HEIDENHAIN

Functions of the CNC PILOT 640

Comparison with the CNC PILOT 4290

The CNC PILOT 640 is always the right choice, allowing you to produce individual parts and series as well as simple and complex components with high efficiency, flexibility, and reliability. The CNC PILOT 640 offers a wide range of functions that cover not only the typical jobs of a conventional lathe but also automated manufacturing tasks. Programming directly on the control is fast and easy, even while other machining processes are in progress.

The CNC PILOT 640 provides intelligent support, including:

3D simulation
Gain a reliable assessment of the results prior to actual machining. The accurate, high-resolution 3D simulation of the CNC PILOT 640 lets you see whether, where, and why errors are to be expected and allows you to correct them before a single chip has fallen.

Touchscreen
Intuitively operate the CNC PILOT 640 with gestures commonly used on mobile devices. Simply select our 15.6-inch or 19-inch monitor with multi-touch functionality. A full ASCII keyboard with a number pad is also available for entering data.

smart.Turn
Create NC programs without needing to know G-codes. All of the required parameters are in user-friendly fillable forms illustrated with context-sensitive help graphics.

TURN PLUS
At the touch of a button, you can generate a work plan, machining strategy, tool selection, cutting data, and NC blocks for your machining operation. You just need to specify the geometry, material, and chucking equipment. The CNC PILOT 640 automatically generates the NC program.

Connected Machining
The responsibility for component quality and delivery date lies where the parts are machined: on the shop floor. That is why all information on machining has to come together here. And that is why you also need to have access to all relevant data from the shop floor and to apply your experience to the process chain. This is where the features of Connected Machining support you.

Send and receive e-mails on the control. Use the control to work directly with Windows applications, such as CAD programs. Forward status information in paperless format from the control. This is how you network your manufacturing process from the design to the shipped component with the shop floor as a command center in the company network.

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

The timely recognition of errors is extremely important in NC programming. With its graphic simulation feature, the CNC PILOT 640 supports you in checking the program for errors—with the real and exact dimensions of the contour and cutting edge, because the simulation works with the geometry values from the tool database.

Graphic simulation enables you to check the following already before machining:
• Approach and departure behavior
• The machining sequence
• The proportioning of cuts
• The finished contour
In the graphic simulation you can also display the tool cutting edge. You can view the cutting-edge radius, width, and position rendered true to scale. This helps you recognize machining details or collision risks in advance.

3D simulation
The accurate, high-resolution 3D simulation is particularly well suited for monitoring the machining results from turning, drilling, or milling processes prior to actual machining, or to monitor the workpiece on the control's screen. The 3D simulation visualizes milling and turning processes using different colors. This also applies to complex multi-channel operations. All workpieces are displayed, as are the tool movements of all slides. The rendered workpiece blank and finished part can be rotated on all axes, thus permitting visual inspection from any angle. With its intuitive gesture control, 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 3D simulation enables you to detect even the smallest error before actual machining.

Of course, the CNC PILOT 640 is also available with the tried and tested line graphics, cutting-path graphics, and movement simulation (material-removal graphic) of the CNC PILOT 4290.

Your benefits:

Recognize critical tool movements
Identify unintended movement of the tool before machining
Minimize scrap
The detailed graphics let you recognize unwanted machining effects early on
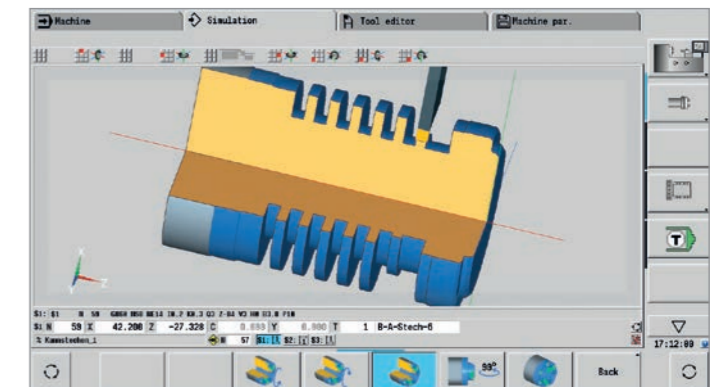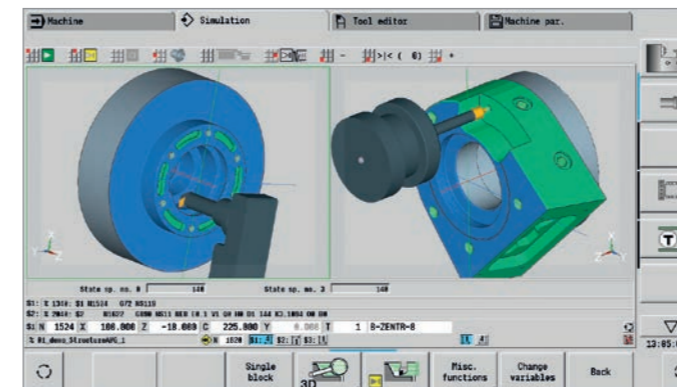**Find necessary changes quickly**
Recognize turning and milling operations at a glance
Complete simulation
Five-axis milling, turning, drilling, sinking, chamfering, and realistic thread depiction
Avoiding collisions
Realistic representation of the tool holder in the simulation

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

Options are functions that are integrated in the control and that allow you to tailor the feature range of the CNC PILOT 640 to your actual requirements, including at a later time. Some options need to be adapted by the machine manufacturer.

The features and specifications described here apply to the following control and NC software versions:
CNC PILOT 640 with NC software versions
688946–08
(export license required)
688947–08
(export license not required)

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

Main features of the CNC PILOT 640
State-of-the-art multi-touch operation and clear-cut display

Programming
Effective, clearly organized and flexible

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

The CNC PILOT 640 is available with two screen formats. The 19-inch TFT color flat screen is particularly suitable for large or multi-channel machines that require a larger field of view.

The smaller screen in the 15.6-inch widescreen format is particularly suitable for compact machines. The widescreen format ensures a clear overview of the programs—particularly in programming mode.

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

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

The lathe controls from HEIDENHAIN were impressing users with their innovative 9-block menu guidance long before the advent of touchscreen operation. Here the numeric keypad serves both for the selection of functions and for data input. This makes fast one-handed operation possible. However, the user interface design based on this technique is also ideal for touch operation. This enhances a simple and effective operating technique with today's touchscreen technology. The familiar input via the ergonomically designed control panel is fully continued in the CNC PILOT 640. At the same time, you benefit from the same intelligent gesture control you've learned to use on your mobile devices.

Your benefits:

Effective operation
Through swiping (kinetic scrolling) in NC programs, tables, and lists
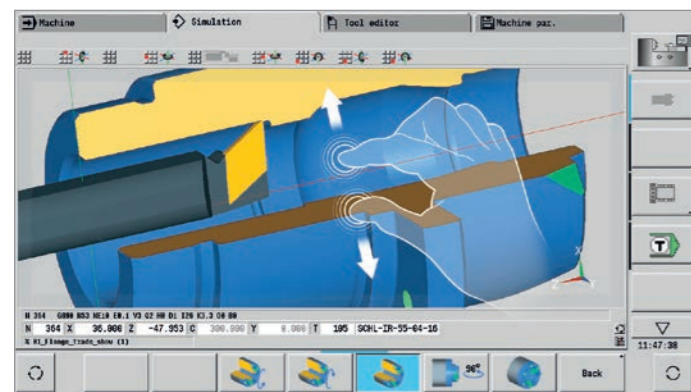**Intuitive operation**
Through gesture control, e.g., in the 3D simulation
Easy to operate
Direct selection of operating elements, soft-key rows, and navigation in menus
Practical touchscreen
Designed for harsh shopfloor conditions: splash-proof, scratch-resistant, and an IP54 rating

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

The smart.Turn principle
The working block—called a unit—plays a pivotal role in smart.Turn programs. A unit completely and unambiguously defines a single working step. The unit includes the tool call, the technology data, the cycle call, the approach and departure strategies, and global data such as the safety clearance. All of these parameters are displayed together in a clearly structured dialog box. The smart.Turn principle gives you the reassurance that the working block is defined correctly and completely. In the NC program, smart.Turn lists the unit's DIN PLUS commands. This gives you an overview of all of the working-block details at any time.

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 blank and finished part description, and the actual machining operation.

Guided by the dialog, you enter the following data in this sequence:
• Program head
• Tool assignment in the turret
• Description of workpiece blank
• Description of finished part
• Individual machining steps
The smart.Turn method not only ensures that the program is easy to read; it also makes it possible to save all of the information required for producing the workpiece in the NC program.

TURN PLUS automatic program generation
With its powerful **TURN PLUS** automatic program generation, the control takes you on the fast lane from the drawing to the finished workpiece. After you have graphically entered the workpiece blank and finished contour, or imported it from a DXF file, simply select the material and fixtures. TURN PLUS does everything else **automatically:**
• Analysis of the contours
• Planning of the work strategy
• Selection of tool data and cutting data
• Generation of NC blocks
As a result, you receive a complete and thoroughly commented NC program in smart.Turn format.

Your benefits:

Better readability of NC programs
Well organized and user-friendly depiction. Program structure can be displayed in a tree view
User-friendly programming
Dialog-guided fillable-form input with help graphics
**Collection of technology data**
Automatic collection and input of technology data into the respective work block
Downward compatibility with existing NC programs
Compatible with predecessor controls through import filters for NC programs
Effective program creation with TURN PLUS
Up to 90% time savings in programming through analysis of the contours, planning of the working strategy and selection of the tool or cutting data

Technology data
Material
Speed
Feed rate

TURN PLUS

Tools
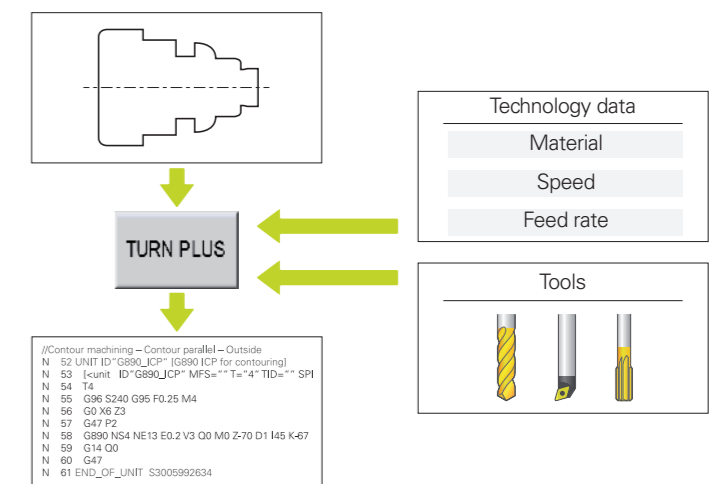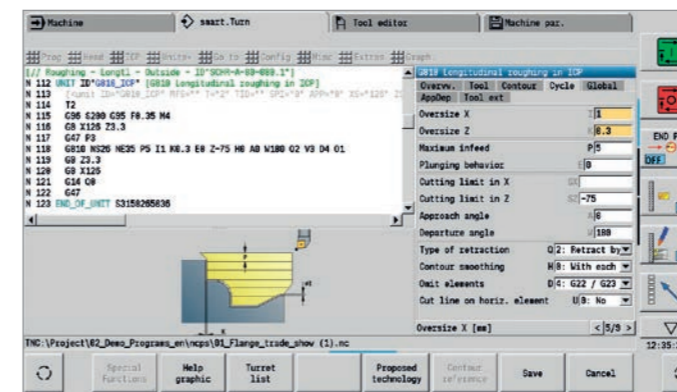
Setup
Tool measurement and workpiece measurement

Workpiece measurement
The CNC PILOT 640 features measuring cycles for checking the geometry of the machined workpieces.
For the measuring cycles, you simply insert a 3D touch probe from HEIDENHAIN into the turret in place of a tool. With a touch probe you can:
- Check whether all machining operations were conducted correctly
- Determine infeeds for finishing
- Detect and compensate for tool wear
- Inspect workpiece geometry and sort parts
- Log measured data
- Ascertain the machining trend

The HEIDENHAIN workpiece touch probe systems help to reduce costs on the shop floor and in serial production:
The setup, measurement, and inspection functions can be executed automatically with the probing cycles of the CNC PILOT 640.

Tool measurement
Exact measurement of the tool dimensions is a decisive factor for ensuring a consistently high level of production quality. Tools can be measured for this purpose with a TT tool touch probe from HEIDENHAIN before or after a machining step.

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

Your benefits:

Easy calibration
Fully automatic calibration of the touch probe
Management of multiple touch probe data sets
Transparent and central management of touch probe data
Probe feed manipulation with override knob
Adjust the speed of probing without affecting accuracy
Risk reduction
Avoid collisions in automatic mode and in manual operation

Program run
Manufacturing complex parts productively

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. On machines with a B-axis, machining in the tilted plane and even 5-axis simultaneous machining are possible.

In multi-channel machining, multiple slides simultaneously perform different machining steps. Several workpieces can also be machined at the same time, e.g., the front side on one part and the back on the other. These complex sequences require a perfect coordination of the individual processing steps.

A synchronization point analysis can be conducted beforehand in the simulation, which shows the time sequence of workpiece machining and the dependence of slides among each other. The CNC PILOT 640 calculates the productive, non-productive, and waiting times, as well as all tool changes and synchronization points. These are clearly displayed in a separate window.

Load Monitoring
The load monitor observes the machine's spindle and motor load while comparing them with the utilization values of a reference operation. You can set limit values for the measured load. If these values are exceeded, the CNC PILOT 640 can either stop the machining or continue with a replacement tool.

Advanced Dynamic Prediction (ADP)
ADP expands the previous pre-calculation of the maximum feed rate profile, providing optimized motion control for faultless surfaces and perfect contours, especially in milling operations. The control dynamically calculates the contour in advance and can therefore use an acceleration-limited motion control with smoothed jerk to adapt the axis velocity at contour transitions.

Simultaneous turning
Easy programming and time-saving machining of complex contours are possible directly on the machine and without a CAM system, with optimized production processes and fewer tool changes. Workpiece contours can be monitored relative to the tool and tool holder, while the tilt angle at the start and end points of the contour can be specified. The result: longer

tool life and prevention of visible transitions during the grinding of complex contours.

Trochoidal milling
The trochoidal milling function gives you prolonged tool life at higher feed rates and cutting depths, as well as lower machining times. This function can be used in all machining planes.

Component Monitoring
Through calculation of the mechanical loads on the spindle bearing during machining, you can avoid additional costs from unplanned machine downtime. You can thereby protect machine components from overloading without the need for additional sensors.

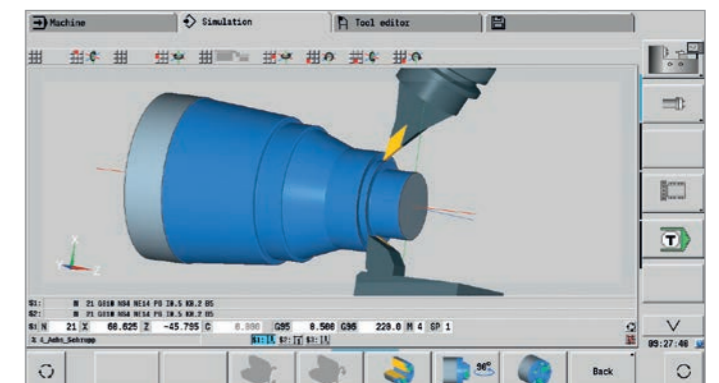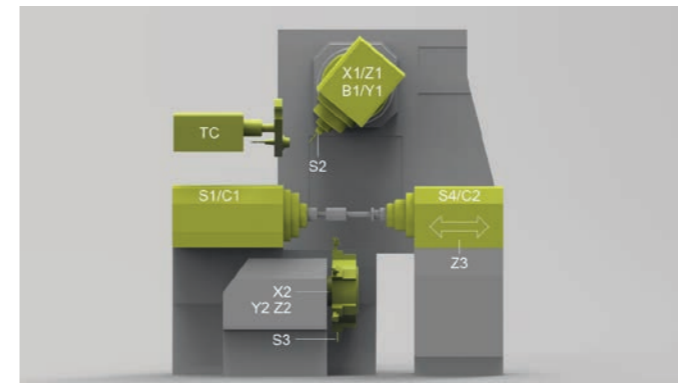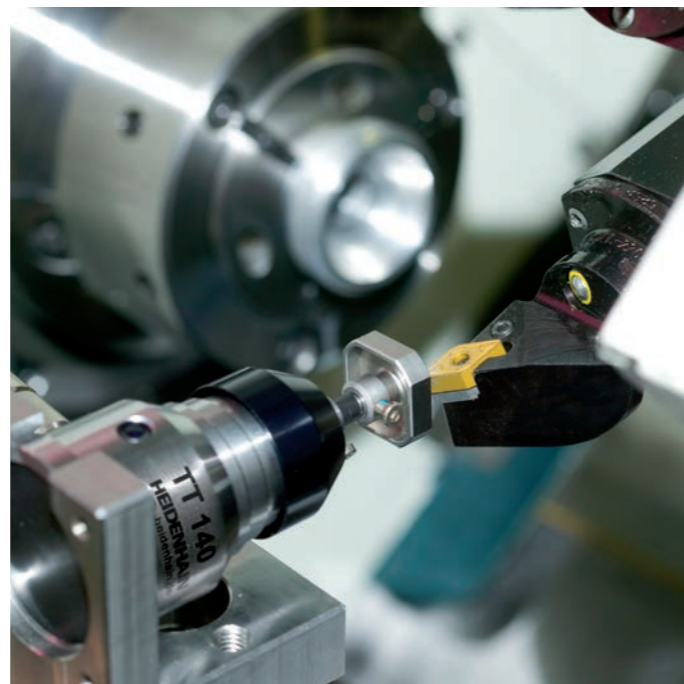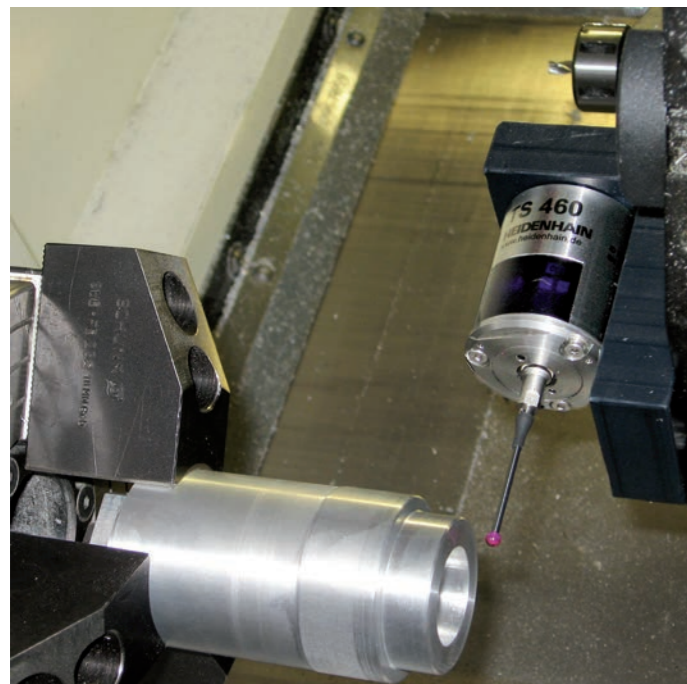**Your benefits:**

Shorter machining times
Through asynchronous multi-channel machining
Higher process reliability
Through spindle and axis supervision via Load Monitoring
Optimal surface definition
Through particularly smooth and precise tool movements thanks to ADP

Open for communication
Uniformly digital job management with Connected Machining

Comparison of functions of the CNC PILOT 640
New functions compared with the CNC PILOT 4290

connected + machining

With Connected Machining, HEIDENHAIN controls support a thoroughly digital job management in production. Connected Machining offers a set of solutions that permit the exchange of information with all areas of the company that accompany production.

The **HEIDENHAIN DNC** software interface connects a control with resource planning and control console systems and to software for machine and production data collection systems. The CNC PILOT 640 provides comprehensive data as the basis for decisions in enterprise management via the DNC interface.

With **Remote Desktop Manager**, the machine operator can easily and safely use all data and information available in the company by directly accessing the company's PCs and servers. Of course, he can also send feedback to all process participants. Direct networking with the order processing department is particularly advantageous. By doing so, the machine operator can save time by directly processing job-related information, such as completion notifications, on MES/ERP portals made available for this purpose.

The PC software **StateMonitor** provides an easy-to-read overview of machine data in production. The machine's efficiency can be ascertained through the evaluation of important data such as current machine status, override settings, and the usage history. Based on the collected data, it indicates the available optimization potential and, if necessary, e-mails a notification if a machine is down or a malfunction occurs.

Extended Workspace enables the machine operator to work in parallel on the machine and on job management. He simply connects a second screen with the control via Ethernet and configures it as an additional screen. The control can then open PDFs, 3D data, DXF data, DXF files, or Remote Desktop Manager on a second screen.

Your benefits:

Windows-based applications right on your control
Operate PC systems or manage job orders
Increased competitiveness through optimized information flow
Gaps in information cost valuable time and resources. Optimize your data flow on the CNC PILOT 640
Using PC functions without reduction of machine performance
No negative effect on the control from remote access
Higher production efficiency
Continuous data exchange and coordinated processes enable economical production
Numerous functions provided as standard features
Even without options, you can use the CNC PILOT 640 to improve many processes on the shop floor

	Functions	Brief description
Touch operation	Quick and easy operation by fingertip	Practical touchscreens for: • Intuitive gesture control • Effective operation through swiping (kinetic scrolling) • Easy navigation in menus
Programming (general)	Import of MANUALplus cycle programs	Import and automatic conversion of DIN and cycle programs
	Programmable secondary axes U, V, and W	Simultaneous movement of NC and secondary axes in the NC program
	Eccentric machining and non-circular turning (G725, G726, G727)	Eccentric turning of out-of-round contours without any additional special mechanical provisions
	Face milling (G797)	Milling various figures as islands
	Helical slot milling (G798)	Milling of slots with any desired thread pitch
	Bore milling (G75)	Milling holes of any desired diameter
	Contour thread (G38)	Manufacturing threads without form tools
	Simultaneous turning with the B axis (G891, G895)	Roughing and finishing of complex contours in a single step
	Multiple plunging (G860)	Expansion of recessing: automatic distribution of recesses, taking the recessing width into account
	Trochoidal milling (G847, G848)	Efficiently machine pockets with different shapes (circle, rectangle, and polygon) and any slots
	Programmable misalignment compensation (G976)	Conical machining or compensation of mechanical offset
	Programmable fluctuating spindle speed (G924)	Reduction of resonance during shaft machining
	LIFTOFF with NC stop	Safe retraction without workpiece damage, even during thread cutting
	Names for milling, drilling, and boring contours	Designation of contour parts with freely definable names to use them in cycles
	Definition of milling patterns for free contours	Creating linear or circular patterns with any contours
	Cycle extensions	Additional cycle functionality: • Aligning the proportioning of cuts to contour edges • Entering a dwell time in spindle revolutions
	Expanded subprogram interface	Transferring string parameters in subprograms and programming return values
	Expanded variable programming	String operations, e.g., for engraving dates and/or times
	Teach-in mode with blank form update	Machining workpieces step-by-step, thereby generating a program

	Functions	Brief description
Programming with smart.Turn	UNIT programming	Structured programming with function blocks
	OEM UNIT and OEM menu	Integrating complete UNITs and menu trees of the machine tool builder into the editing interface
	NC program with tree view	Clear depiction of long and complex programs
Programming with TURN PLUS	Generating programs with TURN PLUS	Generating a complete program with functional blocks (UNITs)
	Interactive tool selection	Convenient selection and adjustment of tools during automatic program creation
	Operating-parameter dialog	Edit frequently used TURN PLUS parameters in an easy-to-read dialog box
Simulation	3D simulation graphics	Display programmed contours as a 3D graphic before machining
	3D holder graphic	Depicting the tool holder in the 3D machining simulation
	3D simulation	3D simulation for turning, boring, drilling, and milling operations: • Accurate, high-resolution view • Section view can be moved • Realistic depiction of threads • Start block can be specified • Genuine C-axis movements • Display mode for engraving • Color scheme based on machining operation (turning / milling) • Simultaneous depiction of multiple contour groups
Program-run graphics	Real-time graphical simulation	Graphical depiction of the current machining operation
Setup	Touch probe cycles	Touch probe cycles in setup mode: • Touch probe cycles for datum and tool measurement • Calibrating the TT tool touch probe
	LAC weighing cycle	Function G239: load ascertainment for LAC (Load Adapt. Control) load-dependent adjustment of control parameters (option 143)
Program run	Advanced Dynamic Prediction (ADP)	Optimized motion control for perfect surfaces and contours, particularly for milling operations
	Thread recutting	Repair and rework of threads
	Batch mode	Automatic execution of several different main programs, e.g., bar loader operation, loading by robots
	Five-axis machining	Milling for free-form surfaces

	Functions	Brief description
Measuring	Probing cycles for tool measurement in manual operation	Calibrating tools with the TT tool touch probe
	Touch probe cycles in NC programs for workpiece measurement	Comprehensive touch probe cycles for: • Datum shift • Tool compensation • Additive correction • Logging
Tool management	Quick-change tool holder	Shortens setup and tool-change times
	Tool-holder database	Acquisition of geometric data on tools, e.g., for the 3D holder graphic
	Tool filter for tool selection or tool search	Comprehensive filter capabilities for fast searches in the tool table
	Tool-magazine management	Special functions for tool magazines: • Magazine pocket management for chain, manual, and circular tool magazines • Support of tool preselection • Tool change in mid-program startup • Loading and unloading during setup
	Tool-specific speed limitation	Assign a permissible maximum rotational speed for each tool
	Tool texts	Freely definable additional texts for each tool
Monitoring functions	Load Monitoring	Detect tool wear and breakage during machining
	Component Monitoring	Monitoring of machine components for overloading and wear
Connectivity	HEIDENHAIN-DNC	Interface for communication between NC and external PC applications
	Remote Desktop Manager	Display and remote operation of external computer units, e.g., a Windows PC
	HEIDENHAIN StateMonitor	Evaluation tool for all important data such as machine status, machine messages, etc.
	Support of handwheels with display	HR 550 FS and HR 520 (FS)
	State Reporting Interface (SRI)	Interface for convenient reporting of machine operating statuses for a higher-level MDA or PDA system

	Functions	Brief description
Improvements for machine manufacturers	Tool identification	Support of tool identification systems
	SQL tables	SQL tables defined by the machine manufacturer for expert programs or PLC functions
	Functional safety (FS)	Integrated functions in hardware and software that support the machine manufacturer in compliance with EN ISO 13849, "Safety of machinery — Safety-related parts of control systems"
	Gantry axes	Position and/or torque coupling of NC axes
	Integrated PLC software development environment	Convenient control-integrated development and diagnostic environment for the PLC user program
	PLC window	Display PLC window with Python
	Integrated TNCdiag motor diagnostics	Diagnostic possibilities of the entire drive train of the supply unit, inverter, and motors up to the connected encoders
	Modern computer hardware	Powerful processors on the NC and CC for compute-intensive functions

Improved functions compared with the CNC PILOT 4290		
Programming	ICP contour programming	Contour programming selectable in DIN or graphically (equally editable in both display modes)
	Recessing cycles with multiple planes	The desired plunging depth can be limited by an additional parameter
	Measuring spindle angular offset (G906)	Replaced by G905
	Spindle parting control (G991, G992)	Replaced by G917
Simulation	2-D simulation	Color marking highlights the areas currently being machined by the tool
	Tool carrier depiction	Any tool carriers, e.g., turrets or B-axis heads, can be displayed in the 2-D graphic
Improvements for the machine tool builder	PLC and commissioning	Various tools for diagnosis and commissioning are available, e.g., TNCopt, ConfigDesign, PLCdesign
	Machine display (dashboard)	Display elements for datum shift, display of the current kinematics, FS status, etc.
	Editing of the machining sequence	Expanded capabilities for definition of the machining sequence in TURN PLUS
	Definition of multi-edges	Simple definition of tools with multiple edges in the tool editor
Differences in function		
	Interpreter instead of compiler	Programs can now be started immediately. A previous compiler run is no longer necessary
	Realizing compensations	Tool and additive compensation values are realized in a movement immediately after a start. The CNC PILOT 4290 realized compensations gradually over the entire traverse distance
	Clamping fixture description	The clamping devices are described directly in the program. A clamping table is no longer required
	Tool life management	The sequence of exchange is defined in the NC program. The CNC PILOT 4290 defined the sequence in the turret table
	Variable programming with expanded functions	All variables are evaluated during run time. There is no need to distinguish between D and V variables. In addition, new functions are available for requesting states internal to the control

Overview

	CNC PILOT 640 NC SW 68894x-08	**CNC PILOT 4290** (in the latest software version)
Lathes and turning-milling machines	Max. 24 drives	Max. 10 drives
Program entry		
• Teach-In mode	Option	–
• smart.Turn	✓	–
• DIN PLUS	✓	✓
• Interactive Contour Programming (ICP)	✓	✓
• DXF Import	Option	Option
• TURN PLUS	Option	Option
Machining time analysis	✓	✓
Tool and workpiece measurement	Option	Option
C-, Y-, and B-Axis Machining	Options	Options
5-axis machining with X, Y, Z, B, C	Option	–
ADP for high surface quality	✓	–
Parallel axes (U, V, W)	✓	–
Counter Spindle	Option	Option
Multi-channel capability (up to three channels for asynchronous multi-slide machining)	Option	Option
Eccentric and non-circular turning	Option	Option
3D simulation	✓	–
Load Monitoring	Option	Option
Advanced synchronization of axes and spindles	Option	–
Magazine management	✓	Limited
Tools and Technology	Option	Option
User administration	✓	–
Connected Machining	✓	–
Component Monitoring	Option	–
State Reporting	Option	–

HEIDENHAIN
Mastering nanometer accuracy

**HEIDENHAIN
worldwide**