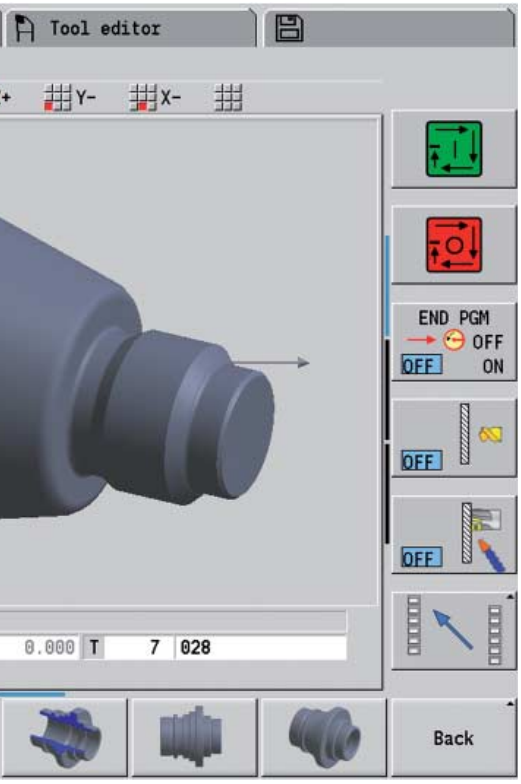
A more accurate contour verification is provided by the **cutting-path graphics**. The cutting path graphics account for the exact geometry of the tool tip. You immediately see if material was left behind, the contour is damaged or the overlaps are too large. The cutting path graphics is especially useful 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 simulates every tool movement at the programmed cutting speed and removes the material.









3-D view

With the 3-D graphics the workpiece blank and finished part are shown in a solid-model view You can rotate the graphic about the principal axes or display them in a sectional view.

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 also simulates machining on the end 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 depicts C-axis machining of the cylindrical surface as an "unrolled" plane surface.

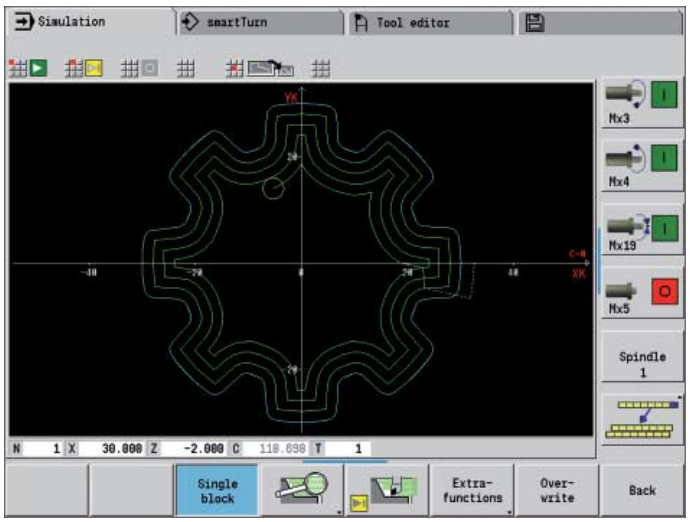
Zoom function

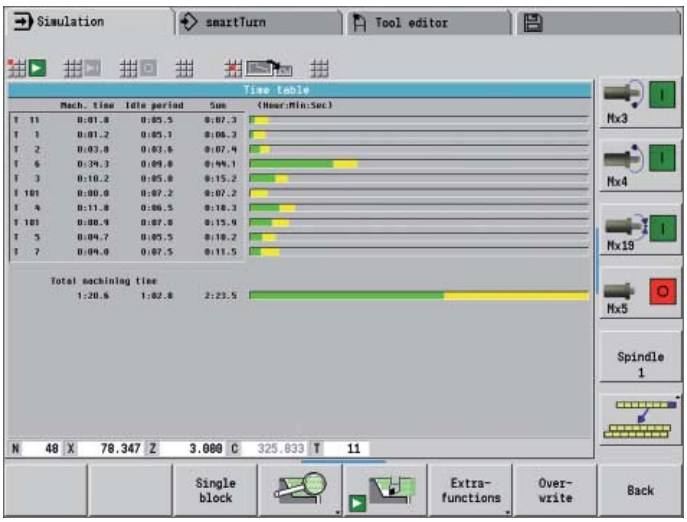
Use the "magnifying glass" to set the view to your current needs: greatly magnified in order to control the machining details, or zoomed-out to show the entire workpiece or current working space in order to get an overview of the machining process.

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 is a valuable aid with its machining time calculator. During simulation of the smart.Turn or DIN PLUS program, the CNC PILOT 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.





Turning, Drilling and Milling in One Setup
– Machining with the C Axis and Y Axis (Option)

You can use the CNC PILOT 620 to drill and mill your workpiece on the end face and lateral surface in one setup. This can be done by expanding the control to include C-axis or Y-axis machining and driven tools.

C axis or positionable spindle*
A driven tool as well as a C axis or positionable spindle are necessary for milling or drilling operations that must be carried out off-center on the end face or lateral surface.

Y axis*
With the Y-axis option of the CNC PILOT 620 you can machine slots or pockets with plane bottoms and perpendicular slot edges. By defining the spindle angle, you can determine the position of the milling contours on the workpiece. For programming and verification of these operations, the workpiece is shown in side and face view. The Y axis is supported in the smart.Turn and DIN programming feature.

* The machine and CNC PILOT 620 must be adapted to this function by the machine tool builder.

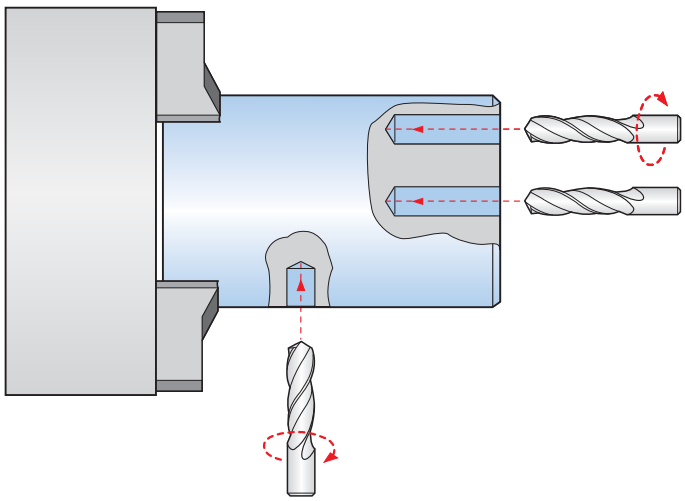
Drilling, deep-hole drilling, tapping
The CNC PILOT drills, pecks and taps individual holes with the C or Y axis. Via parameters you can easily program infeed reductions for the beginning of drilling or when drilling completely through the workpiece.

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 end 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 the thread-milling tools. The CNC PILOT supports thread milling.

Milling slots and simple figures
Slot milling with the CNC PILOT 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 or tapping

Drilling

Deep-hole drilling

Tapping

Thread milling

smart.Turn units and DIN PLUS cycles for drilling

Contour and pocket milling
The milling cycles of the CNC PILOT support contour and pocket milling. You determine all the important details, such as machining direction, milling direction, approach and departure behavior, feed rates, etc. The CNC PILOT automatically compensates for the tool radius.

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

In smart.Turn and DIN programming, the CNC PILOT 620 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 determine all important parameters, such as pitch, milling in multiple infeeds, etc.

Engraving cycles
Do you want to "inscribe" your workpieces? That's no problem with the CNC PILOT 620. 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 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)

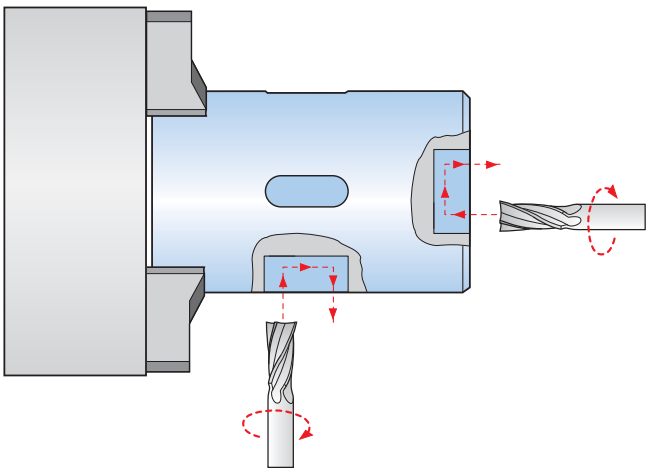


ICP contour milling


Face milling (single surfaces, flattening, polygon)


Helical slot milling

smart.Turn units and DIN PLUS cycles for milling



Face and lateral-surface milling

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 simplify your work.

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

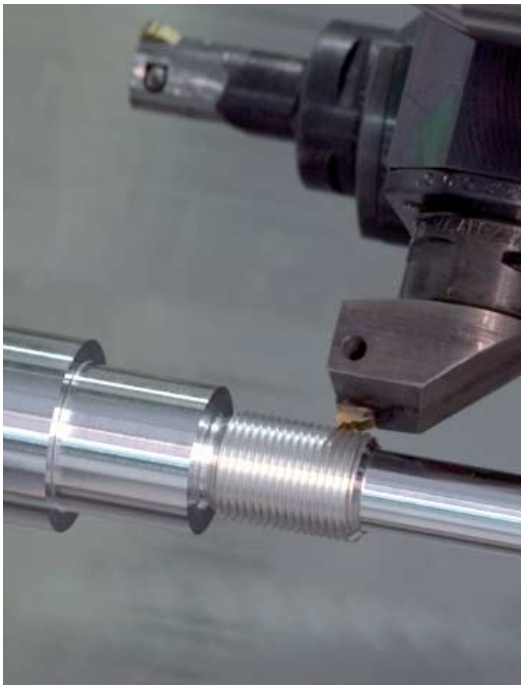
Thread-recut cycles
Even if the workpiece itself was clamped, you can very easily recut a thread with the CNC PILOT.

Simply clamp the workpiece and position the thread threading tool in the middle of a thread. The CNC PILOT 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 has all the information it needs to recut the thread.

Fewer calculations
In Teach-In mode, too, the CNC PILOT 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 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 saves the data. The next time you use the tool, you simply call the tool number. The CNC PILOT automatically adjusts for the correct tool size. You can immediately work to dimension.

When you turn a contour, the CNC PILOT automatically compensates the tool edge cutting radius to increase the dimensional accuracy of your workpiece.



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

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

Undercut form H, form K or form U

Parting

Threads, undercuts, parting

Technology data as default values
The CNC PILOT 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.

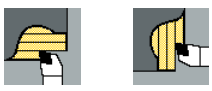
Reference points
You can define the **workpiece datum** by touching the workpiece with the tool or by entering the datum coordinates.

Approach the **tool-change point** once and store this position. After that, a simple cycle call suffices to traverse to the tool-change point again.

Protective zone for the spindle
For every tool movement in the negative Z direction, the CNC PILOT checks whether the programmed protective zone would be violated. If so, it stops the movement and responds with an error message.


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


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
– CNC PILOT Tool Database and Technology Database

Tool database
The CNC PILOT can store 250 tools in the standard tool database. The tool database can be expanded to 999 tools (option). The CNC PILOT differentiates between various types of turning, drilling and milling tools. The required data input varies depending on the tool type. This ensures that all important parameters are included even though data input has been reduced. You enter the tool parameters, such as the cutting radius, tool angle, point angle, cutting material, and the tool description in conversational mode. The input parameters are illustrated in context-sensitive help graphics.

Tool list
The CNC PILOT presents all tool data in the tool list in a clearly structured manner. Different sorting criteria help to find the desired tool quickly.

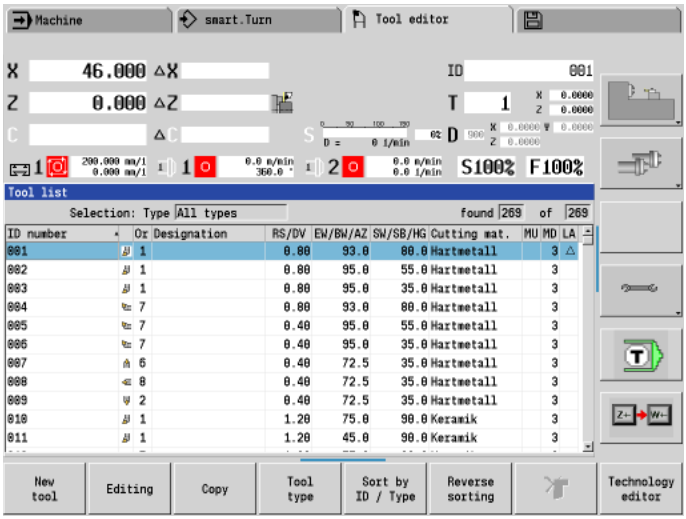
This list not only gives you an instant overview of your tools, it also is the basis for transferring tool data when creating NC programs or during manual machining tasks.

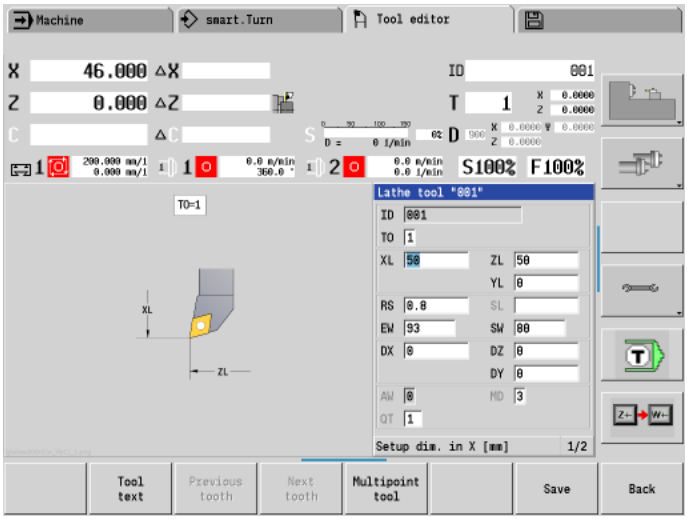
Wear compensation
The CNC PILOT offers a simple and straightforward function for compensating tool wear in both the X and the Z axes. You can enter the compensation values at any time, even during machining or after machining the workpiece.

Tool measurement
The CNC PILOT 620 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.
- By using a tool touch probe* (option). The tool moves in the direction of measurement. When the tool touch probe signal is triggered (e.g. from the TT 140 touch trigger probe with a cuboid probing element), the setting dimension is determined and saved.

Using an optical gauge or a tool touch probe for the measurement of tools gives you the tool data quickly, reliably and accurately.

* The machine and CNC PILOT 620 must be adapted to this function by the machine tool builder.





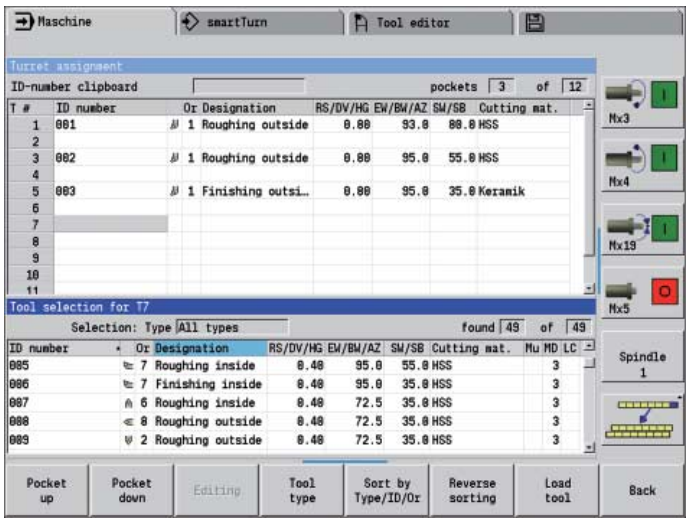
Turret assignment
You can view your machine's programmed turret assignment at any time. The CNC PILOT displays all important tool parameters.

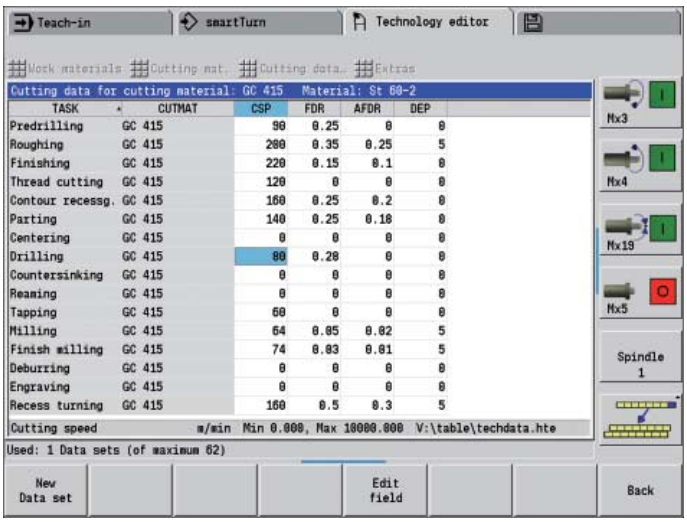
If you want to change the tool assignment or the tools in the turret, you can additionally display the entries of the tool database in the lower window. Now you need only select the desired turret pocket and choose the correct tool from the database. You can transfer the tool data to the turret assignment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

Technology data (option)
With the CNC PILOT 620 you need enter the cutting data only once. The control saves the cutting data according to the criteria of workpiece material, cutting material, machining mode. Thanks to this three-dimensional table, the control always knows the correct feed rate and the correct cutting speed.

The CNC PILOT 620 determines the machining mode from the Teach-In cycle or the unit. The cutting material is defined in 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 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. 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.





Open for Communication
– Fast Data Transfer with the CNC PILOT 620

The networked CNC PILOT 620
The CNC PILOT 620 can be integrated into networks and connected with PCs, programming stations and other data storage devices. Even in its standard version, the CNC PILOT features a latest-generation Fast Ethernet interface. The CNC PILOT communicates with NFS servers and Windows networks in TCP/IP protocol without needing additional software. The fast data transfer at rates of up to 100 Mbps guarantees very short transfer times.

USB interface
The CNC PILOT 620 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 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.

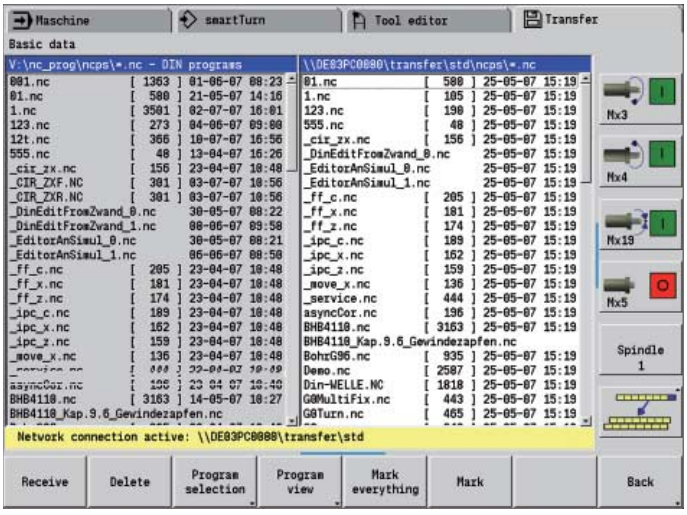
Transferring programs
An especially easy and convenient method of transferring data is to integrate the systems into your company network.

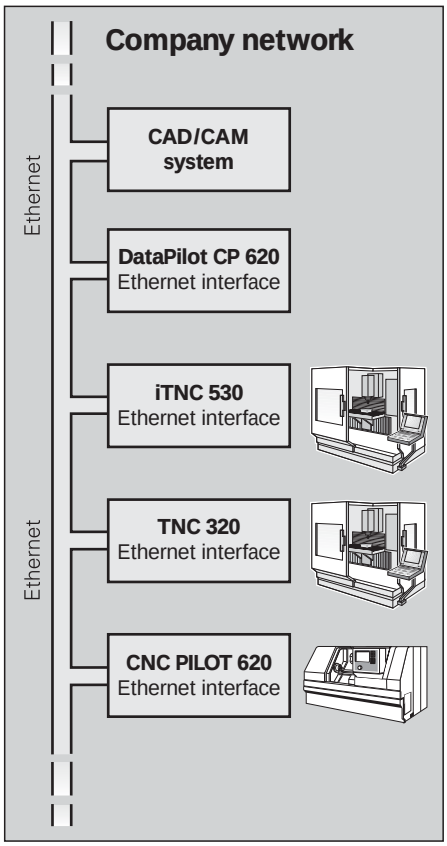
When transferring NC programs, the CNC PILOT even considers the files related to the cycle program, smart.Turn program or DIN PLUS program, such as contour descriptions, DIN macros or subprograms.

Exchanging tool data
Once you have acquired tool data you may also transfer them. This is not only important for data backup, but is also useful when using the DataPilot programming station for PCs. Advantages include consistently current data and not having to acquire the same data twice.

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

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





– The DataPilot CP 620 Programming Station

CP 620 DataPilot is the PC programming station for the CNC PILOT 620 and the organizing system for the workshop and design office.

That is why CP 620 DataPilot is the ideal supplement to the CNC PILOT 620 for program creation, archiving, and apprentice and advanced training.

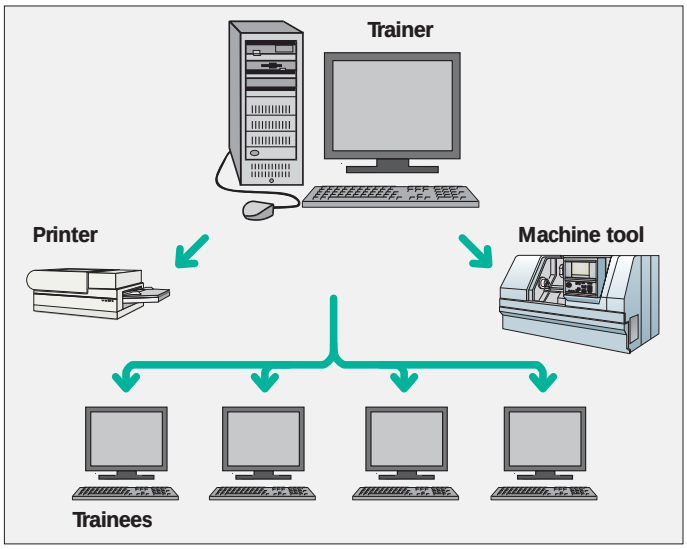
Creating programs
Programming, testing and optimizing the smart.Turn or DIN PLUS programs with DataPilot on your PC substantially reduce idle machine times. You do not need to adjust your way of thinking, since you program and test with DataPilot in exactly the same way as on the lathe. DataPilot has 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 has a large memory capacity, you should also back up your programs on an external system. The CNC PILOT features a USB and an Ethernet interface. This enables you to integrate the CNC PILOT 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 620
Because DataPilot CP 620 is based on the same software as the CNC PILOT 620, 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. Even setup functions such as workpiece datum definition, tool measurement or machining of individual cycles, cycle programs, smart.Turn or DIN PLUS programs are simulated with DataPilot. 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, or Windows 7 operating systems.



Overview
– User Functions

User functions	Standard	Option	
Configuration	•		Basic version: X and Z axis, spindle
		0-3	Positionable spindle and driven tool
		55+0-3	C axis and driven tool
		70+0-3	Y axis
		94+0-3	W axis (as closed-loop PLC axis)
	•		Digital current and speed control
Operating modes			
Manual operation	•		Manual slide movement through axis-direction keys 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		8	Sequential linking of fixed cycles, where each cycle is run immediately after input, or is graphically simulated and subsequently saved.
Program run			All are possible in single-block and full-sequence modes
	•		DIN PLUS programs
		9	smart.Turn programs
		8	Cycle programs
Setup functions	•		Workpiece datum setting
	•		Definition of tool-change position
	•		Definition of protection zone
	•		Tool measurement by touching the workpiece
		17	Tool measurement with a tool touch probe
		17	Tool measurement with an optical gauge
		17	Automatic workpiece measurement
Programming modes		9	The basis is the **unit**, which is the complete description of a machining block (geometry, technology and cycle data)
smart.Turn programming		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 and Y axes for simple holes, milling contours and drilling and milling patterns or those programmed with ICP
		9+55	Special units for activating/deactivating the C axis, subprograms and section repeats
		9	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
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

User Functions	Standard	Option	
ICP (continued)		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+42	DXF import: Import of contours for lathe and milling operations
DIN PLUS Programming	•		Programming in DIN 66025 format
	•		Extended command format (IF.. THEN ... ELSE...)
	•		Simple geometry programming (calculation of missing data)
	•		Powerful machining cycles for turning, recessing, recess turning and thread machining
		55	Powerful machining cycles for boring, drilling and milling with the C axis
		70	Powerful machining cycles for boring, drilling and milling with the Y axis
	•		Subroutines
	•		Programming with variables
		8/9	Contour description with ICP
	•		Program verification graphic 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
Teach-in programs (cycle programs)		8	Turning cycles for simple and complex contours, as well as contours defined with ICP
		8	Contour-parallel turning cycles
		8	Recessing cycles for simple or complex contours, or contours defined with ICP
		8	Repetitions with recessing cycles
			Placing the Teach-In programs at the end of the programming area
		8	Recess turning cycles for simple and complex contours, as well as contours defined with ICP
		8	Undercut and parting cycles
		8	Threading cycles for single or multi-start longitudinal, taper or API threads
		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 with the C axis
		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
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 wire-frame or cutting-path graphics, special identification of the rapid-traverse paths
	•		Machining simulation (2-D material-removal graphic)
	•		Side or face view, or 2-D view of cylindrical surface for verification of C-axis machining
	•		Display of programmed contours
	•		View of face and YZ plane for verification of Y-axis machining
	•		Three-dimensional display of finished part
	•		Shifting and magnifying functions
Machining time analysis	•		Calculation of machining time and idle machine time
	•		Consideration of switching commands triggered by the CNC
	•		Representation of time per individual cycle or per tool change

Overview
– User Functions (Continued)
– Accessories

User functions	Standard	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
	•		Precision path correction via handwheel, transferring compensation values to 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 tool insert wear
	•		Management of multipoint tools (multiple inserts on one tool holder)
Technology database		8/9	Access to cutting data after definition of workpiece material, cutting material and machining mode. The CNC PILOT 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	•		Chinese (simplified), Chinese (traditional), Czech, Danish, Dutch, English, Finnish, French, German, Hungarian, Italian, Polish, Portuguese, Russian, Spanish, Swedish
		41	For more conversational languages, see *Options*

Accessories	
Electronic handwheels	One portable **HR 410** serial handwheel
Tool calibration	**TT 140** touch trigger probe with cuboid probe contact
Workpiece measurement	• **TS 220:** Touch trigger probe with cable connection or • **TS 440:** Touch trigger probe with infrared transmission or • **TS 444:** Touch trigger probe with infrared transmission or • **TS 640:** Touch trigger probe with infrared transmission or • **TS 740:** Touch trigger probe with infrared transmission
DataPilot CP 620	Control software for PCs for programming, archiving, and training for the CNC PILOT 620 • Full version with license for single station or multiple stations • Demo version (free of charge)
Software for PCs	• **TeleService:** Software for remote diagnosis, monitoring, and operation • **TNCremoNT:** Software for data transfer—free of charge • **TNCremoPlus:** Software for data transfer with live-screen function

Overview
– Specifications

Specifications	Standard	
Components	•	MC 6120 main computer with 15-inch color TFT flat panel display • CC 6106/CC 6108 controller unit • TE 615 QT operating panel with – Integrated machine operating panel – Rapid-traverse and feed-rate override – Electronic handwheel
Operating system	•	HEROS real-time operating system for machine control
Memory	•	250 MB (on CFR compact flash memory card) for NC programs
Input resolution and display step	•	X axis: 0.5 µm, diameter 1 µm • Z and Y axis: 1 µm • C axis: 0.001° • W axis: 1 µm
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 • C axis: Interpolation in the linear axes X and Z with the C axis
Feed rate	•	mm/min or mm/rev • Constant surface speed • Max. feed rate (60000/pole pairs × ball screw pitch) at f_{PWM} = 5000 Hz
Spindle		Maximum 40000 rpm (with 2 pole pairs)
Axis feedback control	•	Digital drive control for synchronous and asynchronous motors • Position loop resolution: Signal period of the position encoder/1024 • Position control clock pulse: 3 ms • Speed control clock pulse: 0.6 ms • Current control: 0.1 ms
Error compensation	•	Linear and nonlinear axis error, backlash, reversal spikes during circular movements • Stick-slip friction
Data interfaces	•	100BaseT Fast Ethernet interface • 3 x USB 2.0 (2 on back, 1 x on front)
Diagnosis	•	Fast and simple troubleshooting through integrated diagnostic aids
Surrounding temperature	•	Operation: +5 °C to 45 °C • Storage: –35 °C to +65 °C

Overview
– Options

Option number	Option	ID	Comment
0 **1** **2** **3**	Additional axis	354540-01 353904-01 353905-01 367867-01	**Additional control loops 1 to 4**
8	Teach-in	632226-01	**Teach-in programming** • Contour description with ICP • Cycle programming • Technology database with 9 workpiece-material/tool-material combinations
9	smart.Turn	632227-01	**smart.Turn** • Contour description with ICP • Programming with smart.Turn • Technology database with 9 workpiece-material/tool-material combinations
10	Tools and technology	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	632229-01	**Threads** • Thread recutting • Handwheel superimposition during thread cutting
17	Touch probe functions	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
41	Additional language	530184-01 530184-02 530184-03 530184-04 530184-06 530184-07 530184-08 530184-09 530184-10	Slovenian Slovak Latvian Norwegian Korean Estonian Turkish Romanian Lithuanian
42	DXF-Import	632231-01	**DXF import** • Import of DXF contours
46	Python OEM Process	579 650-01	**Python application on the CNC PILOT 620**
55	C-axis machining	633944-01	**C-axis machining**
70	Y-axis machining	661 881-01	**Y-axis machining**
94	W-axis machining	679 676-01	**Support of W axis**

HEIDENHAIN

DR. JOHANNES HEIDENHAIN GmbH
Dr.-Johannes-Heidenhain-Straße 5
83301 Traunreut, Germany
☏ +49 8669 31-0
FAX +49 8669 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 Technisches Büro Nord**
12681 Berlin, Deutschland
☏ 030 54705-240

HEIDENHAIN Technisches Büro Mitte
08468 Heinsdorfergrund, Deutschland
☏ 03765 69544

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

BA Bosnia and Herzegovina → **SL**

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 **Belarus**
GERTNER Service GmbH
50354 Huerth, Germany
www.gertner.biz

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**
02770 Espoo, 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: service@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**
Chennai – 600 031, India
www.heidenhain.in

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

JP **HEIDENHAIN K.K.**
Tokyo 194-0215, Japan
www.heidenhain.co.jp

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

ME Montenegro → **SL**

MK Macedonia → **BG**

MX **HEIDENHAIN CORPORATION MEXICO**
20235 Aguascalientes, Ags., Mexico
E-mail: info@heidenhain.com

MY **ISOSERVE Sdn. Bhd**
56100 Kuala Lumpur, Malaysia
E-mail: isoserve@po.jaring.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**
02-489 Warszawa, Poland
www.apserwis.com.pl

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

RO **HEIDENHAIN Reprezentanţă Romania**
Braşov, 500338, Romania
www.heidenhain.ro

RS Serbia → **BG**

RU **OOO HEIDENHAIN**
125315 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 **Posredništvo HEIDENHAIN NAVO d.o.o.**
2000 Maribor, Slovenia
www.heidenhain-hubl.si

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

TR **T&M Mühendislik San. ve Tic. LTD. ŞTİ.**
34728 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.gertner.biz

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!*

743 888-21 · 5 · 8/2010 · F&W · Printed in Germany