



# HEIDENHAIN



## TNC 640

Contouring Control for  
Machining Centers and  
Milling-Turning Machines



09/2021

# Contents



TNC 640 with 24-inch touchscreen



TNC 640 with 19-inch touchscreen



TNC 640 with 19-inch touchscreen (portrait)



The functions and specifications described in this brochure apply to the TNC 640 with NC SW 34059x-11

## The TNC 640...

Where can it be used?	<b>Versatile</b> The TNC contouring control for milling and milling-turning machines	<b>4</b>
What does it look like?	<b>Intuitive and user-friendly</b> State-of-the-art multitouch operation The functional user interface	<b>6</b>
What can it do?	<b>Complete part machining</b> Milling, turning, and grinding on one machine (option)	<b>10</b>
	<b>Fast and reliable machining at high contour fidelity</b> Dynamic Precision Optimal tool guidance by the TNC 640 Machining and measuring 3D contours	<b>12</b>
	<b>Machining with five axes</b> Guided tool tip Swivel head and rotary table controlled by the TNC 640	<b>20</b>
	<b>Intelligent machining</b> Dynamic Collision Monitoring option (DCM) Dynamic Efficiency Active Chatter Control option (ACC) Adaptive Feed Control option (AFC) Machine any contour slot with trochoidal milling Optimized roughing processes with OCM (option) Global Program Settings (option)	<b>28</b>
	<b>Automated machining</b> The TNC 640 measures, manages, and communicates Pallet management and serial machining Tool management and workspace monitoring	<b>31</b>
	<b>Minimize setup times</b> The TNC 640 makes setup easy	<b>31</b>
How is it programmed?	<b>Programming, editing, and testing</b> A full range of possibilities with the TNC 640 Graphical support in any scenario	<b>33</b>
	<b>Programming in the shop</b> Straightforward function keys for complex contours Programming free contours and Data Matrix codes Practical cycles for recurring operations Practical turning cycles (option) Practical functions for grinding and dressing (option) Reusing programmed contour elements Rapid availability of all information	<b>35</b>
	<b>Open to outside information</b> Processing CAD files with the TNC 640 Uniformly digital job management with Connected Machining StateMonitor: collect and evaluate machine data The TNC 640 programming station	<b>44</b>
What accessories are there?	<b>Workpiece measurement</b> Setup, preset setting, and measuring with touch trigger probes	<b>50</b>
	<b>Tool measurement</b> Measuring length, radius, and wear inside the machine	<b>51</b>
	<b>Inspecting and optimizing machine accuracy</b> Easy calibration of rotary axes with KinematicsOpt (option)	<b>52</b>
	<b>Positioning with an electronic handwheel</b> Fine motion control of axes	<b>53</b>
... At a glance	<b>Overview</b> User functions, accessories, options, specifications, comparison of controls	<b>54</b>



## Versatile

### The TNC contouring control for milling and milling-turning machines

For more than 40 years, TNC controls from HEIDENHAIN have proven themselves every day on milling machines, machining centers, and drilling machines. Over this period, they have been continually further developed, with focus placed on an operating approach tailored to the needs of the machine operator.

#### Multitouch operation

The TNC 640 comes with a keyboard and touchscreen that puts zooming, rotating, and dragging all at your fingertips for fast and easy operation of the TNC 640.

#### Shop-oriented programming

The TNC 640 lets you program conventional milling, drilling, turning, and grinding operations right on the machine using the shop-oriented Klartext conversational programming language from HEIDENHAIN. With helpful dialog guidance and informative visual aids, the TNC 640 provides optimal programming support, including for turning operations.

For standard machining operations and even complex applications, a wealth of practical machining and coordinate transformation cycles are at your beck and call.

#### Offline programming

The TNC 640 is easy to program remotely as well. Its Ethernet interface ensures extremely short upload times, even for long programs.

#### Easy to operate

When it comes to simple machining tasks, such as face milling or face turning, you don't need to write a program on the TNC 640. That's because the TNC 640 makes manual control equally easy—either by means of the axis keys or, for maximum sensitivity, with an electronic handwheel.

#### Universally deployable

The TNC 640 is particularly well suited to milling, turning, grinding, HSC, and 5-axis machining on machines with up to 24 axes. The following applications play to the strengths of the TNC 640:

#### Milling-turning machines

- Easy, program-controlled switching capability between milling and turning
- Extensive package of turning cycles
- Constant cutting speed
- Cutter radius compensation

#### Universal milling machines

- Shop-floor programming in Klartext, the conversational programming language from HEIDENHAIN
- Rapid preset setting with HEIDENHAIN touch probes
- Electronic handwheel capability

#### High-speed cutting

- Fast block processing
- Short control-loop cycle time
- Motion control with jerk compensation
- High spindle speed
- Fast data transfer

#### Boring mills

- Drilling and boring cycles
- Drilling of oblique holes
- Control of quills (parallel axes)

#### Five-axis machining with swivel head and rotary table

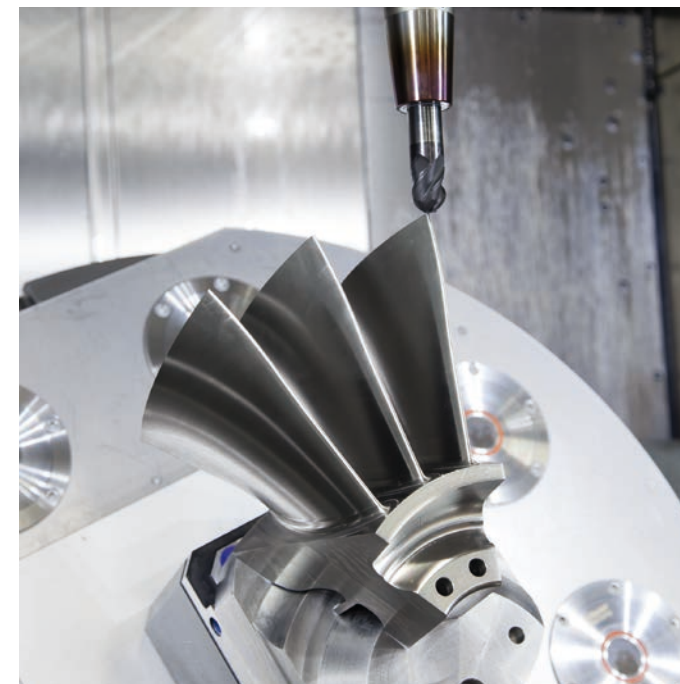
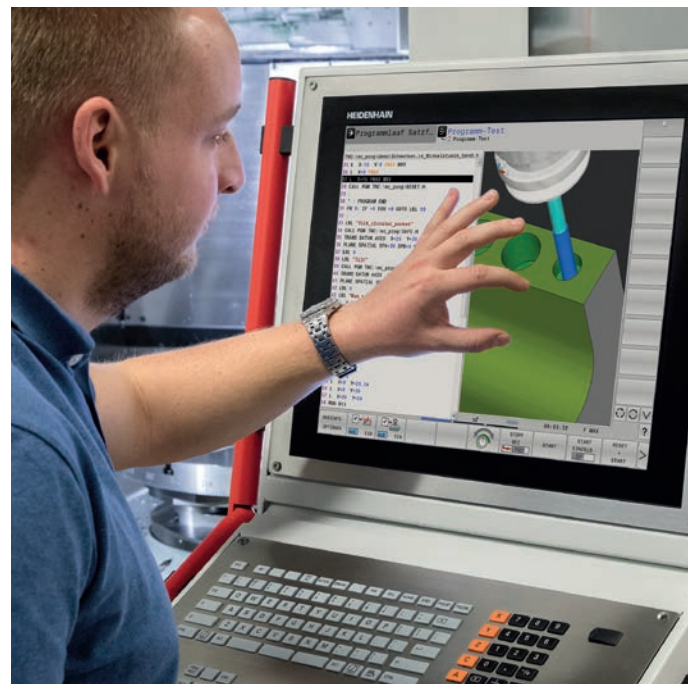
- Tilting the working plane
- Cylinder surface machining
- Tool Center Point Management (TCPM)
- 3D tool compensation
- Fast program execution thanks to short block-processing times

#### Machining centers and automated machining

- Tool management
- Pallet management
- Controlled preset setting
- Preset management
- Automatic workpiece measurement with HEIDENHAIN touch probes
- Automatic tool measurement and breakage inspection
- Connection with host computer

#### Grinding operations

- Convenient functions for jig grinding and dressing
- Reciprocating stroke superimposed onto the tool axis
- User-friendly cycles



# Intuitive and user-friendly

## State-of-the-art multitouch operation

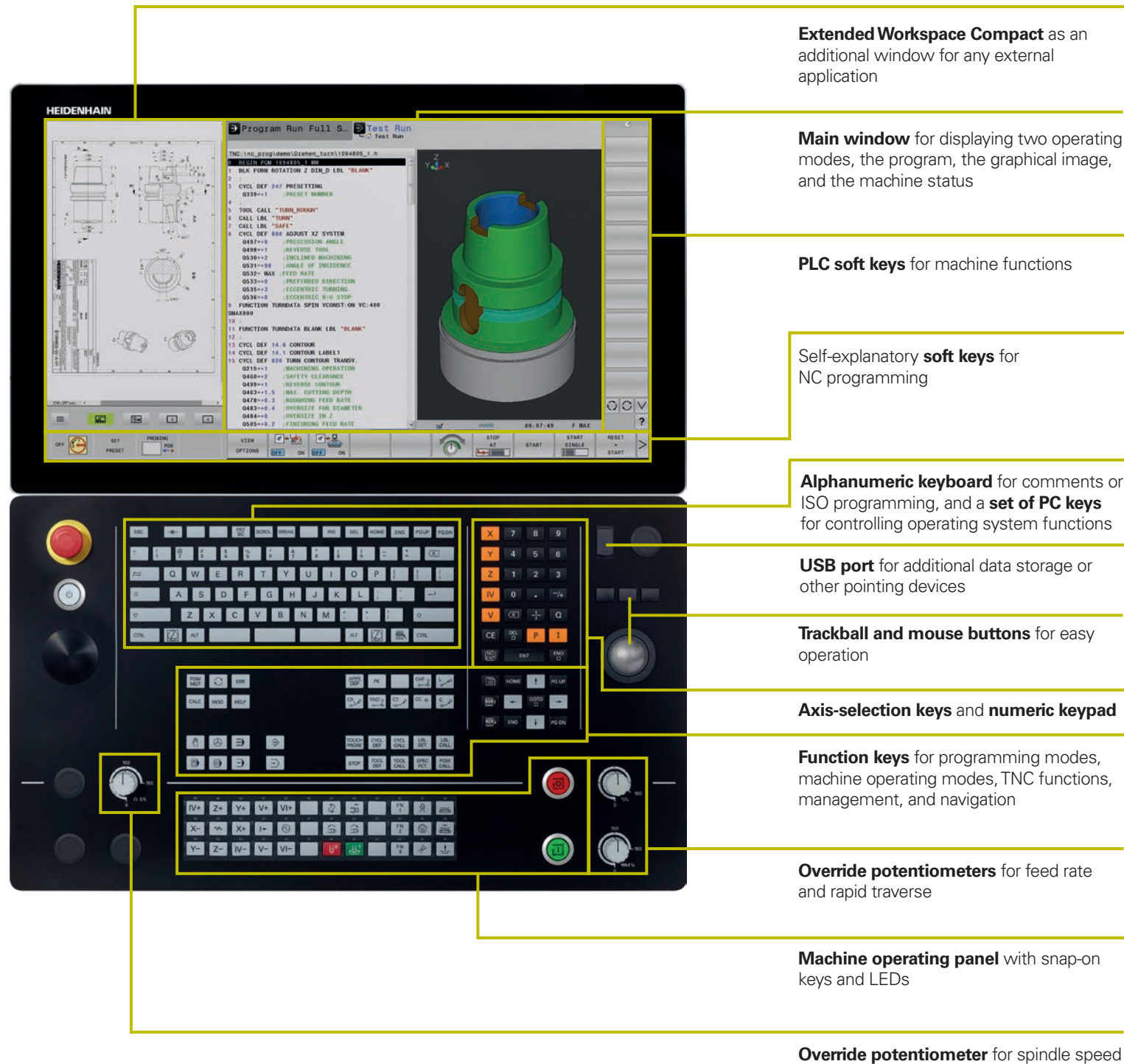
### The screen

In a clear format, the 24-inch widescreen displays all of the information you need for programming, operating, and checking the control and the machine, including program blocks, tips, and error messages.

**Extended Workspace Compact** offers even more information: alongside the main window with its usual control interface is an additional window for displaying any external application. You can also use a **split screen** in the main window to display your NC blocks on one side and the graphical image and status display on the other. While the program is running, **status displays** are available that inform you about the tool position, current program, coordinate transformations, and more. The TNC 640 even displays the current machining time.

### The operating panel

Like all of the TNCs from HEIDENHAIN, the operating panel was designed with the programming process in mind. Program entry is aided by the thoughtful arrangement of keys into **function groups** for programming operating modes, machine operating modes, management/TNC functions, and navigation. Intuitive key assignments, readily understood symbols, and abbreviations unmistakably identify the various functions. The group of keys containing NC Start/Stop and the feed rate/rapid traverse potentiometers let you work through the essential operating sequences in the Program Run, Single Block operating mode with just one hand. This helps you efficiently set up the NC program without losing sight of the machine's workspace. Comments and ISO programs can also be easily entered using the **alphanumeric keypad**. The potentiometers for the axis feed rate and spindle speed, which are clearly separated from each other, allow you to perform fine adjustments. In addition, the operating panel features a complete **set of PC keys** and a **trackball**.



**Extended Workspace Compact** as an additional window for any external application

**Main window** for displaying two operating modes, the program, the graphical image, and the machine status

**PLC soft keys** for machine functions

Self-explanatory **soft keys** for NC programming

**Alphanumeric keyboard** for comments or ISO programming, and a **set of PC keys** for controlling operating system functions

**USB port** for additional data storage or other pointing devices

**Trackball and mouse buttons** for easy operation

**Axis-selection keys and numeric keypad**

**Function keys** for programming modes, machine operating modes, TNC functions, management, and navigation

**Override potentiometers** for feed rate and rapid traverse

**Machine operating panel** with snap-on keys and LEDs

**Override potentiometer** for spindle speed

### Ergonomic and sturdy design

The anodized finish on the keyboard unit is highly resilient to chemical and mechanical factors. The engraved keyboard lettering, such as on the potentiometer scale, exhibits high scratch and abrasion resistance.

### Intuitive touchscreen operation

The operator-friendliness of the TNC 640 has proven itself over many years. Users throughout the world use dialog keys, navigation keys, and soft keys to operate their TNC 640. For even more innovative and user-friendly operation, the TNC 640 now also comes in a touchscreen version. This version combines the proven benefits of HEIDENHAIN controls with state-of-the-art operation through tapping, swiping, and dragging gestures.

### Practical touchscreen

With an IP54 rating, the touchscreen is designed to handle harsh shop conditions thanks to these characteristics:

- Dust-protected
- Splash-proof
- Scratch-proof

To clean your screen, simply select the "Touchscreen Cleaning" mode. This locks the screen to prevent unintended operation.

### Gestures for multi-touch operation

The screen of the TNC 640 can be operated with gestures already familiar to you from your smartphone or tablet. For example, you can zoom in or out with two fingers, or swipe to rapidly navigate through programs, menus, and the soft-key row.

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



# Intuitive and user-friendly

## The functional user interface

The key to safe and low-fatigue work is having an intuitive screen layout and an ergonomic, well-arranged keyboard. Although HEIDENHAIN has been adhering to these principles for a long time, the TNC 640 offers a number of features that make working with the control even simpler and easier than before.

### Leading-edge design

The user interface of the TNC 640 boasts a sleek outer appearance with gently rounded edges, color gradients, and a pleasingly uniform font. The different screen areas are clearly delineated, and the operating modes are indicated by special operating mode symbols.

To help you evaluate the priority of error messages, the TNC 640 displays them in color-coded categories. A color-coded warning triangle is also displayed.

### The smartSelect functions overview

With dialog support, you can now readily select functions that were previously available only via soft key structures. As soon as you open smartSelect, you are shown a tree structure with all of the subordinate functions that can be defined in the control's current operating state. The TNC also displays integrated help on the right-hand side of the smartSelect window. When you make a selection with the cursor or per mouse click, you immediately receive detailed information about the function. The smartSelect feature is also available for the definition of machining cycles, touch probe cycles, special functions (SPEC FCT), and for parameter programming.

### Color-structured programs

The content of a program line can be quite extensive:

- Line number
- Program function
- Input value
- Comment

To make sure you always keep an overview even in complex programs, the TNC 640 displays the individual program elements in different colors. This allows you, for example, to see at a glance where the editable input values are.

### Universal table editor

Regardless of whether you are editing a tool, preset, or pallet table, the layout, method of operation, and functions of the table editor are always the same.

### Info line

In the info line, the TNC 640 helps you orient yourself by displaying the respective status of the submode.

### MOD function

The additional MOD operating mode offers a wide variety of possible settings in a standardized layout, regardless of the operating mode.

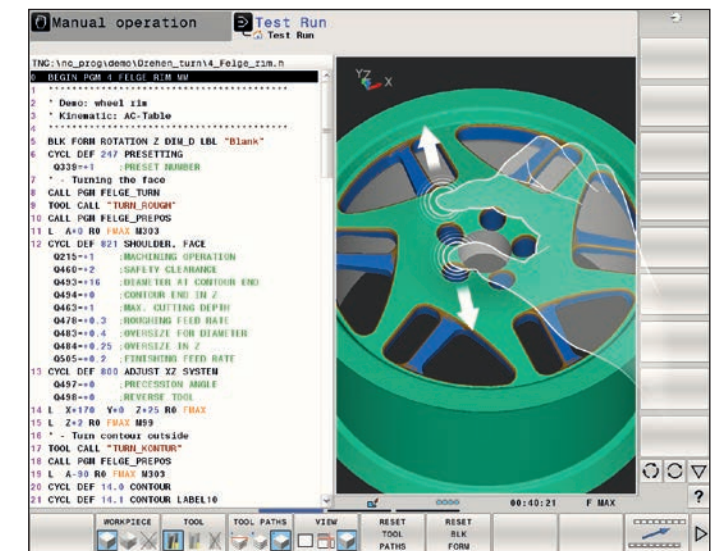
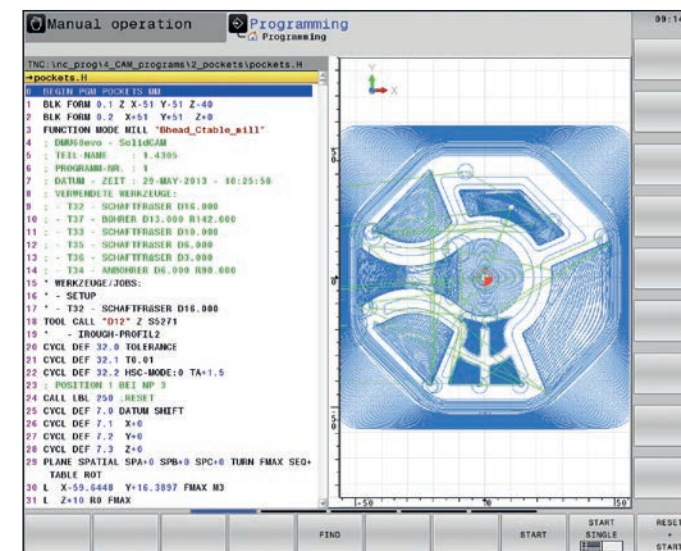
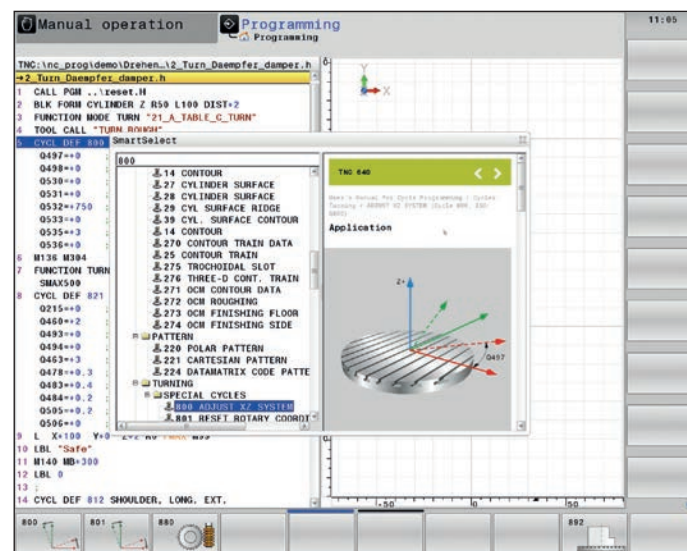
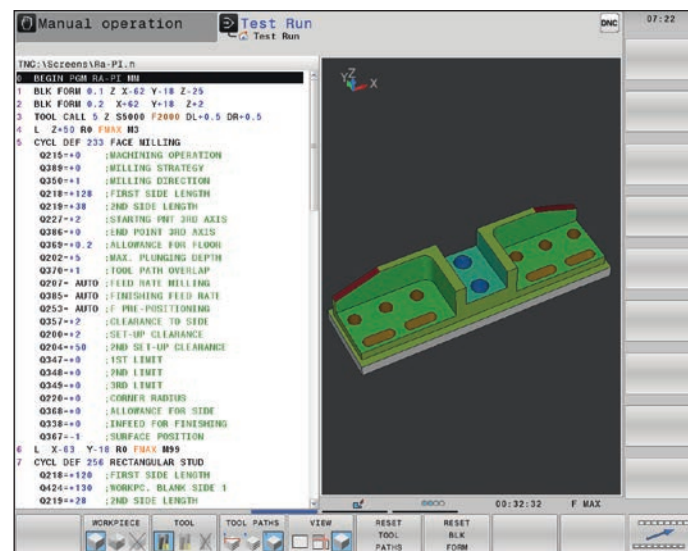
### Easy to operate

The TNC 640 can be conveniently operated with a connected mouse, but many machine shops lack an adequate surface for a mouse. A touchscreen, however, eliminates the need for additional workspace for input devices and makes everyday operation of the TNC 640 even easier thanks to swiping, direct selection

of operating elements, and menu navigation. The touchscreen is particularly helpful when using the CAD Import function, allowing you to rapidly zoom, drag, or select with simple gestures.

### User administration

The user administration feature of the TNC 640 lets you define various roles and access rights for users, ensuring that each user can act only within his assigned rights. This helps prevent the unintentional or unauthorized deletion of files and system file content. Many functions are also available only with the corresponding permissions. Consequently, the user administration feature of the TNC 640 not only provides greater data security but also increases machine operating safety.



# Complete part machining

## Milling, turning, and grinding on one machine (option)

Does your workpiece need to be moved to a lathe or grinding machine for additional machining steps after milling? Is all the additional capacity planning, fixture production, workpiece setup, and finished-part inspection a burden? If so, then a TNC 640 on a milling-turning machine can save you a lot of time. With the TNC 640, you can machine the complete part in one setup using milling, turning, and grinding operations in any sequence desired. And when you're finished, you can measure the finished part inside the machine using a HEIDENHAIN touch probe.

The TNC 640 provides powerful functions for NC-program-driven switching between turning, dressing, and milling mode. This gives you complete freedom to decide how and when you want to combine the various machining methods. And naturally, the ability to switch between machining modes is independent of the machine and its axis configuration. When switching between modes, the TNC 640 automatically handles all of the internal changes, such as switching to diameter display, setting the preset in the center of the rotary table, and machine-dependent functions such as clamping the tool spindle.\*

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

### Programming as usual

Turning operations can be programmed as always using the convenient HEIDENHAIN Klartext conversational programming language. In addition to the standard path functions for defining the turning contour, you can also employ FK free contour programming, which is an easy way to create contour elements not suitably dimensioned for an NC program. In addition, turning-specific recessing and undercutting contour elements are available and can be defined with informative help illustrations. Even grinding operations can be programmed as usual with HEIDENHAIN Klartext dialog guidance. If the contour is available as a CAD file, then you can easily import it with the aid of the CAD Import function (option).

### Milling, turning, and grinding cycles

HEIDENHAIN controls have always been known for their comprehensive and technologically advanced package of cycles. The cycles of the TNC 640 include multi-step, frequently recurring operations. As you program, conversational guidance and informative help graphics illustrating the required parameter data provide support. In addition to the well-known TNC milling and drilling cycles, the TNC 640 also provides a plethora of turning cycles for roughing, finishing, recessing, thread cutting, recess turning, and more. These turning functions are based on software from the long-proven HEIDENHAIN turning

controls, allowing you to program even complex turning operations with ease on the machine.

The TNC 640's more complex turning cycles employ the same techniques as those used for milling. As a result, TNC programmers can build on their prior knowledge to rapidly enter the world of turning operations on a milling machine without retraining. The control also features grinding cycles, including:

- Define reciprocating stroke
- Activate wheel edge
- Profile dressing

### Interpolation turning

Although you can create ring slots, recesses, tapers, or any turning contour using conventional turning operations, you can also take advantage of interpolation turning. In interpolation turning, the tool executes a circular movement using the linear axes. For outside turning, the cutting edge is oriented towards the circle center, and for internal turning, away from the center. By varying the circle radius and the axial position, any rotationally symmetric body can be produced, even in a tilted working plane. The TNC 640 provides two cycles for interpolation turning:

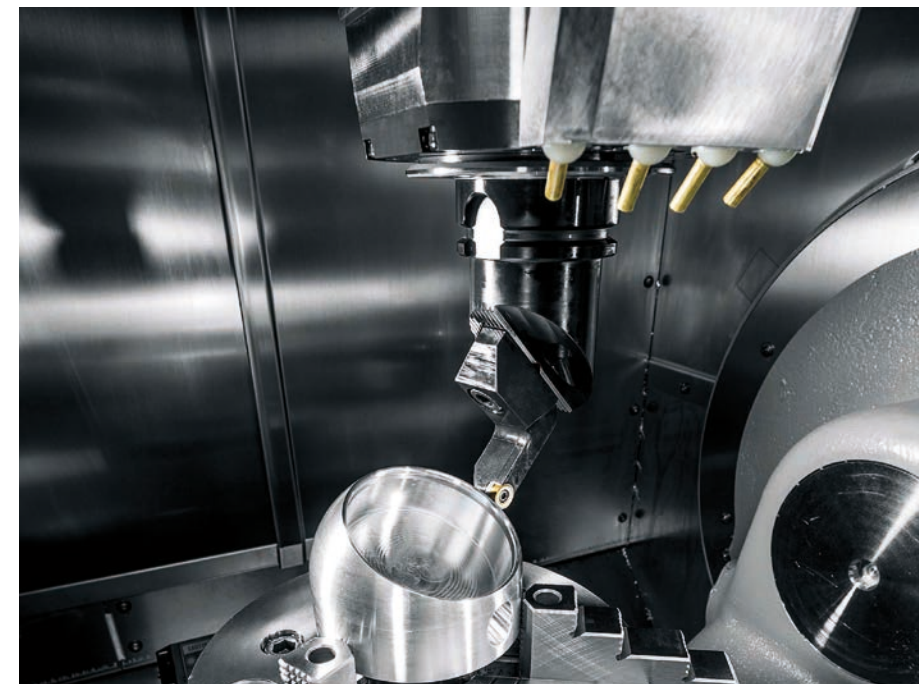
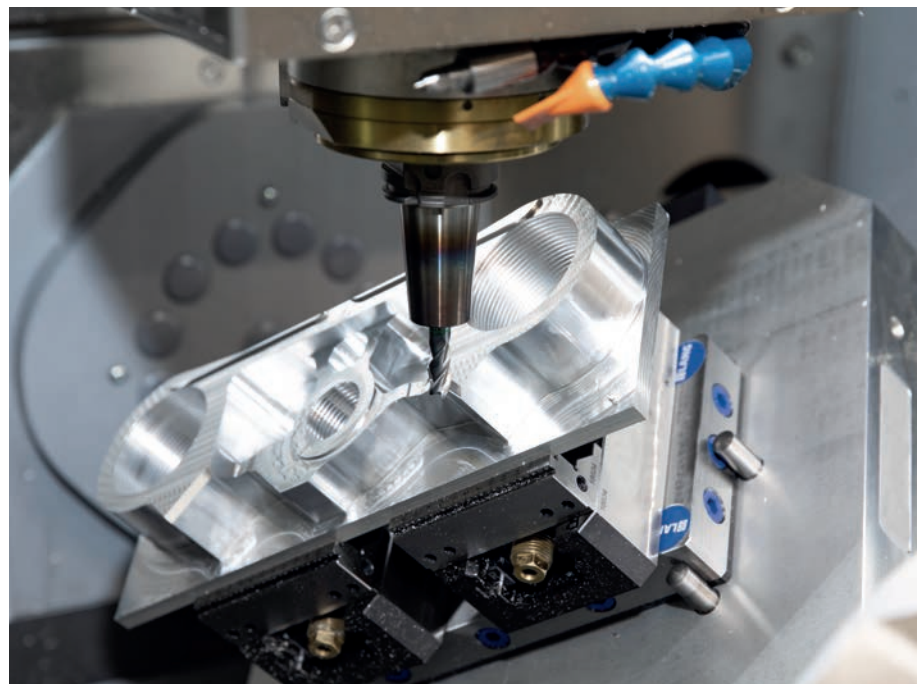
- Cycle 291 activates the coupling of the spindle positions with the axis positions, allowing you to program any desired axial or radial tool movement
- Cycle 292 activates spindle coupling and automatically controls the machining of a programmed turning contour

### Turning with a facing slide

A facing slide allows you to perform turning operations on a stationary workpiece, thereby enabling off-center or tilted-plane turning operations. With a facing slide, rotational movement is implemented by the spindle, where an axis integrated into the facing slide determines the extension of the turning tool (facing stroke). With the TNC 640, you don't need to worry about the complex motion involved. You simply select the facing-slide mode with a programming command and program as always with standard turning cycles. The TNC 640 handles all of the calculations and executes the motion sequences.

### Polar kinematics

With polar kinematics, the path contours in the working plane are performed by one linear axis and one rotary axis. This greatly increases the machining operations possible on machines which have only two linear axes. For example, front-face operations become possible on rotary and cylindrical grinding machines. And on milling machines, suitable rotary axes can replace various linear axes, for example when working on very large surfaces on large machine tools.





# Fast and reliable machining at high contour fidelity

## Dynamic Precision



The TNC 640 is designed for exceptional accuracy and surface quality at high machining speeds, regardless of whether you are milling or turning. A variety of technologies, cycles, and functions operating separately or in concert ensure perfect surface finishes in extremely short machining times:

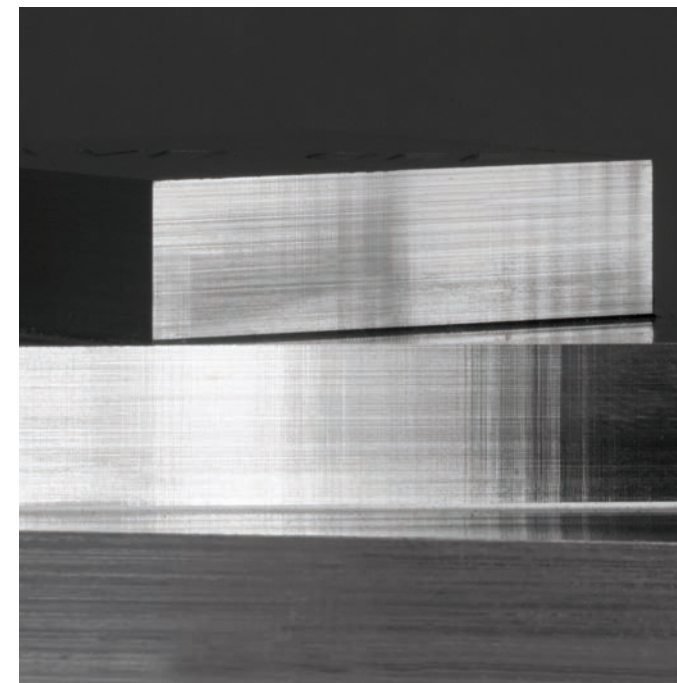
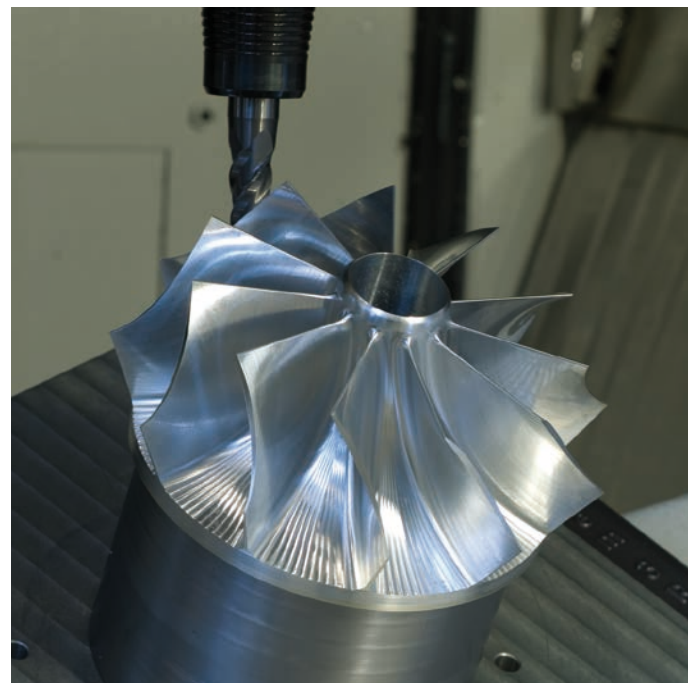
- Optimized motion control
- Effective jerk reduction
- Dynamic contour look-ahead

The umbrella term **Dynamic Precision** covers a number of HEIDENHAIN cutting solutions that can dramatically improve the dynamic accuracy of a machine tool. These solutions shed new light on the competing demands of accuracy, high surface quality, and short machining times. The dynamic accuracy of machine tools manifests itself in deviations at the tool center point (TCP). These deviations depend on kinetic quantities such as speed and acceleration (including jerk) arising from the vibration of machine components and other factors.

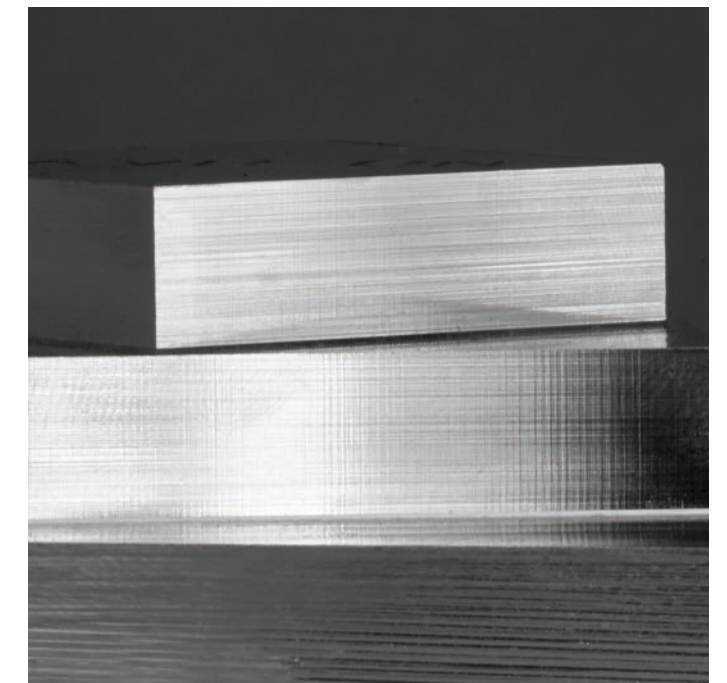
All of these factors contribute to dimensional inaccuracy and the formation of defects on the workpiece surface, thereby impacting both quality and productivity (in the case of quality-related scrap). Dynamic Precision counteracts these problems with intelligent control technology to help improve the quality and dynamic performance of a machine tool. As a result, production time and cost are reduced.

**Dynamic Precision** contains the following functions, which can be deployed separately or in combination.

Function	Benefits	
<b>CTC Cross Talk Compensation</b>	Compensation of acceleration-dependent position errors at the TCP	Greater accuracy during acceleration phases
<b>MVC Machine Vibration Control</b>	Damping of machine oscillations  <b>AVD</b> (Active Vibration Damping): Compensates undesired effects of vibrations on the feed axes  <b>FSC</b> (Frequency Shaping Control): Suppresses the inducement of vibrations through a correspondingly filtered feedforward control	A superior surface finish
<b>CTC + MVC</b>	–	Faster and more accurate machining
<b>PAC Position Adaptive Control</b>	Position-dependent adaptation of control parameters	Greater contour fidelity
<b>LAC Load Adaptive Control</b>	Load-dependent adaption of control parameters and the maximum axis acceleration	Higher accuracy whatever the load
<b>MAC Motion Adaptive Control</b>	Motion-dependent adaptation of control parameters	Less vibration and higher maximum acceleration during rapid traverse



Vibration can significantly impair surface quality.



Visibly superior surface quality can be attained with MVC.



## Fast and reliable machining at high contour fidelity

Optimal tool guidance by the TNC 640

### High contour fidelity and surface quality

TNC controls from HEIDENHAIN are known for their **low-jerk and speed/acceleration-optimized motion control**, enabling optimized surface quality and workpiece accuracy. With the TNC 640, you are able to leverage the latest technological developments. The TNC 640 anticipates and thinks along with you, dynamically calculating the contour ahead of time. In addition, special filters suppress machine-specific natural frequencies.

With its **Look Ahead** function, the TNC 640 recognizes directional changes ahead of time and adapts the traversing speed to the shape of the contour and the surface to be machined. You simply program the maximum machining speed as the feed rate and, in **Cycle 32 TOLERANCE**, enter the maximum permissible deviations from the ideal contour. The TNC 640 automatically adapts the machining to the tolerance that you define. This method prevents contour defects.

### Advanced Dynamic Prediction (ADP)

expands upon the previous look-ahead calculation of the permissible maximum feed rate profile. ADP compensates for differences in the feed rate profiles resulting from the point distribution on neighboring paths, especially in NC programs originating from CAM systems. Among other benefits, this results in particularly symmetrical feed rate behavior on the back-and-forth paths during bidirectional finish milling, as well as very smooth feed rate curves on neighboring milling paths.

### Fast machining and computing processes

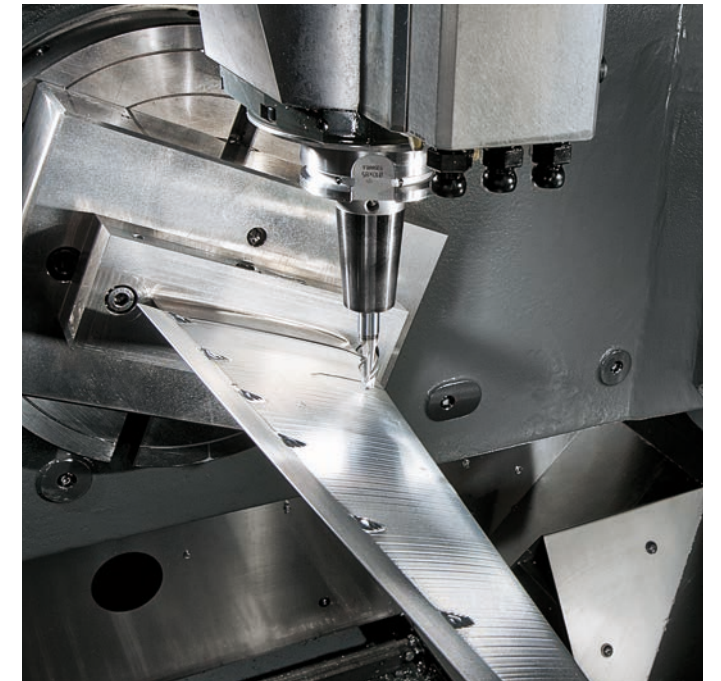
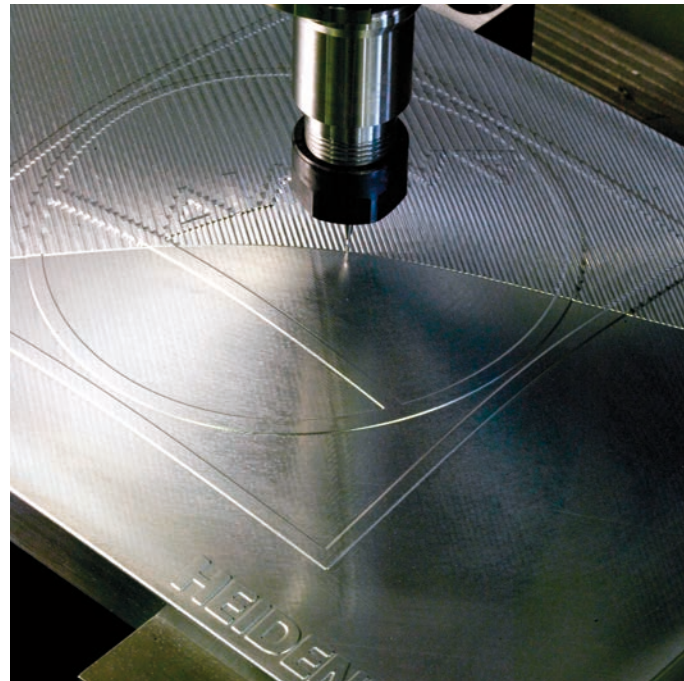
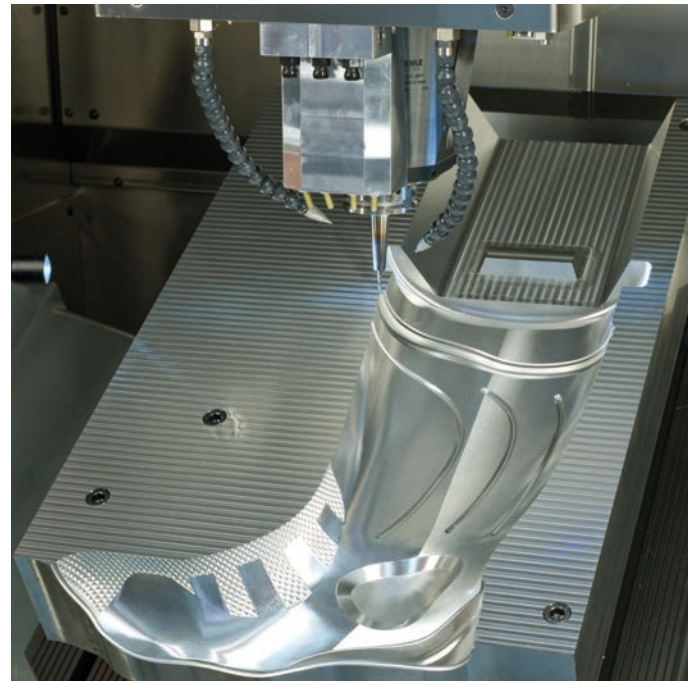
With a short block-processing time of at most 0.5 ms, the TNC 640 can run fast look-ahead calculations for optimal use of the machine's dynamic parameters. Functions such as ADP and Look Ahead therefore ensure not only high contour fidelity and surface quality but also optimize the machining time.

One basis for the TNC 640's high speed is its **uniformly digital control design**. This includes the integrated digital drive technology from HEIDENHAIN as well as digital interfaces connecting all of the control components with each other:

- Control components via HSCI (HEIDENHAIN Serial Controller Interface)
- Encoders via EnDat 2.2

As a result, exceptionally high feed rates can be implemented. In the process, the TNC 640 is capable of interpolating up to five axes at the same time. To attain the required cutting speeds, the TNC 640 can digitally control spindle speeds of up to **100000 rpm**.

The TNC 640's powerful 5-axis machining capability even lets you economically manufacture complex 3D contours. Programs for such contours are usually created offline with CAM systems and typically contain numerous short line blocks that are transferred to the control. With its short block-processing time, the TNC 640 quickly executes even complex NC programs. The control's computing power can handle complex look-ahead calculations in simpler NC programs as well. Thus, regardless of the data volume of the NC programs from your CAM system, the TNC 640 will machine the workpiece to near-perfect conformity with your program.





### Compensating for tool form error

3D-ToolComp (option 92) puts a powerful, three-dimensional tool radius compensation function at your disposal. Angle-dependent delta values describing the deviation of the tool from a perfect circle can be defined in a compensation table (see graphic).

The TNC 640 then corrects the radius value defined for the tool's current point of contact with the workpiece. In order to determine the exact point of contact, the NC program must be created with surface-normal blocks (LN blocks) by a CAM system. The surface-normal blocks specify the theoretical center point of the tool and, if applicable, the tool orientation relative to the workpiece surface.

The ideal method is to create the compensation table fully automatically. This is done by measuring the shape of the tool with a laser system and a special cycle such that the TNC 640 can make immediate use of the table. If the form errors of the tool being used are available in a calibration chart from the tool manufacturer, then you can also create the compensation table manually.

### Measuring 3D geometries

Cycle 444, 3D probing, allows you to measure points on 3D geometries. To do so, you enter the coordinates of the measuring point and the associated normal vectors into the cycle. After probing, the TNC automatically calculates whether the measuring points are within a prescribed tolerance. You can then call the result via system parameters in order, for example, to have the program initiate rework. You can also trigger a program stop and output a message. After measurement, the cycle automatically generates a straightforward measurement report in HTML format. To obtain even more accurate results, you can perform a 3D calibration of the touch probe prior to running Cycle 444. This cycle then compensates for the touch probe's individual triggering behavior in any direction. Option 92 is required for 3D calibration.

CAM systems generate 5-axis programs using a postprocessor. In principle, such programs contain either all of the coordinates of the machine's existing NC axes, or they contain NC blocks with surface normal vectors. During five-axis machining with three linear axes and two additional tilting axes\*, the tool is always perpendicular to the workpiece surface or is tilted at a specific angle relative to it (inclined tool machining).

Regardless of the type of five-axis programs you wish to run, the TNC 640 performs all of the necessary compensating movements in the linear axes arising from movements in the tilting axes. The TNC 640's **TCPM function** (Tool Center Point Management), an improvement upon the proven TNC function M128, provides optimal tool guidance and prevents contour gouging.

\* The machine and TNC must be adapted for these functions by the machine manufacturer.

With TCPM, you can define the behavior of the tilting and compensating movements that are automatically calculated by the TNC 640:

TCPM defines the interpolation between the start and end positions:

- **Face Milling:** In face milling, primary cutting is performed using the front face of the tool (or the corner radius in the case of toroid cutters). In the process, the tool center point moves along its programmed path.
- **Peripheral Milling:** In peripheral milling, primary cutting is performed with the lateral surface of the tool. The tool tip also travels along its programmed path, but machining with the circumference of the tool results in a clearly defined plane. As a result, peripheral milling is well suited for the production of precise surfaces using gear hobbing processes.

TCPM defines the **effect of the programmed feed rate** with two possibilities to choose from:

- The actual speed of the tool tip relative to the workpiece: in the case of large compensating movements while machining close to the center of tilt, very high axis feed rates may arise.

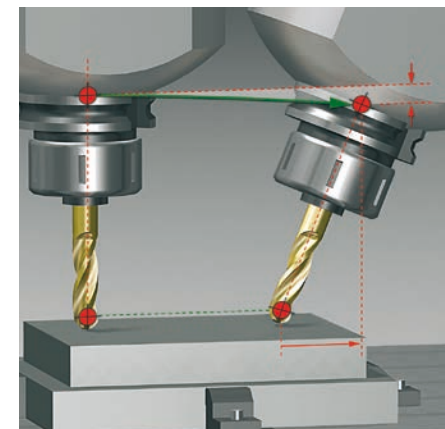
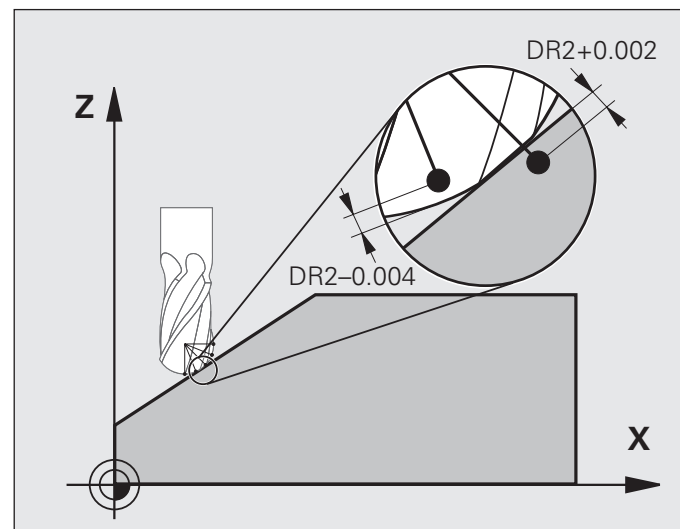
- Contouring feed rate of the axes programmed in the respective NC block: the feed rate is generally lower, but you attain better surface quality with large compensating movements.

In order to achieve better surface quality, an **inclination angle** is often configured for contour machining, with corresponding angle values in the NC program. The effect of the inclination angle can also be configured via TCPM:

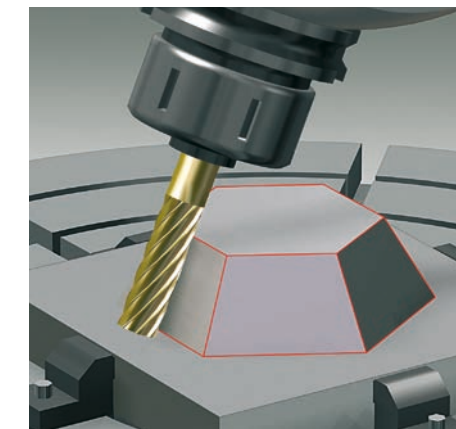
- Angle of inclination defined as an axis angle
- Angle of inclination defined as a spatial angle

The TNC takes the inclination angle into account in all 3D machining operations, including with 45° swivel heads or tilting tables. You can specify the angle of inclination in the NC program via a miscellaneous function or manually set it with an electronic handwheel. The TNC 640 ensures that the tool remains on the contour and does not damage the workpiece.

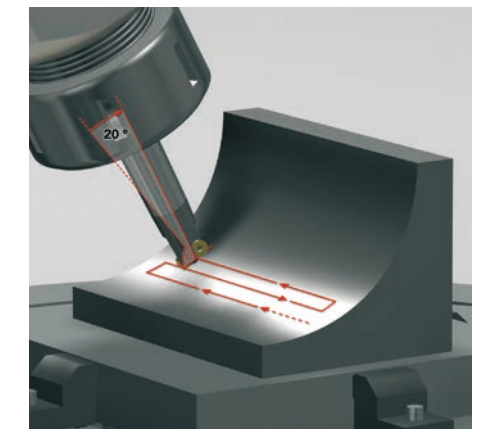
You can use Cycle 444 to measure 3D geometries even when TCPM is active. The TNC 640 then automatically takes the inclination angle of the touch probe into account.



Face milling



Peripheral milling



Inclined-tool machining

# Machining with five axes

## Swivel head and rotary table controlled by the TNC 640

Many five-axis operations that initially appear to be highly complex can be broken down into common 2D movements that have been tilted about one or more rotary axes or wrapped around a cylindrical surface. In order for you to create and edit such programs quickly and easily without a CAM system, the TNC supports you with helpful functions.

### Tilting the working plane\*

Programs for contours and holes on inclined surfaces are often extremely complex and require time-consuming computing and programming work. But the TNC 640 can save you a great deal of programming time. You simply program the machining operation as usual in a plane, such as the XY plane; the machine then performs the operation in a tilted plane.

The PLANE function makes it easy for you to define a tilted working plane: tilted working planes can be specified in seven different ways depending on the information available in the workpiece drawing. Help images support you during program entry.

You can also use the PLANE function to define the positioning behavior during tilting to avoid unpleasant surprises when the program is run. The settings for defining the positioning behavior are identical for all PLANE functions, making operation much easier.

\* The machine and TNC must be adapted for these functions by the machine manufacturer.

### Cylinder surface machining\*

Programming contours made up of straight lines and circles on cylindrical surfaces for rotary tables is no problem with the TNC 640. You simply program the contour in the plane of an unrolled cylinder surface. The TNC 640 then performs the machining operation on the cylindrical surface of the cylinder.

The TNC 640 features four cycles for cylindrical surface machining:

- Slot milling (the slot width and tool diameter are equivalent)
- Guide-groove milling (the slot width is greater than the tool diameter)
- Ridge milling
- Outside contour milling

\* The machine and TNC must be adapted for these functions by the machine manufacturer.

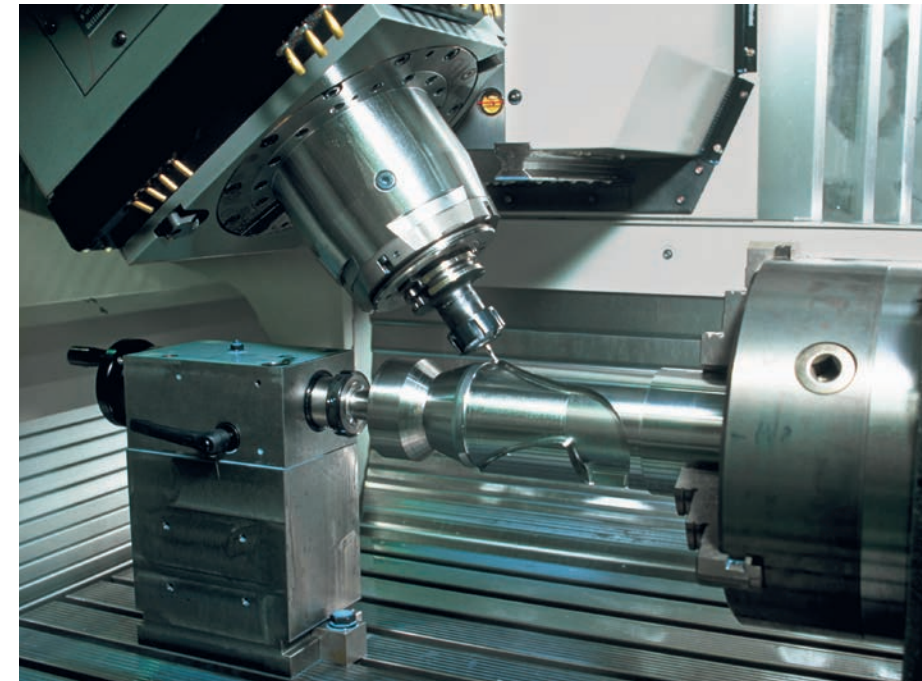
### Manual axis motion in the tool direction on 5-axis machines

Correct tool retraction is very important in five-axis machining. The virtual tool axis function provides assistance by allowing you to move the tool in the direction of the tool axis using external direction keys or a handwheel. This function is especially useful if you want to

- retract the tool in the direction of the tool axis during an interruption of a five-axis machining program,
- use the handwheel or external direction keys to perform an operation in Manual mode with an inclined tool, or
- move the tool with the handwheel in the active tool axis direction during machining.

### Feed rate for rotary tables in mm/min\*

By default, the feed rate of rotary axes is programmed in degrees/min. However, the TNC 640 can interpret this feed rate in mm/min as well. The feed rate at the contour is then independent of the distance of the tool center point from the center of the rotary axis.





# Intelligent machining

## Dynamic Collision Monitoring option (DCM)

The complex machine movements and normally high traversing speeds of five-axis machining make axis movements difficult to predict. Collision monitoring is thus a valuable function that makes things easier for the machine operator and protects the machine from damage.

NC programs from CAM systems may be able to avoid collisions of the tool or tool holder with the workpiece but do not account for machine components in the workspace—unless you invest in expensive offline machine simulation software. But even then, there is no guarantee that the arrangement of

machine components still matches the simulation. In the worst case scenario, collisions won't manifest themselves until the part is actually machined.

For cases such as these, the **Dynamic Collision Monitoring (DCM)\*** feature of the TNC 640 provides the machine operator with peace of mind. The control interrupts the machining process whenever a collision is imminent, thus increasing safety for both the machine and its operator. This, in turn, helps prevent machine damage and any resulting costly downtime. Unattended shifts also become more reliable.

Dynamic Collision Monitoring (DCM) even works in both **automatic mode** and **manual operation**. If the machine operator is on a "collision course" during workpiece setup, for example, then the TNC 640 will detect the impending collision and stop the axis with an error message.

The fixture import feature of DCM not only enables graphical representation of the fixtures, but also provides for collision checking during simulation and the actual machining process. An additional level of reliability is gained with the new enhanced checking for collisions between the workpiece and the non-cutting part of the tool or the tool holder.

\* The machine and TNC must be adapted for these functions by the machine manufacturer.

Of course, the TNC 640 also shows the operator which machine components are on a path toward collision, both in the error message and in a graphical depiction. If a collision warning occurs, the TNC permits tool retraction only in directions that will increase the clearance between the colliding objects.

The machine manufacturer defines the machine components by using geometric bodies to describe the workspace and the collision objects. For tilting devices, the machine manufacturer can also use the description of the machine kinematics to define the collision objects.

A new 3D file format for collision objects offers further appealing benefits:

- Convenient data transfer of standard 3D file formats
- Fully detailed illustration of machine components
- Greater exploitation of the machine's workspace

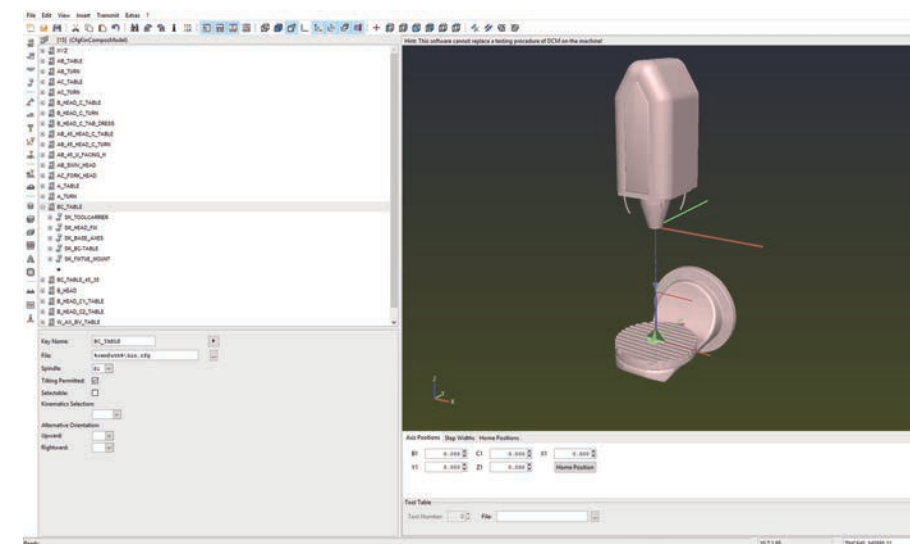
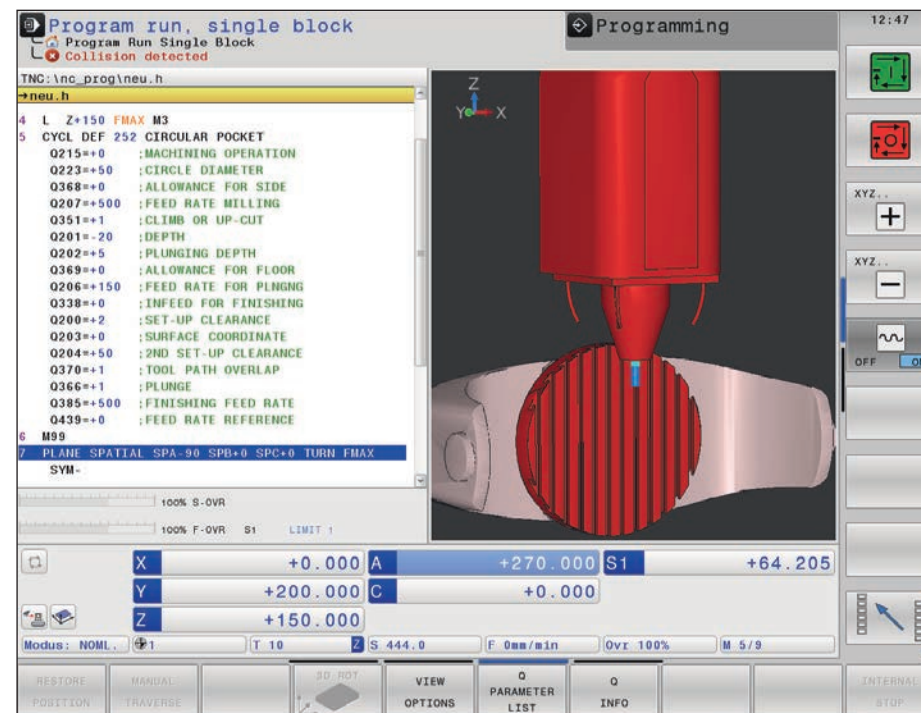
The TNC 640 can also monitor tool carriers such as holders for milling cutters or touch probe housings. To this end, a tool-carrier kinematic model is assigned to the tool in the tool table. When the tool is inserted, the respective tool carrier is activated in the collision monitoring function.

Because the machine design itself prevents collisions between certain machine components, not all of the machine parts require monitoring. For

example, a HEIDENHAIN TT tool touch probe clamped to a machine table for tool measurement will never collide with the machine cabin. The machine manufacturer can therefore specify which machine elements could collide with each other.

When using Dynamic Collision Monitoring, please note the following:

- DCM can help reduce the danger of collision, but DCM cannot completely prevent all collisions.
- Only the machine manufacturer can define machine components.
- Collisions between machine components (such as a swivel head) and the workpiece cannot be detected.
- DCM cannot be used in servo-lag operation (no feedforward).
- DCM cannot be used for eccentric turning.





Under the rubric of **Dynamic Efficiency**, HEIDENHAIN offers innovative TNC functions that help the user make heavy machining more efficient and improve process reliability. These software functions not only support the operator but also make production processes faster, more reliable, and more predictable—in other words, more efficient. Dynamic Efficiency permits higher removal rates and therefore increases productivity without making the user resort to special tools. At the same time, it prevents tool overloading and premature cutter wear. Dynamic Efficiency makes your manufacturing overall more economical while increasing process reliability.

**Dynamic Efficiency** encompasses four software functions:

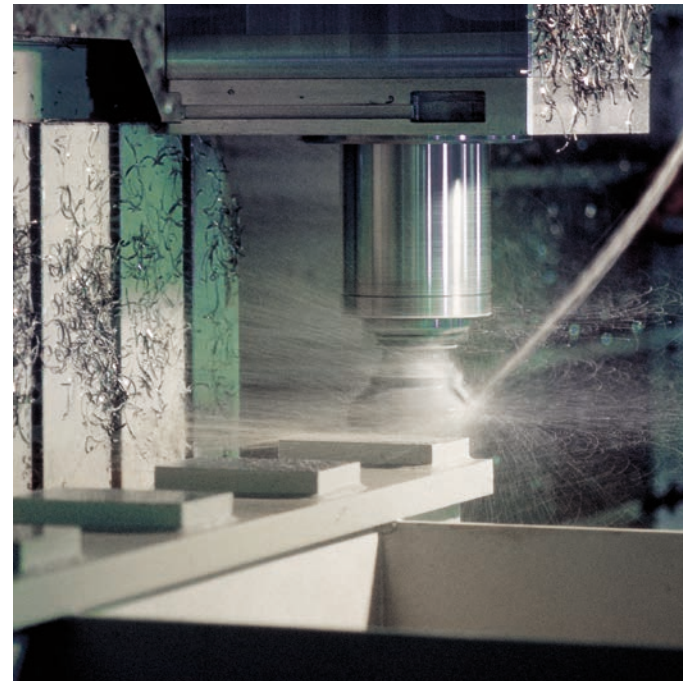
- **Active Chatter Control (ACC)** is an option for reducing chatter tendencies and permits higher infeeds
- **Adaptive Feed Control (AFC)** is an option for regulating the feed rate based on the machining situation
- **Trochoidal milling** is a function for tool- and machine-friendly roughing of slots and pockets
- **Optimized Contour Milling (OCM)** is an option for tool-friendly machining of any pocket and island under consistent process conditions

Individually, each of these solutions delivers key improvements to the machining process. But in combination, these TNC functions bring out the full potential of the machine and tool while reducing the mechanical load. They also prove their worth under changing machining conditions, such as interrupted cuts, varying plunging processes, or simple roughing operations. In practice, removal rate increases of **20 to 25 percent** are possible.

Roughing operations (power milling) give rise to strong milling forces. Depending on the rotational speed of the tool, the resonances in the machine tool, and the material removal rate during milling, the tool may begin to “chatter.” Chatter puts the machine under heavy strain and causes blemishes on the workpiece surface. Tool wear is also accelerated and less evenly distributed. In extreme cases, the tool may even break.

To reduce chatter susceptibility, HEIDENHAIN now offers a highly effective optional control function called Active Chatter Control (ACC). This option is particularly beneficial during heavy machining:

- Improved cutting performance
- Higher removal rates (up to 25 % and more)
- Lower forces on the tool, with increased tool life
- Less strain on the machine



Heavy machining without ACC (upper image) and with ACC (lower image)





### dynamic + efficiency

HEIDENHAIN controls have always allowed the operator to enter the feed rate for each block or cycle and also manually adjust the feed rate to the actual machining situation using an override potentiometer. But this approach relies on the experience of the operator and requires him to be at the machine.

Adaptive Feed Control (AFC) automatically regulates the feed rate of the TNC, taking into consideration the given spindle power and other process data. The TNC first records the maximum spindle power during a teach-in cut. Prior to machining, you define the limits (in a table) within which the TNC is allowed to influence the feed rate in "control" mode. You can flexibly choose among various overload reactions that have been defined by the machine manufacturer.

Adaptive Feed Control (AFC) offers a series of benefits:

#### Process reliability

Roughing at high material removal rates involves strong cutting forces, which in practice can cause tool defects. If the user is unable to react quickly, perhaps because he is responsible for multiple machines or because the shift is unattended, then significant secondary damage and cost may be incurred:

- Costly rework on the workpiece
- Irreparable workpiece damage
- Damage to the tool holder
- Machine downtime due to spindle damage

Continuous monitoring detects any increase in spindle power due to tool wear or defective inserts, and a replacement tool

can be automatically inserted.\* In this manner, AFC prevents potential secondary damage due to tool wear while increasing process reliability.

#### Reduced machining time

AFC regulates the feed rate of the TNC based on the given spindle power. In machining areas requiring less material removal, the feed rate is appropriately increased. This can significantly reduce the machining time.

#### Gentler on the machine

When the maximum teach-in power is exceeded, the feed rate is reduced down to the reference spindle power. As a result, the machine undergoes less strain, and the main spindle is protected from overloading.

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

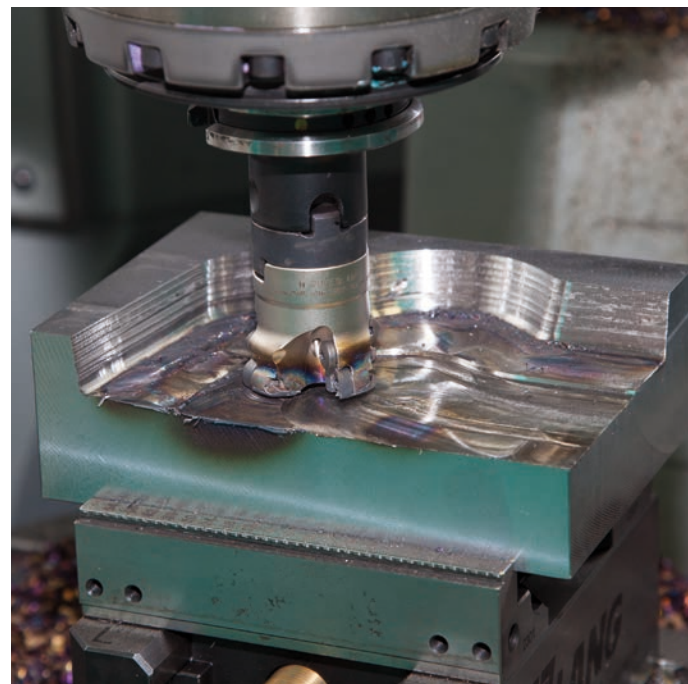
### dynamic + efficiency

The benefit of trochoidal milling is its ultra-efficient machining of slots of all kinds. In this cycle, roughing is performed with circular movements, onto which a forward linear movement is superimposed. This procedure is referred to as trochoidal milling. It is used particularly for milling high-strength or hardened materials, where the high loads placed on the tool and machine usually only permit small infeeds.

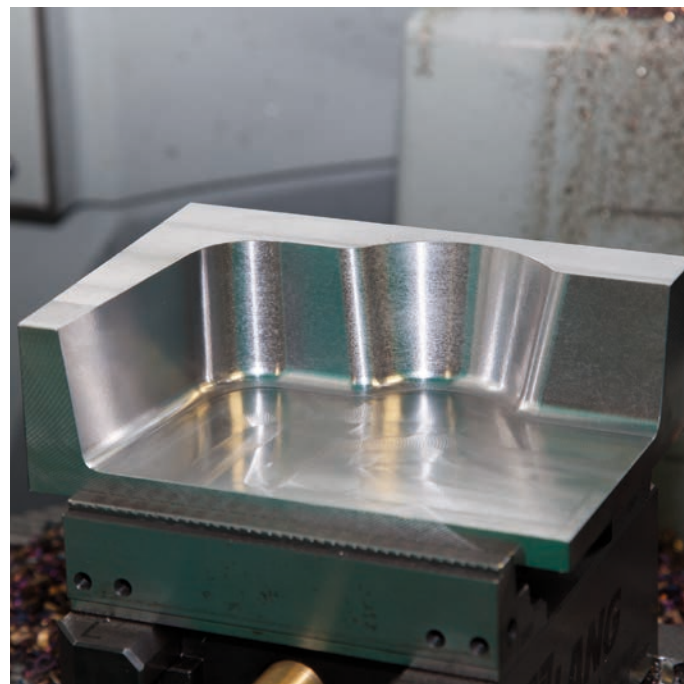
Thanks to the special cutting motion used in trochoidal milling, however, there are no added wear-inducing influences on the tool, and higher infeeds are possible. During hobbing, the entire cutting edge can be used as well. You thereby attain a higher material removal rate per tooth. Circular-motion plunging into the material exerts less radial force on the tool. This reduces the mechanical load on the machine and prevents vibration. Enormous time savings can be realized by combining this milling method with the integrated Adaptive Feed Control (AFC) option.

The slot to be machined is described in a contour subprogram as a contour train. You define the dimensions of the slot and the cutting data in a separate cycle. Any residual material can then be easily removed with a subsequent finishing cut.

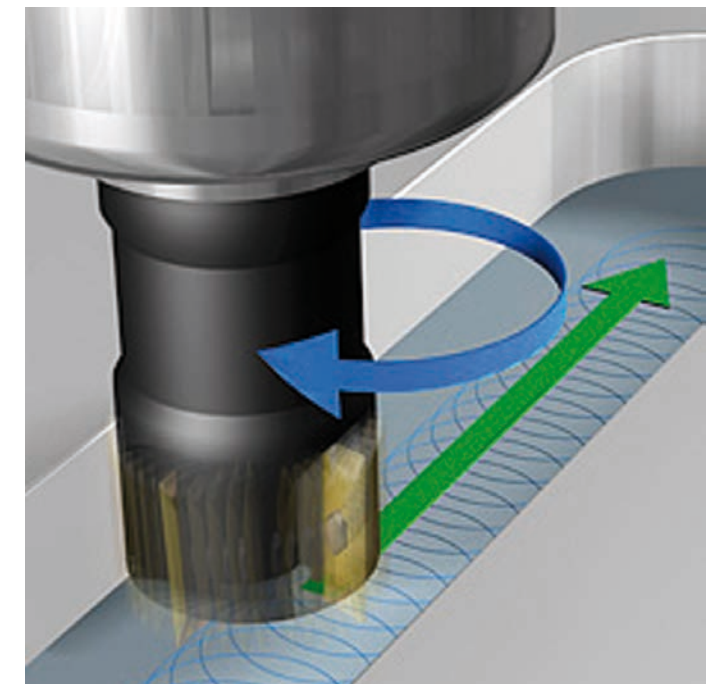
- The main benefits are as follows:
- Contact over entire cutter length
  - Higher removal rates
  - Less strain on the machine
  - Less vibration
  - Integrated finishing of the side wall
  - Superior chip removal



Workpiece with secondary damage resulting from a broken indexable insert



Fully machined workpiece protected by AFC



# Intelligent machining

## Optimized roughing processes with OCM (option)

Efficient machining strategies are a key basis for economical NC manufacturing. Roughing processes, in particular, offer great potential for optimization because they usually make up a sizeable portion of the total machining time.

With OCM, you can rough out any pocket and island with high process reliability and lower tool wear thanks to highly consistent process conditions. You simply program the contour as usual directly in Klartext or make use of the convenient CAD Import function. The control then automatically calculates the complex movements required for maintaining consistent process conditions.

Process reliability and maximum removal rates require optimal adaptation of the cutting data to the characteristics of the tool and part material. OCM gives you a cutting data calculator that draws on a comprehensive integral materials database. You can then adapt the automatically calculated cutting values by taking the specific mechanical and thermal load on the tool into account. Tool service life can be managed with process reliability even at the highest possible removal rates.

### Advantages of OCM over conventional machining:

- Reduced thermal load on the tool
- Superior chip removal
- Uniform cutting conditions

### OCM is an effective, reliable, and convenient way to improve your throughput

- Create programs on the shop floor for any pocket or island
- Significantly higher machining speed
- Considerable reduction in tool wear
- More chips in less time

The OCM option provides practical cycles for roughing, side-wall finishing, and floor finishing.

OCM also permits chamfering and deburring of contours. With this capability, only those areas that can be machined without collisions due to the tool geometry are machined.

#### Conventional machining

S5000, F1200,  $a_p$ : 5.5 mm

Overlap factor: 5 mm

Machining time: 21 min 35 s

Tool: VHM end mill (Ø 10 mm)  
Workpiece material: 1.4104

#### Machining with OCM

S8000, F4800,  $a_p$ : 22 mm

Overlap factor: 1.4 mm

Machining time: 6 min 59 s

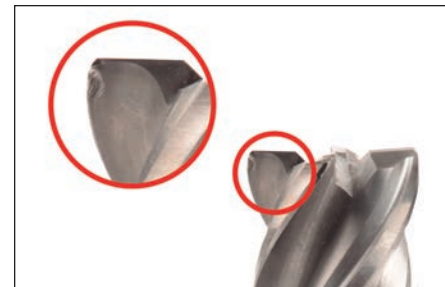
Tool: VHM end mill (Ø 10 mm)  
Workpiece material: 1.4104



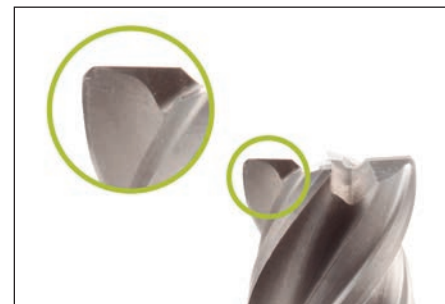
## dynamic + efficiency

In order to machine standard shapes, OCM offers various figures that are then used as a pocket, island, or boundary for face milling in conjunction with other OCM cycles.

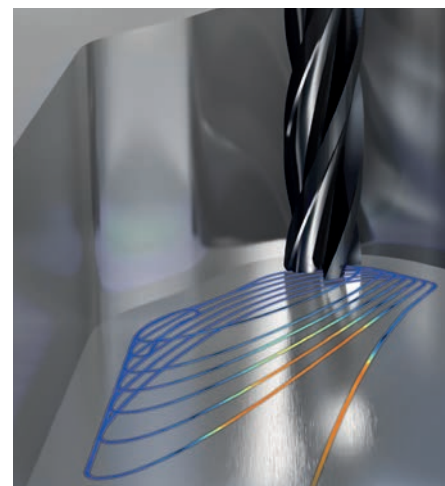
In the following machining example, both the machining time and tool wear were reduced **by a factor of three**.



Tool after two parts



Tool after six parts



## Global Program Settings (option)

The Global Program Settings option is especially used in large-scale moldmaking and is available in Program Run and MDI modes. This option allows you to perform a variety of coordinate transformations and settings, and then apply them globally to a selected NC program without actually modifying it.

You can even change the Global Program Settings mid-program during a program stop using a handy fillable form. When the program is started, the TNC 640 moves to a new position as needed in accordance with a positioning logic that you have defined.

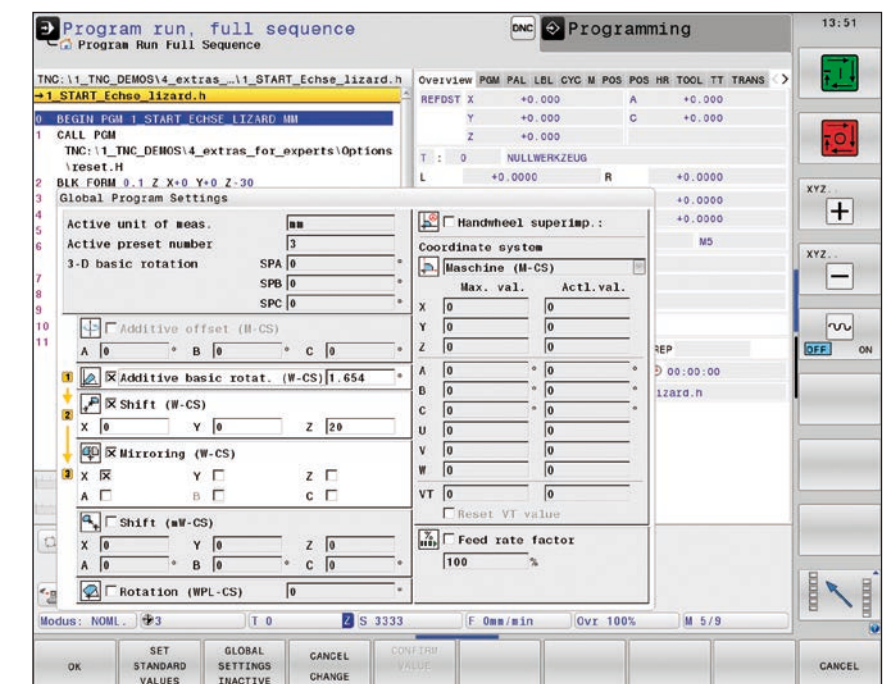
The following functions are available:

- Additional, additive datum shift
- Superimposed mirroring
- Handwheel superimpositioning, with axis-specific memory of the paths covered by the handwheel, including in a virtual axis direction
- Superimposed basic rotation
- Superimposed rotation
- Globally valid feed-rate factor
- Mirroring of axes

Handwheel superimpositioning is possible in various coordinate systems:

- Machine coordinate system
- Workpiece coordinate system (the active basic rotation is taken into account)
- Tilted coordinate system

You can select the desired coordinate system in a clearly structured form.





# Automated machining

The TNC 640 measures, manages, and communicates

The requirements gap between machining centers and typical tool and moldmaking machines is becoming ever narrower. Of course, the TNC 640 is also very capable of controlling automated manufacturing processes. It features the necessary functionality to initiate the proper machining operation in concatenated machining scenarios with individual workpieces in any workholding arrangement.

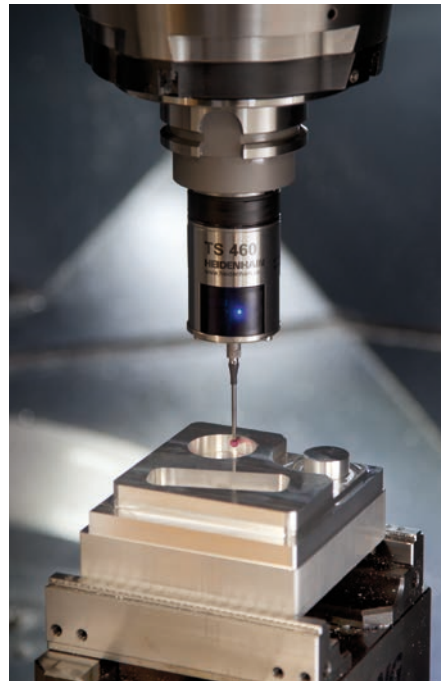
## Inspecting workpieces for completed machining and dimensional accuracy

The TNC 640 features numerous measuring cycles for inspecting the geometry of machined workpieces. For the measuring cycles you insert a touch probe from HEIDENHAIN (see page 50) into the spindle in place of a tool. The resulting functionality allows you to do the following:

- Recognize a workpiece and call the appropriate part program
- Check whether all machining operations were conducted correctly
- Determine infeeds for finishing
- Detect and compensate for tool wear
- Check the workpiece geometry and classify parts
- Log measured data
- Ascertain the machining error trend
- Check whether tolerances were maintained, and configure various reactions, such as an error message

## Milling cutter measurement and automatic tool data compensation

In conjunction with the TT tool touch probe (see page 51), the TNC 640 lets you automatically measure milling tools in the machine. The TNC 640 then stores the determined values, such as tool length and radius, in its tool memory. By inspecting the tool during machining, you can quickly detect wear or breakage for preventing scrap and rework. If the measured deviations are out of tolerance, or if the monitored tool life is exceeded, then the TNC 640 locks the tool and automatically inserts a replacement tool.



# Pallet management and serial machining

## Pallet management

Pallet management functionality lets you automatically machine workpieces in any order. Upon insertion of the pallet, the correct machining program and preset are automatically selected. You can also apply coordinate transformations and measuring cycles in the part programs.

## Batch Process Manager (option)

Batch Process Manager is a powerful function for pallet machining and serial production. With its intuitive user interface, you can plan your production process and receive important information about upcoming machining operations.

Batch Process Manager automatically checks for missing tools, insufficient tool life, and required manual tool changes. The results are displayed in the status overview.

Batch Process Manager shows the following information in advance:

- The machining sequence
- The time of the next manual intervention
- The program duration and run time
- Status information about the preset, tool, and program

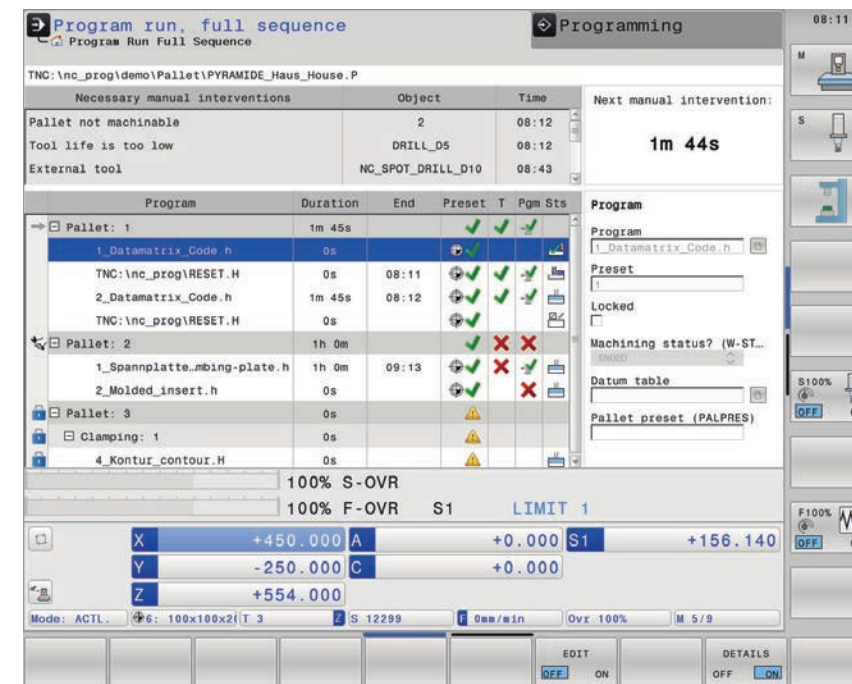
In Program Run mode, you also have the option of using soft keys to perform a collision check for all pallet subprograms with active collision checking (possible only with DCM (software option 40)).

## Tool-oriented machining

In tool-oriented machining, a single machining step is performed on all of a pallet's workpieces before moving to the next machining step. This keeps the number of tool changes to a minimum and significantly shortens the machining time.

The TNC 640 supports you with convenient fillable forms that allow you to assign a tool-oriented operation to a pallet with multiple workpieces. As always, you still write the machining program in the normal workpiece-oriented sequence.

You can also use this function even if your machine does not support pallet management. In this case, you simply define the position of the workpiece on the machine table in the pallet file.



## Tool management

For machining centers with an automatic tool changer, the TNC 640 offers a central tool memory area for numerous milling, grinding, dressing, and turning tools. This tool memory area is a freely configurable file that can be optimally adapted to your needs. You can even have the TNC 640 manage your tool names. The control prepares the next tool change while the current tool is still cutting. This significantly reduces the non-cutting time required for tool changes.

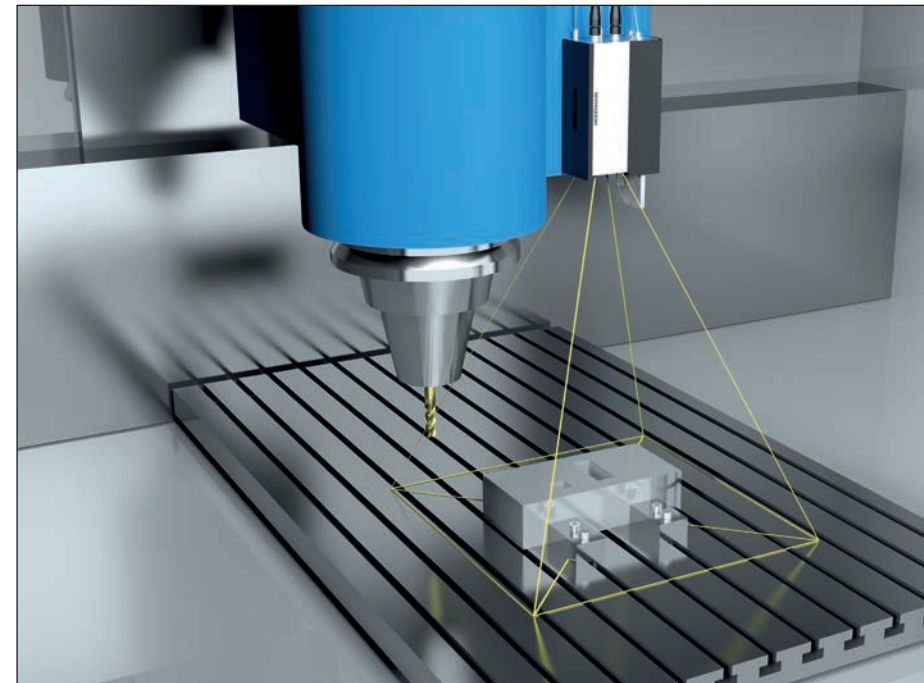
With the Expanded Tool Management option, the control shows only the required input fields regardless of the selected tool type. Any data can also be graphically depicted.\*

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



## Workspace monitoring

With the Visual Setup Control (VSC) option, the TNC can automatically monitor the current setup or machining situation during program run. This option uses a camera system to take reference photos of the first parts in a series in order to then compare them with photos of subsequent parts. User-friendly cycles let you specify multiple locations within the NC program at which the control should perform a visual comparison of the nominal-to-actual condition. If an error is detected, the TNC reacts in a manner selected by the user. VSC not only helps you avoid expensive damage to the tool, workpiece, and machine, but it also lets you identify missing machining operations or document recurring setup configurations.



## Component monitoring

During their lifecycle, load-sustaining machine components such as guideways and ball screws exhibit wear, thus causing the quality of axis motion to diminish. This, in turn, affects production quality. With Component Monitoring (option 155) and a cycle, the control is able to measure the current condition of the machine. As a result, any deviations from the machine's shipping condition due to wear and aging can be measured. The machine manufacturer can read and evaluate the data, and react using predictive maintenance, thereby avoiding unplanned machine downtimes.

# Minimize setup times

## The TNC 640 makes setup easy

Before machining can begin, the workpiece must first be clamped, the machine must be set up, the position of the workpiece in the machine must be determined, and the preset must be set. This is a time-consuming but indispensable procedure because any error directly reduces the machining accuracy. Particularly in small and medium-sized production runs, as well as in the case of very large workpieces, setup times become a significant factor.

The TNC 640 features application-oriented, real-world setup functions that support the user, help reduce non-productive time, and enable overnight, unattended production. In conjunction with the **touch probes**, the TNC 640 offers numerous probing cycles for automatic workpiece setup, setting the preset, and measuring workpieces and tools.

## Fine manual axis control

During setup, you can use the axis-direction keys to move the machine axes manually or with an incremental jog. A simpler and more reliable way, however, is to use the electronic handwheels from HEIDENHAIN (see page 53). With these handwheels, you are always close to the action, enjoy a close-up view of the setup process, and can control the infeed responsively and precisely.

## Adapting the probing speed

In many cases, the workpiece must be probed at unseen positions or in cramped spaces, and the standard probing feed rate is usually too fast for this. In such situations, you can use the override knob to change the feed rate during probing without affecting the accuracy.

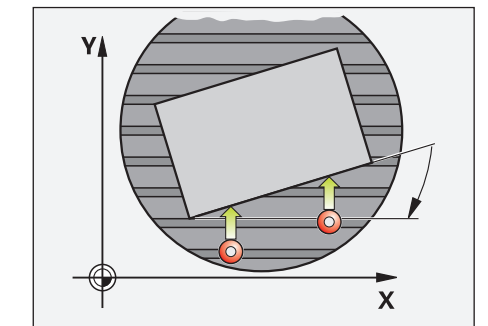
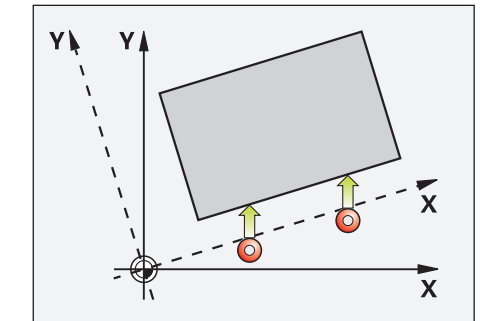


## Workpiece alignment

Touch probes from HEIDENHAIN (see page 50) and the probing functions of the TNC 640 keep you from having to perform time-consuming workpiece alignment:

- You start by simply clamping the workpiece in any position.
- The touch probe probes a surface to determine the amount of workpiece misalignment.
- The TNC 640 subsequently compensates for the misalignment by performing a "basic rotation," in which either the NC program is executed at the ascertained angle of rotation or the rotary table is turned in order to correct the misalignment.
- The TNC 640 offers manual, automatic, and semiautomatic cycles for compensating for misalignments in two or three dimensions

**Workpiece misalignment compensated for** by rotating the coordinate system or rotating the table





# Programming, editing, and testing

## A full range of possibilities with the TNC 640

### Setting presets

A preset allows you to assign a defined value displayed by the TNC to any position on the workpiece. Finding this point quickly and reliably reduces non-productive time and increases machining accuracy.

The TNC 640 features probing cycles for automatically setting presets. Ascertained presets can be saved as follows:

- Using the preset management
- In a datum table
- Through direct setting of the display

### Preset management with the preset table

Preset management enables flexible machining, shorter setup times, and higher throughput. Setting up your machine is greatly simplified.

In preset management, you can save **any number of presets** and assign an individual basic rotation to each one. In order to permanently save fixed presets in the machine's workspace, you can also write-protect individual lines.

There are three possibilities for rapidly saving the presets:

- Via a soft key in Manual mode
- Via the probing functions
- With the automatic probing cycles

### Saving datums

Datum tables let you save positions or measured values from the workpiece. Datums are always defined relative to the active preset.

Besides being universally deployable, the TNC 640 is equally flexible when it comes to programming and machining.

### Positioning with Manual Data Input

You don't need to create a complete program in order to start machining with the TNC 640. You can machine a workpiece step by step using both manual machining and automatic positioning in any sequence.

### Programming at the machine

HEIDENHAIN controls are designed for shop-friendly programming right at the machine. Instead of requiring you to know G-codes, the Klartext conversational programming language provides its own buttons and soft keys for programming straight lines, arcs, and cycles. At the push of a button, HEIDENHAIN Klartext dialog guidance opens and actively helps you during programming on the TNC. Its clear instructions prompt you to enter all of the required data.

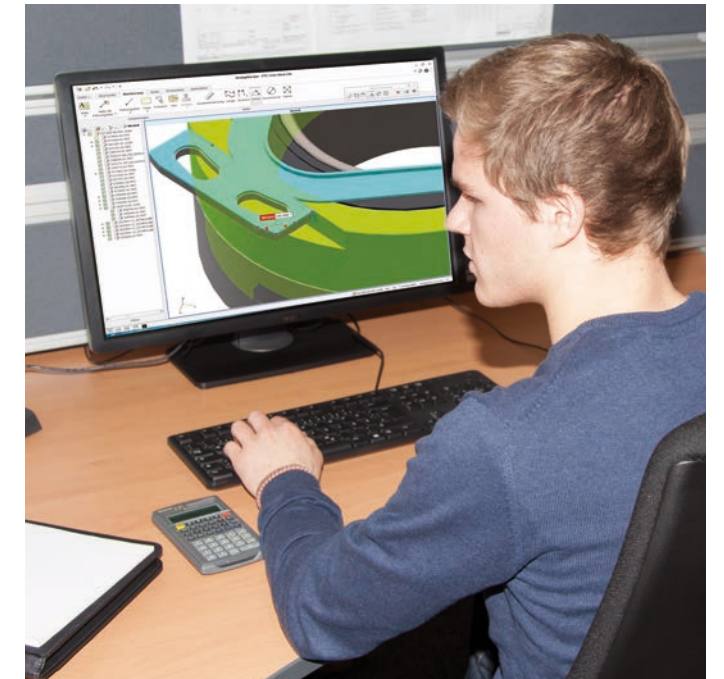
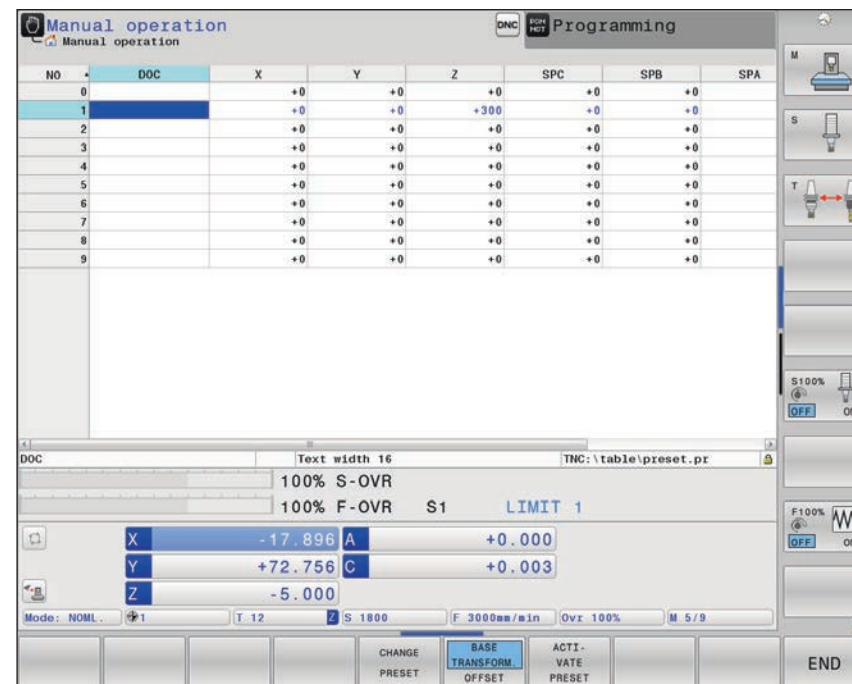
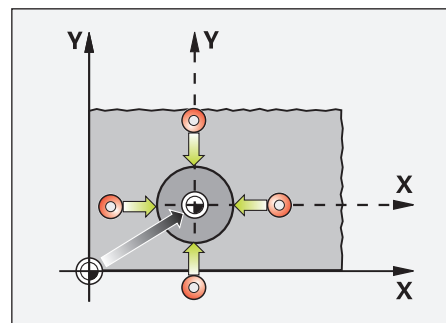
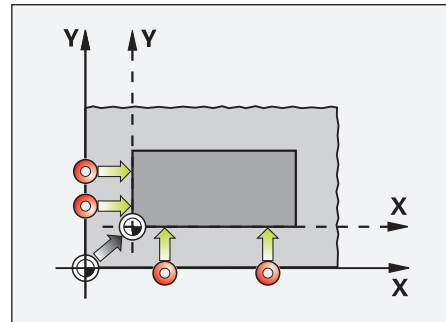
All interface texts, including Klartext tips, dialog prompting, program steps, and soft keys, are available in numerous languages.

And even if you prefer **ISO programming**, the TNC is still the right control: you can enter ISO command letters via soft keys or directly on the alphanumeric keyboard.

### Creating programs offline

The TNC 640 is also well equipped to handle offline programming. Through its interfaces it can be integrated into networks and connected with programming stations or other data storage devices.

Setting a preset at a corner, for example, or in the center of a circular stud



**Programming graphics**

The two-dimensional programming graphics give you additional assurance: while you are programming, the TNC 640 renders every programmed movement on the screen. You can opt for a plan view, side view, or front view. Tool paths or rapid-traverse movements can be hidden, and the image's scale can be changed.

**Program verification graphics**

For added certainty prior to machining, the TNC 640 can simulate and render the machining of the workpiece at high resolution. The TNC 640 provides various view formats of the simulation:

- Plan view at different levels of depth
- Three projections
- 3D view

The type and quality of the image can be selected, and details can be displayed at magnification. The TNC 640 also displays the calculated machining time in hours, minutes, and seconds. You can use STL files to integrate workpiece blanks and finished parts in the program.

The 3D view lets you display the programmed tool-center path in three dimensions, and a powerful zoom function lets you see even the tiniest details. The 3D line graphics are especially helpful in verifying programs created offline, allowing you to avoid undesired machining marks on the workpiece, such as those that occur when the postprocessor outputs points incorrectly. The 3D view of the TNC also provides a measuring function. When you move the mouse pointer anywhere within the image, the coordinates of that position are displayed. The measurement window also displays tool information.

During the machining simulation, the TNC 640 can also display all of the machine components defined by the machine manufacturer, in addition to the workpiece and tool. This rendering capability is particularly useful in Test Run mode, informing you ahead of time which positions are too close to components and when the traversing path will be insufficient. This is a huge benefit during machining with tilting axes.

**Program-run graphics**

The program-run graphics display the workpiece in real time in order to show you the machining progress. Direct observation of the workpiece is usually impossible due to coolant and the safety enclosure. During machining, you can switch between various operating modes at any time in order to create programs, for example. And at the press of a button, you can cast a glance at the machining progress while programming.

**Programming 2D contours**

Two-dimensional contours are the bread and butter of a modern machine shop, which is why the TNC 640 offers a variety of ways to handle them. And regardless of whether you are programming a milling or turning contour, you can always use the same tools. This allows you to program as always without a shift in thinking.

**Programming with function keys**

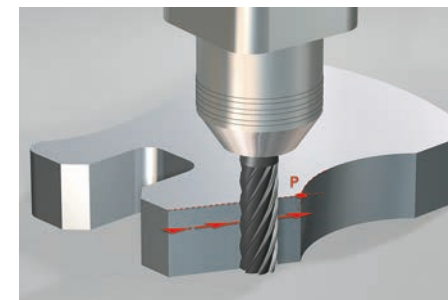
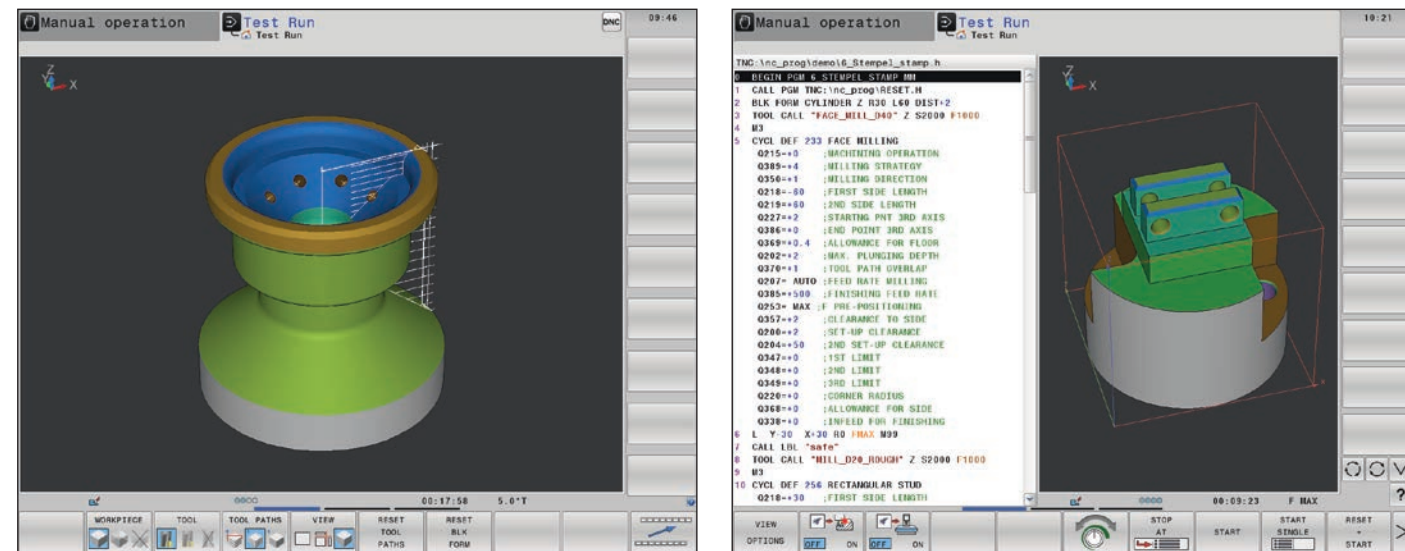
Are the contours properly dimensioned for NC machining? In other words, are the end points of the contour elements provided in either Cartesian or polar coordinates? If yes, then you can create the NC program straightaway using function keys.

**Straight lines and circular elements**

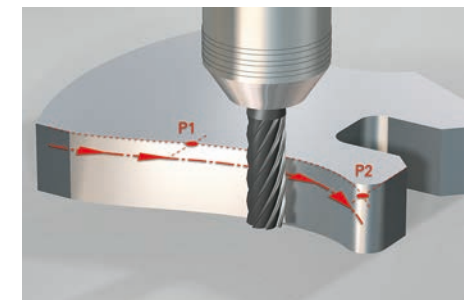
To program a line segment, for example, simply press the linear traverse key. The Klartext conversational programming language of the TNC 640 then prompts you for all of the information needed by the programming block, including the target coordinates, feed rate, tool compensation value, and machine functions. Function keys for circular movements, chamfers, and corner rounding also simplify your programming work. In order to avoid surface blemishes during approach or departure from the contour, movement must be seamless—that is, tangential.

You simply specify the starting point or end point of the contour and the approach or departure radius of the tool. The control handles the rest.

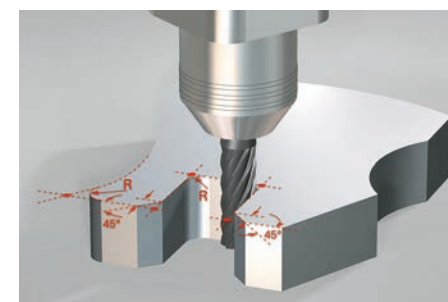
The TNC 640 can foresee a contour with radius compensation by up to 99 blocks, thus taking back-cutting into account and avoiding contour gouging. This could be the case, for example, when roughing a contour with a large tool.



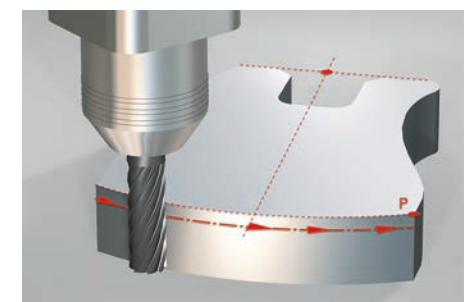
**L** Straight line defined by its end point



**CT** Circular path with smooth (tangential) connection with the preceding contour element, defined by end point

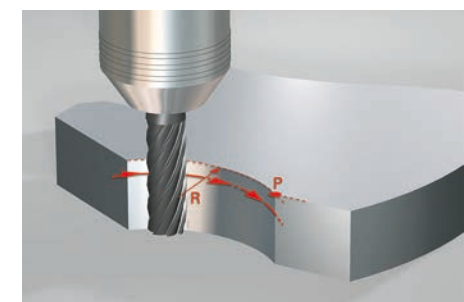


**RND** Corner rounding: circular path with a seamless (tangential) transition, defined by its radius and corner point



**CC + C** Circular path defined by its center, end point, and rotational direction

**CHF** Chamfer defined by the corner point and chamfer length

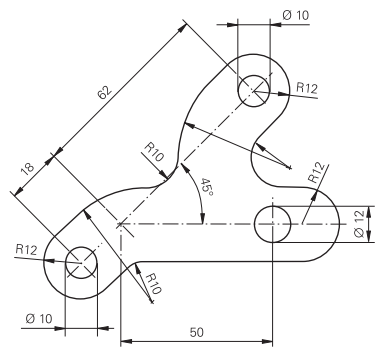


**CR** Circular path defined by its radius, end point, and rotational direction



**FK free contour programming**

Not all workpieces are properly dimensioned in accordance with the ISO standard. Thanks to FK, the control's programming function for free contours, you can simply enter the known data without needing to convert or calculate anything! In such cases, individual contour elements can remain undefined as long as the overall contour itself is defined. If the data from the drawing yields more than one possible mathematical solution, then the helpful TNC 640 programming graphics show you the possible variants for your selection.



**Programming machining patterns with ease and flexibility**

Positions to be machined are often arranged in patterns on the workpiece. The TNC 640's graphical support helps you program a wide range of machining patterns with ease and exceptional flexibility. You can define as many point patterns as desired with various numbers of points. During machining, you can then have all of the points executed as a unit, or each point individually.



**Fast and easy programming of a scannable Data Matrix code**

Cycle 224 (Data Matrix code pattern) lets you convert any text into a Data Matrix code and machine it onto your workpiece as a point pattern using, say, drilling operations. The Data Matrix code can be decoded with common scanning devices. Serial numbers and production data can thus be permanently machined onto the part. All you need to do after defining a machining cycle is enter the text (up to 255 characters) into the cycle and specify the size and position of the drilling pattern. The control automatically calculates the Data Matrix code and performs the machining work on its own.



**Extensive machining cycles for milling and drilling**

Frequently recurring operations consisting of multiple machining steps are stored as cycles in the TNC 640. Dialog guidance and help graphics illustrating the required input parameters support you in programming them.

**Standard cycles**

In addition to drilling and tapping cycles (with or without a floating tap holder), a variety of selectable cycles are available:

- Thread milling
- Reaming
- Engraving
- Boring
- Hole patterns
- Milling cycles for face milling a flat surface
- Roughing and finishing of pockets, slots, and studs



**Cycles for complex contours**

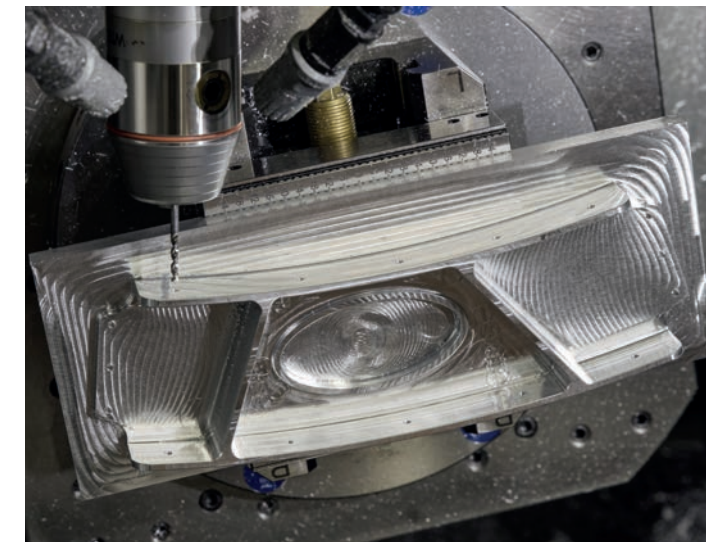
Of great help in roughing pockets with any contour are the control's **SL cycles** (SL = Subcontour List) and **OCM** (Optimized Contour Milling) functionality. These functions refer to machining cycles for pilot drilling, roughing, and finishing in which the contour or subcontours are defined in subprograms. Thus, a single contour description can be used for more than one operation with different tools.

Up to twelve **subcontours** can be superimposed for the machining operation. The control automatically calculates the resulting contour and the tool paths for roughing or clearing the surfaces. Subcontours can be pockets or islands. Multiple pockets are combined to form a single pocket, with the tool bypassing any islands.

During roughing, a **finishing allowance** on the wall and floor surfaces is also taken into account by the TNC 640. When **roughing** with different tools, the control identifies residual material to be removed by smaller tools. A separate cycle is used for finishing to the final dimensions.

**OEM cycles**

Machine manufacturers can incorporate their specialized production expertise in the form of additional machining cycles and store them in the TNC 640. The end user can write his own cycles as well. HEIDENHAIN makes this possible with its PC software CycleDesign, which allows you to create input parameters and arrange the TNC 640's soft key structure as desired.



# Programming in the shop

## Practical turning cycles (option)

### 3D machining with parametric programming

Simple, mathematically easy-to-describe 3D geometries can be programmed by means of parametric programming. At your disposal are basic arithmetical operations, trigonometric functions, roots, powers, logarithmic functions, parentheses, and logical comparisons with conditional jump instructions. Parametric programming also gives you a simple way to implement 3D operations. Of course, parametric programming is also well-suited for **2D contours** that cannot be described with lines or circular arcs but rather through mathematical functions.

### Producing gears more effectively\*

Highly complex motion sequences are necessary during the machining of gears. The TNC 640 gives you multiple cycles that allow you to produce gears easily and economically. You can choose between these two machining technologies:

- Skiving
- Hobbing

During **skiving**, cutting action is performed by means of an axial tool feed and a rolling motion. Skiving is suitable for the machining of straight or oblique internal and external gear teeth. Significant advantages can be gained through skiving particularly when machining internal gear teeth. The continuous cutting process involved also enables exceptional surface quality with short process times.

**Hobbing** is a continuous cutting method suitable for the production of straight or oblique external gear teeth. This method lets you manufacture gears productively, accurately, and flexibly.

In both methods, axial motion is superimposed onto a synchronized rotation of the tool and workpiece. The tool and the workpiece axis must both be positioned at defined angles relative to each other. The TNC 640 handles these complex calculations for you and automatically executes all of the required motion and synchronization.

\* Option 157 (with turning mode, option 50 as well)

The TNC 640 also offers an extensive and technologically sophisticated package of turning cycles. These cycles are equivalent to the well-proven and long-standing core functions of the HEIDENHAIN lathe controls. The appearance and functions of the user interface, however, are aligned with the familiar and well-proven Klartext programming language. Cycle parameters used for both milling and turning are naturally designated by the same number. Graphical illustrations assist you with program entry in the usual manner for turning as well.

### Machining simple contours

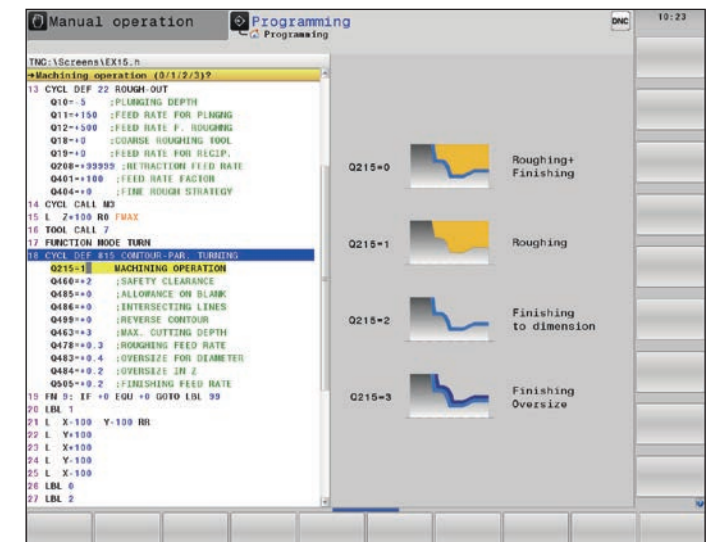
Various cycles are available for the longitudinal and transverse machining of simple contours. The area to be machined can also be inclined, thus requiring a plunging movement. Of course, the TNC 640 automatically takes the angle of the turning tool into account.

### Machining any contours

If the contours to be machined are complex and can no longer be defined with simple cycle parameters, you can still describe them using contour subprograms. This process is identical to the procedure used for SL cycles in milling, for which you use Cycle 14 to define the subprogram describing the finished contour. You can define the technology parameters in the respective turning cycle.

For the contour description, you use exactly the same Klartext functions as you would for defining a milling contour. FK free contour programming is included as well. Turning-specific recess and undercut contour elements are also available, which you can insert between contour elements in the same manner as chamfers and rounding arcs. Besides radial and axis recesses, E, F, H, K, and U undercuts are available, as are thread undercuts.

Depending on the cycle, the TNC 640 machines parallel to the axis or the contour. You define the machining operations (roughing, finishing) or the oversize with dialog guidance in the corresponding parameters.





# Programming in the shop

## Practical functions for grinding and dressing (option)

### Recessing

For these operations, as well, the TNC 640 features ample flexibility and functionality. Simple longitudinal and transverse recessing operations are possible, as are contour recessing operations in which the cycle is machined along any desired contour. You can work particularly effectively when performing recess turning: due to the direct alternation between infeed and cutting, air cuts are hardly necessary. Here, too, the TNC takes the technological constraints into account (width of recessing tool, overlap, feed-rate factor, etc.) and executes the operations both quickly and reliably.

In multiple plunging the solid material is repeatedly plunged into along the contour, and subsequently the remaining material is cut away. This makes it possible to safely machine materials that are difficult to cut, since there are no radial loads and chips are removed centrally.

### Simultaneous machining

The simultaneous roughing cycle and the simultaneous finishing cycle both adapt the inclination angle of the B axis to the

workpiece contour during turning. During this process, the complex tool movements are continuously monitored in order to avoid collisions between the tool / tool holder and the workpiece. In simultaneous machining, the tool is continuously engaged at the optimal inclination angle, allowing you to attain perfect surfaces while increasing tool service life.

### Thread machining

Simple and extended cycles are available for the longitudinal and transverse machining of cylindrical or tapered threads. You can use cycle parameters to define the manner in which the thread is produced. This enables you to machine a wide variety of materials.

### Blank form update

Another highlight of the TNC 640 is its blank form update feature. If you define the workpiece blank at the beginning of your program, the control computes the new blank form for each following step. The machining cycles always adapt to the current workpiece blank. The blank form update feature helps you avoid air cuts and optimize approach paths.

### Orientation of the turning tool

On milling-turning machines, it may be necessary to incline the tool during turning or change the side from which machining should be performed. With the help of a cycle, the TNC can change the inclination angle of the tool for back cutting or can convert an external turning tool into an internal turning tool without the operator needing to adjust the cutter position or the orientation angle in the tool table.

### Eccentric turning (option)

With the eccentric turning function, you can perform turning operations even if the workpiece axis is not aligned with the axis of rotation due to the setup configuration. During machining, the TNC 640 compensates for any eccentricity with compensating movements in the linear axis coupled with the rotating spindle.

The TNC 640 also lets you perform grinding operations on your machine. Cycles for jig grinding and dressing allow you to program the respective function with great convenience. In addition, the TNC 640 can superimpose a tool-axis reciprocating stroke onto the programmed movements. Optimized tool management supports the user during every process, including grinding and dressing. The TNC 640 is thus the perfect basis for attaining the highest degree of surface quality and accuracy in your application.

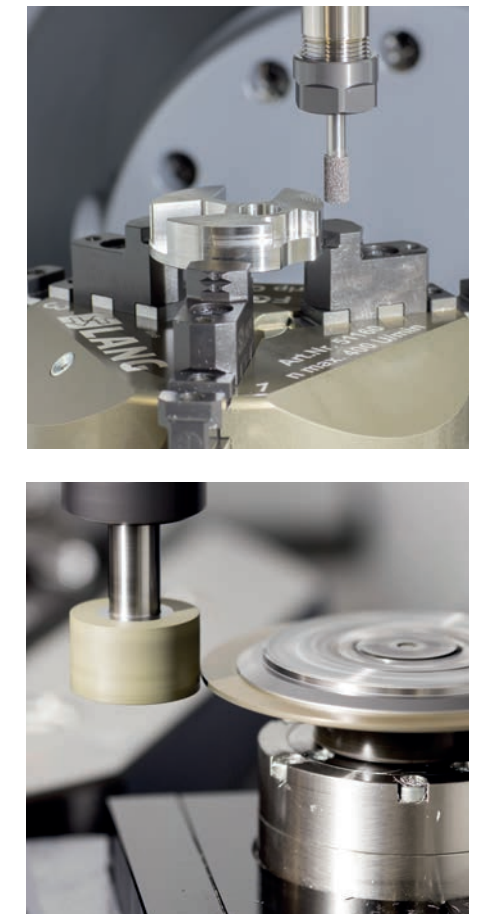
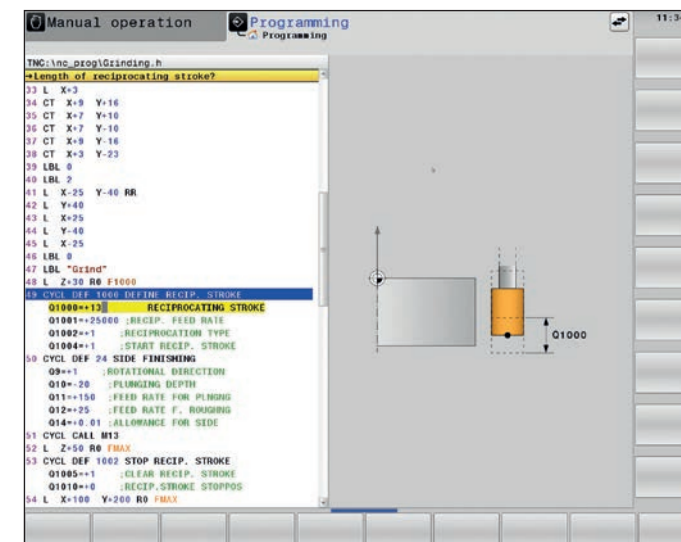
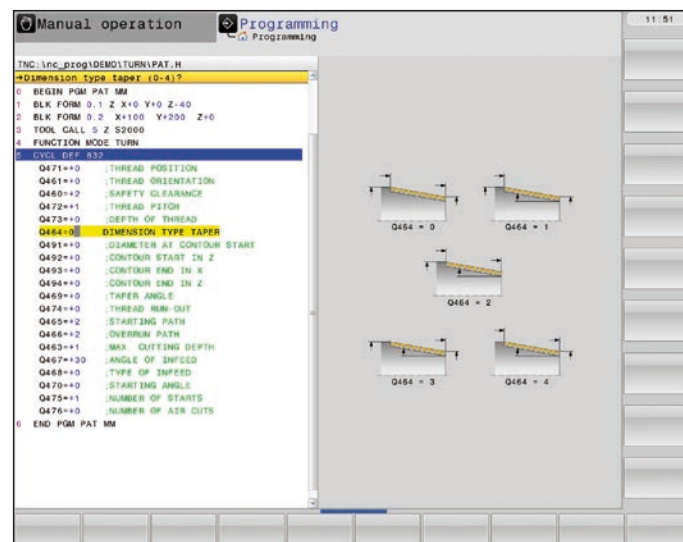
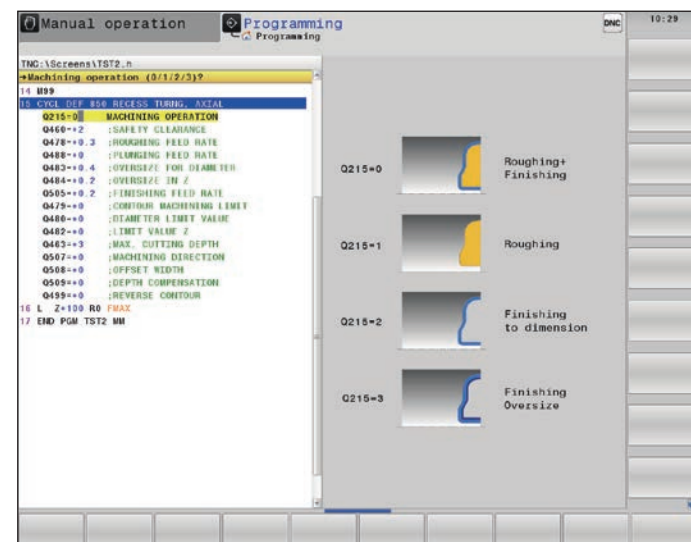
### Jig grinding\*

Jig grinding is the grinding of a 2D contour. On a milling machine, jig grinding is primarily used for finishing holes or pre-machined contours using a grinding tool. You can program open and closed grinding contours in the predefined grinding cycles and then machine them. You also have the option of superimposing a tool-axis reciprocating stroke onto the tool's motion. Special cycles allow you to define, start, and stop the reciprocating stroke. The reciprocating stroke enables uniform wear on the grinding tool and precise geometries on ground surfaces.

### Dressing\*

Dressing cycles allow you "true up" grinding tools inside the machine. During dressing, the grinding tool is machined with a special dressing tool. Klartext cycles are available for dressing the grinding tool's diameter or profile.

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



# Programming in the shop

## Reusing programmed contour elements

### Coordinate transformation

If you need a pre-programmed contour at different locations and in different sizes on the workpiece, then the TNC 640 offers you a convenient solution in the form of coordinate transformations. Depending on the operation, you can **shift the datum** (milling, grinding, and turning) and **rotate** the coordinate system (milling and grinding) as well as **mirror** the contour (milling and grinding). With a **scaling factor** (milling and grinding), you can enlarge or reduce contours to account for shrinkage or oversizes.

### Program-section repeats and subprograms

Many machining operations repeat themselves either on the same workpiece or on different workpieces. Once you have programmed a detail, there is no need to program it again. With its subprogram technology, the TNC can save you a great deal of programming time.

In **program-section repeats**, you can label a section of the program, and the TNC will repeat this section as many times as required.

You can mark a program section as a **subprogram** and then call it at any point in the program and as often as you want.

With the **program call** function, you can even use a completely separate program at any location in your current program. This allows you to leverage already programmed and frequently required working steps or contours.

Of course, you can also combine these programming techniques as often as desired.

## Rapid availability of all information

Do you have questions about a programming step but your User's Manual is not at hand? No problem: both the TNC 640 control and the TNC 640 programming station feature TNCguide, a convenient help system that displays the user documentation in a separate window.

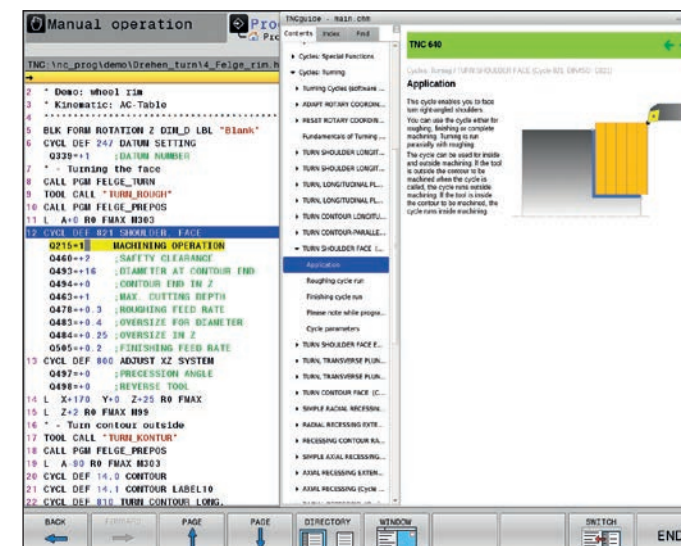
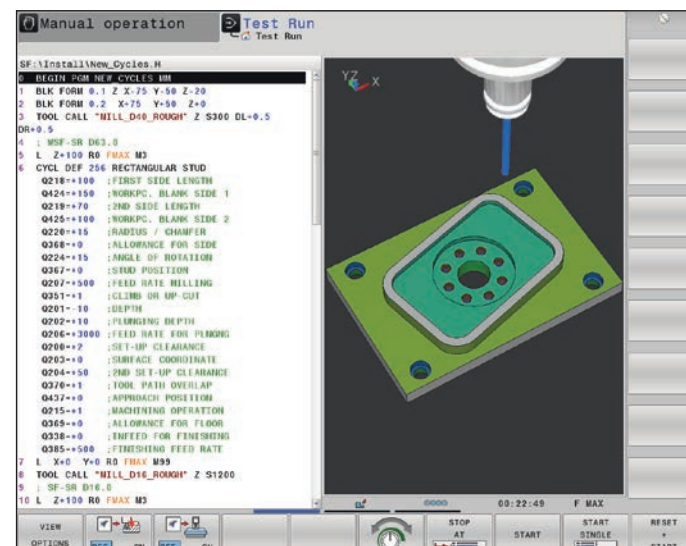
You can activate TNCguide simply by pressing the HELP key on the TNC keyboard or by clicking any soft key when the mouse pointer has switched to a question mark. This is easily done by clicking the help icon permanently displayed on the TNC screen.

TNCguide generally displays the information instantaneously and in the correct context (context-sensitive help), immediately giving you the information you currently need. This function is particularly helpful for soft keys and provides a detailed explanation of their effects.

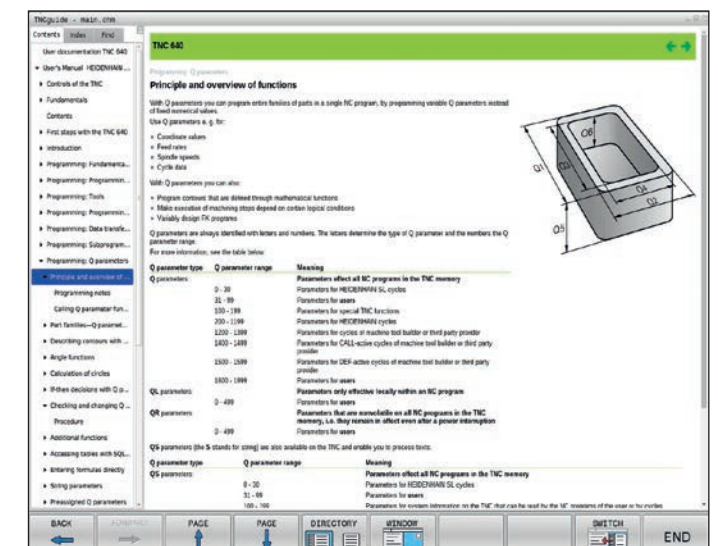
You can download the documentation in the desired language free of charge from the HEIDENHAIN website and save it to the appropriate language directory of the TNC hard disk.

The following User's Manuals are available in the help system:

- Klartext Conversational Programming
- Setup, Testing, and Running NC Programs
- Programming of Machining Cycles
- Programming of Measuring Cycles for Workpieces and Tools
- ISO Programming
- TNC 640 Programming Station (installed only with the programming station)



TNCguide integrated in the control, e.g. on the TNC 640 ...



... or on the programming station.



# Open to outside information

## Processing CAD files with the TNC 640

### CAD viewer

With the standardly available CAD viewer software, you can open 3D CAD models and drawings directly on the TNC 640. Various view options as well as rotation and zoom capabilities allow for a detailed visual inspection and analysis of your CAD data. You can also use the viewer to ascertain position values from a 3D model. You simply select an arbitrary reference point in your drawing and select the desired contour elements. The CAD viewer then displays the coordinates of the elements in a window. The CAD viewer can depict the following file formats:

- STEP files (.STP and .STEP)
- IGES files (.IGS and .IGES)
- DXF files (.DXF)

### CAD Import (option)

Why program complex contours if you already have the drawing as a DXF, STEP, or IGES file? After all, you have the ability to extract contours or machining positions from these types of CAD files. This not only saves time otherwise spent on programming and testing, but you can also be sure that the finished contour is exactly in accordance with the design engineer's specifications.

Extracting machining information directly from CAD data offers additional possibilities, particularly for the creation of NC programs with a tilted machining plane. You can also define the preset with a 3D basic rotation of the 3D model and set a datum with the appropriate 3D rotation in the desired working plane.

You can easily save the working plane to the clipboard and transfer it to the NC program with the appropriate transformation and the associated PLANE command. In the defined working plane, you can extract contours and machining positions, and apply them to the NC program.

Selecting contours is very convenient: you can select any element by clicking it with the mouse. Once you select a second element, the TNC detects your desired machining direction and initiates automatic contour detection. In doing so, the TNC automatically selects all clearly identifiable contour elements until the contour closes or branches out. This method lets you define elaborate contours with just a few clicks of the mouse. Then you can simply copy the selected contour to an existing Klartext program using the clipboard.

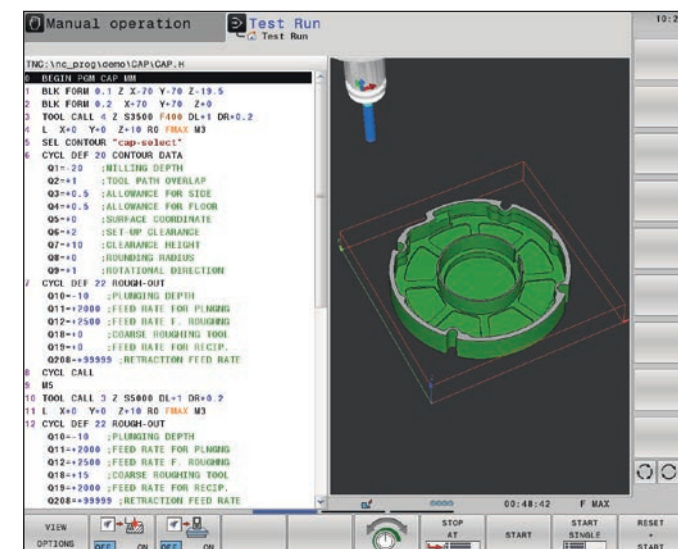
You can also select **machining positions** and save them as point files, especially in order to apply hole positions or starting points for pocket machining. This can be done very easily by selecting the desired area with the mouse. In a pop-up window with a filter function, the TNC displays all of the hole diameters within the area you have selected. To select the desired hole diameters and restrict the number of hole positions, simply click the corresponding filter symbol to change the filter limits. A zoom function and various configuration possibilities round out the functionality of CAD Import.

It is also possible to define the contour program's resolution for using it in older TNC controls. You can also define a transition tolerance if the contour elements are not completely connected.

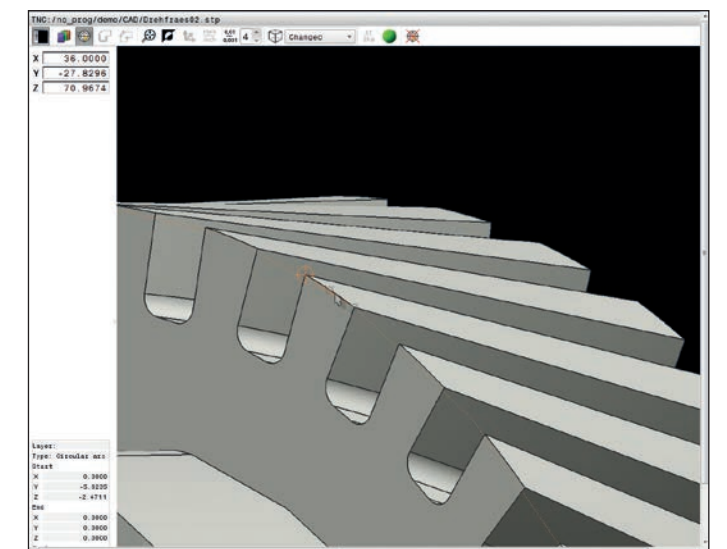
The following positions can be defined as presets:

- The beginning, end, or mid-point of a line
- The beginning, end, or center point of a circular arc
- Quadrant transitions or center point of a circle
- Intersection of two lines, regardless of their extension
- Intersection of a line and a circular arc
- Intersection of a line and a circle

If multiple intersections between two elements are possible (e.g., between a straight line and a circle), then you can select the correct intersection with a mouse click.



Part program on the basis of the imported DXF file



Display of a 3D model in the CAD viewer

# 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](http://www.heidenhain.de)  
For complete and further addresses see [www.heidenhain.com](http://www.heidenhain.com)

<b>DE</b>	<b>HEIDENHAIN Vertrieb Deutschland</b> 83301 Traunreut, Deutschland ☎ 08669 31-3132 FAX 08669 32-3132 E-Mail: hd@heidenhain.de	<b>DK</b>	Denmark → SE	<b>NZ</b>	<b>Llama ENGINEERING Ltd</b> 5012 Wellington, New Zealand E-mail: info@llamaengineering.co.nz
	<b>HEIDENHAIN Technisches Büro Nord</b> 12681 Berlin, Deutschland ☎ 030 54705-240	<b>ES</b>	<b>FARRESA ELECTRONICA S.A.</b> 08028 Barcelona, Spain www.farresa.es	<b>PH</b>	<b>MACHINEBANKS' CORPORATION</b> Quezon City, Philippines 1113 E-mail: info@machinebanks.com
	<b>HEIDENHAIN Technisches Büro Mitte</b> 07751 Jena, Deutschland ☎ 03641 4728-250	<b>FI</b>	<b>HEIDENHAIN Scandinavia AB</b> 01740 Vantaa, Finland www.heidenhain.fi	<b>PL</b>	<b>APS</b> 02-384 Warszawa, Poland www.heidenhain.pl
	<b>HEIDENHAIN Technisches Büro West</b> 44379 Dortmund, Deutschland ☎ 0231 618083-0	<b>FR</b>	<b>HEIDENHAIN FRANCE sarl</b> 92310 Sèvres, France www.heidenhain.fr	<b>PT</b>	<b>FARRESA ELECTRÓNICA, LDA.</b> 4470 - 177 Maia, Portugal www.farresa.pt
	<b>HEIDENHAIN Technisches Büro Südwest</b> 70771 Leinfelden-Echterdingen, Deutschland ☎ 0711 993395-0	<b>GB</b>	<b>HEIDENHAIN (G.B.) Limited</b> Burgess Hill RH15 9RD, United Kingdom www.heidenhain.co.uk	<b>RO</b>	<b>HEIDENHAIN Reprezentantă Romania</b> Braşov, 500407, Romania www.heidenhain.ro
	<b>HEIDENHAIN Technisches Büro Südost</b> 83301 Traunreut, Deutschland ☎ 08669 31-1337	<b>GR</b>	<b>MB Milionis Vassilis</b> 17341 Athens, Greece www.heidenhain.gr	<b>RS</b>	Serbia → BG
		<b>HR</b>	Croatia → SL	<b>RU</b>	<b>GERTNER Service</b> 119002 Moscow, Russian Federation www.heidenhain.ru
		<b>HU</b>	<b>HEIDENHAIN Kereskedelmi Képviselet</b> 1239 Budapest, Hungary www.heidenhain.hu	<b>SE</b>	<b>HEIDENHAIN Scandinavia AB</b> 12739 Skärholmen, Sweden www.heidenhain.se
<b>AR</b>	<b>NAKASE SRL.</b> B1653AOX Villa Ballester, Argentina www.heidenhain.com.ar	<b>ID</b>	<b>PT Servitama Era Toolsindo</b> Jakarta 13930, Indonesia E-mail: ptset@group.gts.co.id	<b>SG</b>	<b>HEIDENHAIN PACIFIC PTE LTD</b> Singapore 408593 www.heidenhain.com.sg
<b>AT</b>	<b>HEIDENHAIN Techn. Büro Österreich</b> 83301 Traunreut, Germany www.heidenhain.de	<b>IL</b>	<b>NEUMO VARGUS MARKETING LTD.</b> Holon, 5885948, Israel E-mail: neumo@neumo-vargus.co.il	<b>SK</b>	<b>KOPRETINA TN s.r.o.</b> 91101 Trenčin, Slovakia www.kopretina.sk
<b>AU</b>	<b>FCR MOTION TECHNOLOGY PTY LTD</b> Ravenhall Victoria 3023, Australia E-mail: sales@fcrmotion.com	<b>IN</b>	<b>HEIDENHAIN Optics &amp; Electronics India Private Limited</b> Chetpet, Chennai 600 031, India www.heidenhain.in	<b>SL</b>	<b>NAVO d.o.o.</b> 2000 Maribor, Slovenia www.heidenhain.si
<b>BE</b>	<b>HEIDENHAIN NV</b> 1760 Roosdaal, Belgium www.heidenhain.be	<b>IT</b>	<b>HEIDENHAIN ITALIANA S.r.l.</b> 20128 Milano, Italy www.heidenhain.it	<b>TH</b>	<b>HEIDENHAIN (THAILAND) LTD</b> Bangkok 10250, Thailand www.heidenhain.co.th
<b>BG</b>	<b>ESD Bulgaria Ltd.</b> Sofia 1172, Bulgaria www.esd.bg	<b>JP</b>	<b>HEIDENHAIN K.K.</b> Tokyo 102-0083, Japan www.heidenhain.co.jp	<b>TR</b>	<b>T&amp;M Mühendislik San. ve Tic. LTD. ŞTİ.</b> 34775 Y. Dudullu – Umraniye-Istanbul, Turkey www.heidenhain.com.tr
<b>BR</b>	<b>HEIDENHAIN Brasil Ltda.</b> 04763-070 – São Paulo – SP, Brazil www.heidenhain.com.br	<b>KR</b>	<b>HEIDENHAIN Korea Ltd.</b> Anyang-si, Gyeonggi-do, 14087 South Korea www.heidenhain.co.kr	<b>TW</b>	<b>HEIDENHAIN CO., LTD.</b> Taichung 40768, Taiwan www.heidenhain.com.tw
<b>BY</b>	<b>GERTNER Service</b> 220026 Minsk, Belarus www.heidenhain.by	<b>MX</b>	<b>HEIDENHAIN CORPORATION MEXICO</b> 20290 Aguascalientes, AGS., Mexico E-mail: info@heidenhain.com	<b>UA</b>	<b>GERTNER Service</b> 02094 Kiev, Ukraine www.heidenhain.ua
<b>CA</b>	<b>HEIDENHAIN CORPORATION</b> Mississauga, Ontario L5T2N2, Canada www.heidenhain.com	<b>MY</b>	<b>ISOSERVE SDN. BHD.</b> 43200 Balakong, Selangor E-mail: sales@isoserve.com.my	<b>US</b>	<b>HEIDENHAIN CORPORATION</b> Schaumburg, IL 60173-5337, USA www.heidenhain.us
<b>CH</b>	<b>HEIDENHAIN (SCHWEIZ) AG</b> 8603 Schwerzenbach, Switzerland www.heidenhain.ch	<b>NL</b>	<b>HEIDENHAIN NEDERLAND B.V.</b> 6716 BM Ede, Netherlands www.heidenhain.nl	<b>VN</b>	<b>AMS Co. Ltd</b> HCM City, Vietnam E-mail: davidgoh@amsvn.com
<b>CN</b>	<b>DR. JOHANNES HEIDENHAIN (CHINA) Co., Ltd.</b> Beijing 101312, China www.heidenhain.com.cn	<b>NO</b>	<b>HEIDENHAIN Scandinavia AB</b> 7300 Orkanger, Norway www.heidenhain.no	<b>ZA</b>	<b>MAFEMA SALES SERVICES C.C.</b> Kyalami 1684, South Africa www.heidenhain.co.za
<b>CZ</b>	<b>HEIDENHAIN s.r.o.</b> 102 00 Praha 10, Czech Republic www.heidenhain.cz				



# Open to outside information

## Uniformly digital job management with Connected Machining



The smooth exchange of knowledge is crucial for the success of a company. In order for knowledge to be transferred rapidly and without loss, e-mail communication capability is just as important as the continuous availability of electronic production documents or the transfer of data to ERP and production-activity control systems. Tool and raw material inventories, tool data, workholding drawings, CAD data, NC programs, and inspection instructions must all be available to machine operators on all shifts. Cost-effective manufacturing therefore requires an efficient process chain and a networked control.

The TNC 640, with its **Connected Machining** package of functions, integrates itself flexibly into your process chain and helps you to optimize the transfer of knowledge within your company. So let your workshop, as well,

benefit from all of the information available within your company. **Connected Machining** makes uniformly digital job management possible within a networked manufacturing environment. You thereby benefit from the following:

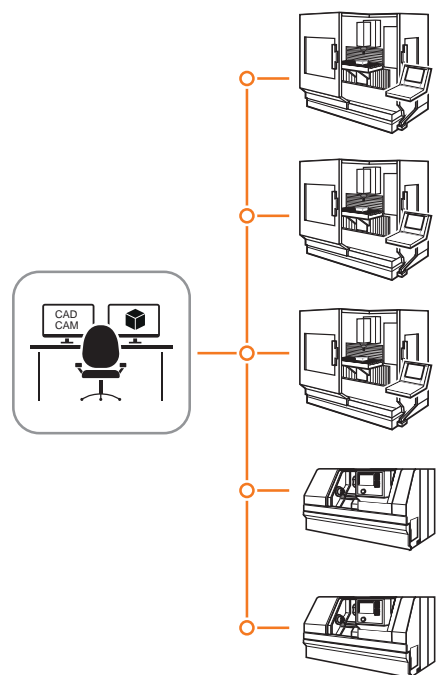
- Ease of data usage
- Time-saving procedures
- Transparent processes

### The networked TNC 640

Integrate the TNC 640 with the functions of **Connected Machining** into your corporate network and, via the control, connect the shop floor to PCs, programming stations, and other data storage media in these areas of the company:

- Design
- Programming
- Simulation
- Production planning
- Production

Even the basic version of the TNC 640 is equipped with two Gigabit Ethernet ports of the latest generation. The TNC 640 communicates with NFS servers and Windows networks in the TCP/IP protocol without additional software. The fast data transfer of up to 1000 Mbit/s ensures very short transmission times. The TNC 640 therefore offers the optimal technological platform for **Connected Machining**, the networking of the control in the shop with all production-related areas of your company.



### Standard range of functions

In order for you to be able to use the data that you have transferred to the TNC 640 via the standard network connection, the TNC 640 offers attractive applications, even in its standard range of functions. The PDF viewer or Mozilla Firefox web browser enable the simplest form of **Connected Machining**: access to production process data directly on the control. As a result, operating web-based documentation and ERP systems is just as easy as accessing your e-mail account. The following other file formats can also be opened directly on the TNC 640:

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

### Data transfer with Connected Machining

Another solution for full digital job management from **Connected Machining** is the free-of-charge **TNCremo PC** software. With it, you can

- bidirectionally transfer remotely stored part programs and tool or pallet tables, including over the Ethernet, and
- start the machine.

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

### Provision of operating states

Option 137, State Reporting Interface (SRI), gives you a simple and reliable interface for recording the operating data of your machine. It records the idle times, run times, and fault messages of your machine. SRI also provides historical operating data that can still be retrieved even after the corporate network has been down for hours at a time.

### Job-related data

With **REMOTE DESKTOP MANAGER (option 133)**, you can operate a Windows PC from the TNC 640. On the control, you gain direct access to EDP systems within the process chain and make your machine setup processes significantly more efficient by reducing inconvenient legwork between the machine and the office. Technical drawings, CAD data, NC programs, tool data, work instructions, parts lists, and warehouse information are digitally available at the machine. E-mails can be sent and received with ease. With a simple keystroke on the machine operating panel, you can switch between the control screen and the screen of the Windows PC. The PC can be a computer in the local network or an industrial PC (IPC) in the electrical cabinet of the machine.

### HEIDENHAIN OPC UA NC Server

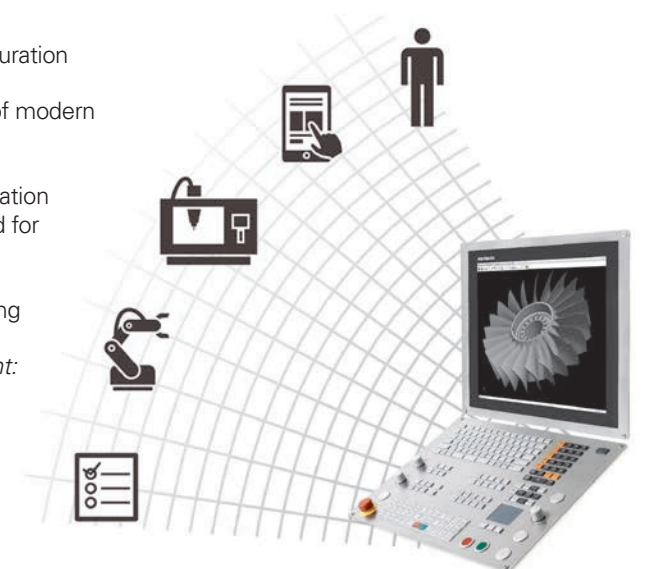
With OPC UA NC Server, HEIDENHAIN offers a reliable interface for the integration of modern industrial applications. The standardized protocol and the application-oriented information model enable seamless and economical integration of the TNC control into your production environment.

- **State-of-the-art IT security:** Authentication, authorization, and cryptography
- **Uncomplicated:** Guided connection configuration
- **Application-oriented:** Tailored to the demands of modern industrial applications
- **Standardized:** OPC UA is the communication technology recommended for Industry 4.0
- **Freedom of choice:** Open selection of operating system and toolkit
- **Virtual testing environment:** Free HEIDENHAIN programming station



### Detailed data for the optimal organization of production processes

HEIDENHAIN DNC enables, among other things, the connection of TNC controls to inventory management systems and production-activity control systems. Automatic feedback messages about active production processes, for example, can be set up over this interface.



# Open to outside information

## StateMonitor: collect and evaluate machine data

StateMonitor collects and visualizes the statuses of production machines. By evaluating important data such as the current machine status, machine messages, override positions, and utilization history, StateMonitor provides in-depth information about the machine's degree of utilization. Based on the data it collects, StateMonitor also reveals machine-specific optimization potential. By entering comments regarding machine downtimes and setup times, the operator can also uncover potential for improvement at the organizational level. Via the Messenger function, StateMonitor notifies the right person by e-mail based on individually combinable machine signals and statuses.

StateMonitor collects and visualizes the following information from the networked machines:

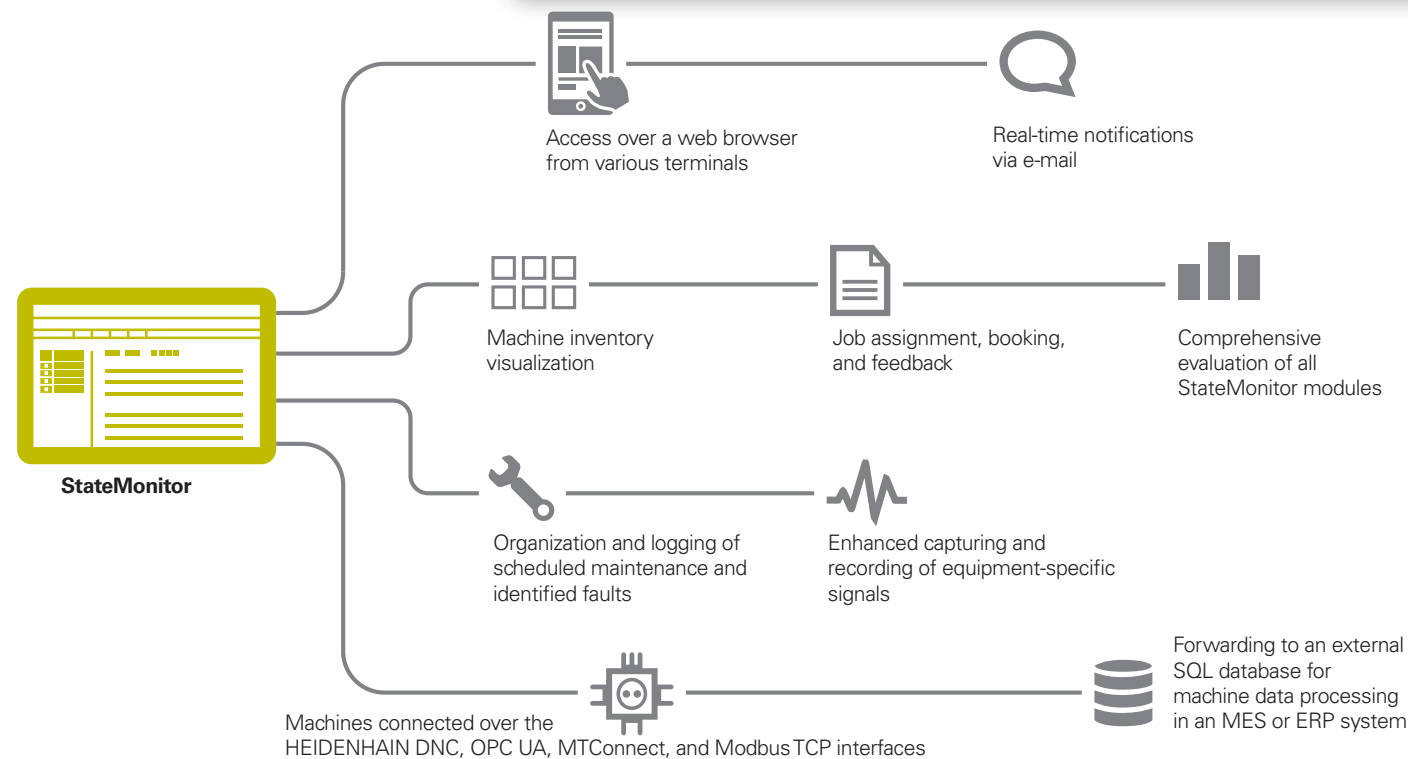
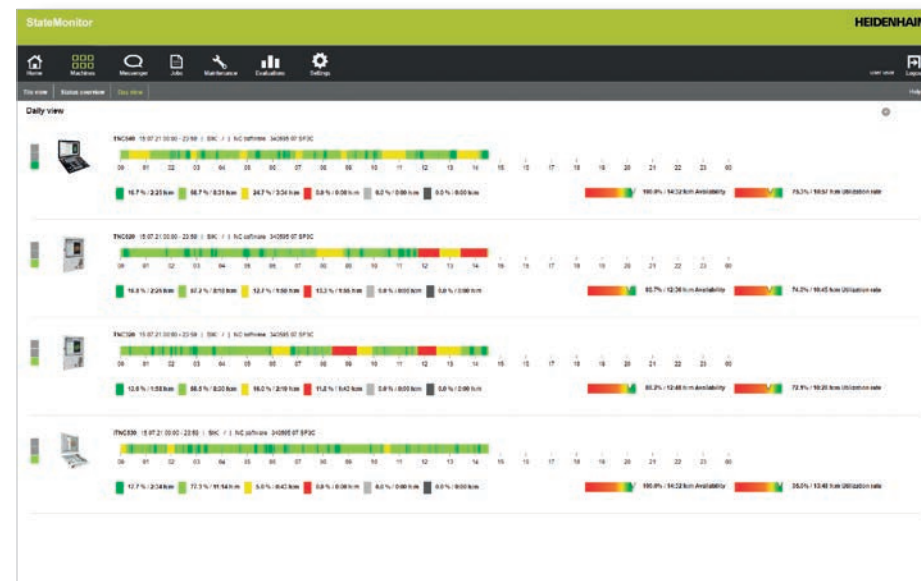
- Operating modes
- Override positions (spindle, rapid traverse, feed rate)
- Program status and program name, as well as subprograms if applicable
- Program run time
- SIK number and software number
- Machine messages

Active support for production planning with an extensive range of functions for job data collection:

- Create and assign jobs
- Start and terminate jobs
- Plan setup times and interruptions
- Store additional job data, such as quantities produced

StateMonitor even lets you connect machines with different controls and supports the following protocol types: HEIDENHAIN DNC, OPC UA, MTConnect, and Modbus TCP.

For more information, please contact HEIDENHAIN.



## The TNC 640 programming station

### What's the purpose of a programming station?

As easy as it is to write a part program on a TNC 640 at the machine while a different part is being machined, short reloading times or high machine utilization may hinder attentive programming work on the shop floor. The TNC 640 programming station lets you program the same way you would at the machine but away from all the noise and distractions.

### Creating programs

The programming, testing, and optimizing of HEIDENHAIN Klartext or ISO programs with the programming station substantially reduces machine idle times. And it's easy to get used to because every keystroke feels the same: the programming station's keyboard is just like the one on the machine.

### Testing programs created offline

Naturally, you can also test programs that were written on a CAD/CAM system. The various views of the program verification graphics help you easily spot contour damage and hidden details.

### Training with the programming station

Since the TNC 640 programming station is based on the same software as the TNC 640 control, it is ideal for apprentice and advanced-level training. Programming is performed on the original keyboard unit, and even the test run feature runs exactly as it would on the machine. This gives the trainee the experience needed to safely work on the real machine.

The TNC 640 programming station is also optimally suited for teaching TNC programming skills in schools, since the TNC 640 can be programmed in Klartext and with ISO codes.

### Your workstation

The programming station software runs on a PC. The PC screen shows you the same TNC user interface as the control and offers the familiar graphical support. Depending on the version of the programming station, there are multiple ways of using it.

The free **demo version** contains all of the TNC 640's function and allows short programs to be saved. Programming is performed on the PC keyboard. The version with a **TNC operating panel** lets you create programs in the accustomed manner on a keyboard featuring the same function keys as the control on the machine. It also has a PC keyboard for ISO programming, file names, and comments.

You can also work without the TNC operating panel: a **virtual keyboard** for operating the programming station is shown on the PC screen. This virtual keyboard provides the TNC 640's most important dialog-opening keys.





## Workpiece measurement

Setup, preset setting, and measuring with touch trigger probes

Workpiece touch probes\* from HEIDENHAIN help you keep costs down in the shop and in serial production: when used in conjunction with the TNC 640, they enable the automated execution of setup, measurement, and inspection functions.

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

The touch probes are inserted directly into the machine tool spindle and can be equipped with various taper shanks depending on the machine. The ball tips, which are made of ruby, are available in different diameters with various stylus lengths.

\* The machine and TNC must be specially adapted by the machine manufacturer.



TS 460

### Cable-bound touch probes

For machines requiring manual tool changes, as well as for grinding machines and lathes:

#### TS 150/TS 750

- Axial or radial cable connection on its base
- High probing accuracy and repeatability
- Compact dimensions

#### TS 260

- Axial or radial cable connection
- High scanning accuracy

### Wireless touch probes

For machines with an automatic tool changer:

#### TS 460

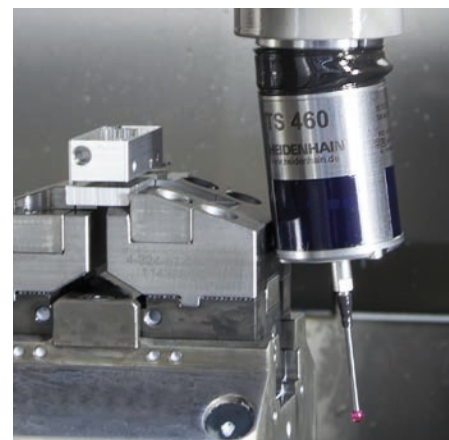
- Standard touch probe for radio and infrared transmission
- Compact dimensions
- Energy-saving mode
- Optional collision protection
- Thermal decoupling

#### TS 642

- Activation via a switch in the taper shank
- Infrared transmission

#### TS 740

- High scanning accuracy
- High reproducibility
- Low probing forces
- Infrared transmission



TS 460 with collision protection

### Transceiver unit

Radio or infrared transmission is established between the TS or TT touch probe and the SE transceiver:

#### SE 660

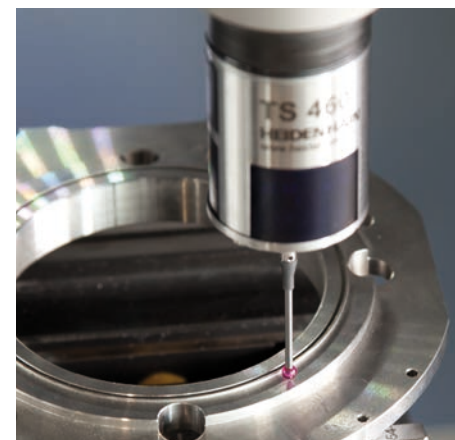
- For radio or infrared transmission (hybrid technology)
- Same SE for TS 460 and TT 460

#### SE 661

- For radio or infrared transmission (hybrid technology)
- Same SE for TS 460 and TT 460
- EnDat functionality for transmission of the switching state, diagnostics, and additional information



SE 660



## Tool measurement

Measuring length, radius, and wear inside the machine

A key factor in obtaining consistently high production quality is, of course, the tool. This requires exact measurement of the tool dimensions and periodic inspection of the tool for breakage, wear, and tooth shape. For tool measurement, HEIDENHAIN offers the TT triggering tool touch probes.

These touch probes are installed directly within the machine's workspace, where they enable tool measurement either before machining or during interruptions.

The **TT tool touch probes** measure the tool length and radius. During probing of the rotating or stationary tool, such as for individual tooth measurement, the contact plate is deflected and a trigger signal is transmitted directly to the TNC 640.

The **TT 160** uses cable-bound signal transmission, whereas the **TT 460** sends its signals via radio or infrared transmission. It is thus particularly well suited for use on rotary/tilting tables.

To prevent limiting the machining space and avoid collisions, the TT touch probe must occasionally be removed from the machine. The touch probe's new magnetic base features three contact points and a setting screw. As a result, the touch probe requires calibration only during initial installation and can then be re-installed and removed with a quick hand movement.

Here are the benefits in a nutshell:

- Fast re-installation without recalibration
- Low-profile base
- Same accuracy as a permanent installation



TT 460



### Further information:

You can find detailed descriptions of workpiece touch probes on the Internet at [www.heidenhain.com](http://www.heidenhain.com) or in the *Touch Probes for Machine Tools* brochure.

### Further information:

Detailed descriptions of tool touch probes are available on the Internet at [www.heidenhain.com](http://www.heidenhain.com) or in the *Touch Probes for Machine Tools* brochure.

## Inspecting and optimizing machine accuracy

### Easy calibration of rotary axes with KinematicsOpt (option)

Accuracy demands are on the rise, particularly in the realm of 5-axis machining. Complex parts must be manufactured with both precision and reproducible accuracy, including over extended periods of time.

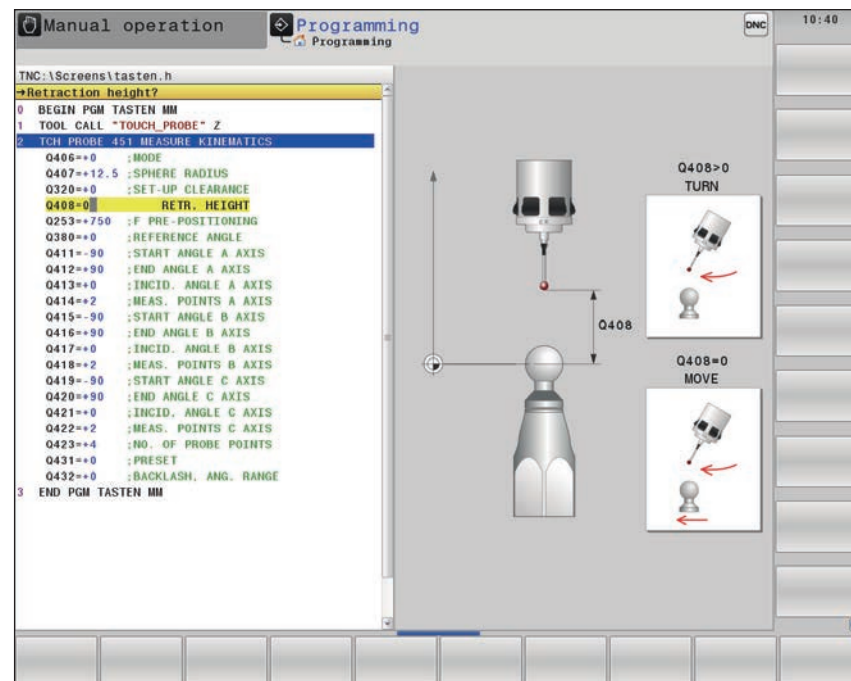
The TNC function **KinematicsOpt** is an important ingredient in helping you meet these high requirements: with a HEIDENHAIN touch probe inserted, a cycle measures your machine's rotary axes fully automatically. The measuring process is the same, regardless of whether the rotary axis is a rotary table, tilting table, or a swivel head.

For calibrating the rotary axes, a calibration sphere is mounted at any position on the machine table and probed with the HEIDENHAIN touch probe. Prior to this, you define the resolution of the measurement and the range that you want to measure for each rotary axis.

Based on the measured values, the TNC calculates the static tilting accuracy. The software minimizes the spatial error arising from the tilting movements and, at the end of the measuring process, automatically saves the machine geometry in the respective machine constants of the kinematics description.

Of course, a detailed log file is also available in which the measured and optimized dispersion (measure of the static tilting accuracy) is saved along with the actual measured values and the actual compensation values.

Optimal use of KinematicsOpt requires a highly rigid calibration sphere for reducing the deformations caused by probing forces. For this purpose, HEIDENHAIN offers calibration spheres with highly rigid holders of various lengths.



## Positioning with an electronic handwheel

### Fine motion control of axes

You can set up the workpiece by manually jogging the axes with the axis direction keys, but this task is easier and more finely controllable with the electronic handwheels from HEIDENHAIN.

In this method, the axis slides are moved by the feed motor in accordance with the handwheel's rotation. For particularly high sensitivity, you can incrementally set the amount of traversing distance for each handwheel rotation.

**HR 130 panel-mounted handwheels**  
The HR 130 panel-mounted handwheel from HEIDENHAIN can be integrated into the machine operating panel or mounted to a different part of the machine.

**HR 510, HR 520, and HR 550 portable handwheels**  
The HR 510, HR 520, and HR 550 portable handwheels are particularly helpful when you work near the machine's working space. The axis keys and certain function keys are built into the housing. In this way, you can switch axes and set up the machine at any time—regardless of where you happen to be standing. As a wireless handwheel, the HR 550 is ideal for use on large machines. If you no longer need the handwheel, simply attach it to the machine using its built-in magnets.

- Expanded range of functions of the HR 520 and HR 550**
- Definable traversing distance per revolution
  - Display for operating mode, actual position value, programmed feed rate, handwheel offset, spindle speed, and error messages
  - Override potentiometers for feed rate, handwheel offset, and spindle speed
  - Selection of axes by means of keys and soft keys
  - Keys for continuous traverse of the axes
  - Emergency stop button
  - Actual position capture
  - NC start/stop
  - Spindle on/off
  - Soft keys for machine functions defined by the machine manufacturer



HR 550





# Overview

## User functions

User function	Standard	Option	
<b>Short description</b>	✓	0-7 77 78	Basic version: 3 axes plus closed-loop spindle } In total: 14 additional NC axes or 13 additional NC axes plus second spindle
	✓		Digital current and speed control
<b>Program entry</b>	✓ ✓	42	HEIDENHAIN Klartext ISO programming Importing of contours or machining positions from CAD files (STP, IGS, DXF) and saving them as a Klartext contour program or Klartext point table
<b>Position entry</b>	✓ ✓ ✓		Nominal positions for lines and arcs in Cartesian coordinates or polar coordinates Incremental or absolute dimensions Display and entry in mm or inches
<b>Tool compensation</b>	✓ ✓	9	Tool radius in the working plane, and tool length Radius-compensated contour look-ahead for up to 99 blocks (M120) Three-dimensional tool-radius compensation for changing tool data without having to recalculate an existing program
<b>Tool tables</b>	✓		Multiple tool tables with any number of tools
<b>Cutting data</b>	✓	167	Automatic calculation of spindle speed, cutting speed, feed per tooth, and feed per revolution OCM: automatic calculation of overlap factor, milling feed rate, spindle speed, climb or up-cut milling, lateral infeed, cutting speed, material removal rate, recommended cooling
<b>Constant contour speed</b>	✓ ✓		Relative to the path of the tool center point Based on the tool's cutting edge
<b>Parallel operation</b>	✓		Program creation with graphical support while another program is running
<b>3D machining</b>	✓	9 9 9 9 9 92	Motion control with highly smoothed jerk 3D tool compensation via surface-normal vectors Changing the swivel-head angle with the electronic handwheel during program run without affecting the tool tip position (TCPM = Tool Center Point Management) Keeping the tool perpendicular to the contour Tool radius compensation perpendicular to the tool direction Manual traverse in the active tool-axis system 3D radius compensation based on the tool's contact angle
<b>Rotary table machining</b>		8 8	Programming of contours for machining on a cylinder as if in two axes Feed rate in mm/min
<b>Fixture monitoring</b>		40	The DCM Collision function (option 40) allows you to integrate fixtures, such as a vise, in your NC program (using CFG or STL files). The control displays the fixtures in the simulation and monitors them for collisions

User function	Standard	Option	
<b>Turning</b>		50 50 50 50 50 50 50 50 50 50 158	Program-controlled switchover between milling and turning Constant cutting speed Cutter radius compensation Cycles for roughing, finishing, recessing, thread turning, and recess turning Blank-form updating in contour cycles Turning-specific contour elements for recesses and undercuts Orientation of the turning tool for external turning / internal turning Inclined turning Speed limiting Eccentric turning (additionally required: option 135) Simultaneous turn finishing
<b>Contour elements</b>	✓ ✓ ✓ ✓ ✓ ✓ ✓	50 50	Straight line Chamfer Circular path Circle center Circle radius Tangentially connecting circular arc Corner rounding Recess Undercut
<b>Contour approach and departure</b>	✓ ✓		Via straight line: tangential or perpendicular Via circular arc
<b>Adaptive Feed Control</b>		45	AFC adapts the contouring feed rate to the current spindle power
<b>Collision monitoring</b>		40 40 40	Dynamic Collision Monitoring (DCM) Graphic depiction of the active collision objects Tool carrier monitoring
<b>Free contour programming</b>	✓		FK free contour programming in HEIDENHAIN Klartext format with graphical support for workpiece drawings not dimensioned for NC
<b>Program jumps</b>	✓ ✓ ✓		Subprograms Program-section repeat Any program as a subprogram

# Overview

## User functions

User function	Standard	Option	
<b>Fixed cycles</b>	✓		Drilling, tapping (with or without floating tap holder), rectangular and circular pockets
	✓		Peck drilling, reaming, boring, counterboring, centering
		50	Area clearance cycles, longitudinal and transverse, paraxial and contour-parallel
		50	Recessing cycles, radial/axial
		50	Radial/axial recess turning cycles (combined recessing and roughing motion)
	✓		Milling of internal and external threads
		50	Turning of internal and external threads
		50+	Hobbing
		131	
		96	Interpolation turning
		50+	Simultaneous turning (roughing and finishing) for turning operations
		158	
		156	Function for grinding operation
		157	Gear-system production
	✓		Clearing of level and oblique surfaces
	✓		Complete machining of rectangular and circular pockets, rectangular and circular studs
	✓		Complete machining of straight and circular slots
✓		Point patterns on a circle and lines	
✓		Point pattern: Data Matrix code	
✓		Contour train, contour pocket	
✓		Contour slot with trochoidal milling	
✓		Engraving cycle: engrave text or numbers in a straight line or on an arc	
✓		Integratability of OEM cycles (special cycles developed by the machine manufacturer)	
	167	Optimized Contour Milling (OCM) cycles: optimization of roughing processes	
<b>Coordinate transformation</b>	✓		Shifting, rotating, mirroring, scaling (axis-specific)
		8 44	Tilting the working plane, PLANE function Global Program Settings make it possible to manually define shifts, rotations, and handwheel superimpositioning
<b>Q parameters</b> Programming with variables	✓		Mathematical functions =, +, -, *, /, sin α, cos α, tan α, arc sin, arc cos, arc tan, a <sup>n</sup> , e <sup>n</sup> , ln, log, √a, √a <sup>2</sup> + b <sup>2</sup>
	✓		Logical operations (=, = /, <, >)
	✓		Calculating with parentheses
	✓		Absolute value of a number, constant π, negation, truncation of digits before or after the decimal point
	✓		Functions for calculation of circles Functions for text processing
<b>Programming aids</b>	✓		Calculator
	✓		Complete list of all current error messages
	✓		Context-sensitive help function for error messages
	✓		TNCguide: the integrated help system; user information available directly on the TNC 640
	✓		Graphical support for the programming of cycles
	✓		Comment and structure blocks in the NC program
<b>Teach-in</b>	✓		Application of actual positions directly in the NC program
<b>Program verification graphics</b> Display modes	✓		Graphical simulation of the machining operation, even while another program is being executed
	✓		Plan view / projection in three planes / 3D view, also in tilted working plane / 3D line graphics
	✓		Detail zoom
<b>Programming graphics</b>	✓		In the Programming and Editing mode, the contours of entered NC blocks are rendered (2D pencil-trace graphics), even while another program is running
<b>Program-run graphics</b> Display modes	✓		Real-time graphical simulation during execution of the milling program
	✓		Plan view / view in three planes / 3D view

## User functions and accessories

User function	Standard	Option	
<b>Machining time</b>	✓		Calculation of machining time in the Test Run operating mode
	✓		Display of the current machining time in the Program Run operating modes
<b>Returning to the contour</b>	✓		Mid-program startup at any block in the program and movement to the calculated nominal position for continuation of the machining operation
	✓		Program interruption, contour departure, and return
<b>Preset management</b>	✓		One table for saving any reference points (presets)
<b>Datum tables</b>	✓		Multiple datum tables for storing workpiece-specific datums
<b>Pallet tables</b>	✓		Workpiece-oriented execution of pallet tables (with any number of entries for the selection of pallets, NC programs, and datums)
		154	Production process planning with Batch Process Manager
<b>Touch probe cycles</b>	✓		Touch probe calibration
	✓		Manual or automatic compensation of workpiece misalignment
	✓		Manual or automatic preset setting
	✓	48	Automatic tool and workpiece measurement KinematicsOpt: automatic measurement and optimization of the machine's kinematic model
<b>Parallel secondary axes</b>	✓		Compensation of movement in the secondary axes U, V, W through the principal axes X, Y, Z
	✓		Movements of parallel axes included in the position display of the associated principal axis (sum display)
	✓		Defining the principal and secondary axes in the NC program enables execution on different machine configurations
<b>Conversational languages</b>	✓		English, German, Czech, French, Italian, Spanish, Portuguese, Swedish, Danish, Finnish, Dutch, Polish, Hungarian, Russian (Cyrillic), Chinese (traditional, simplified), Slovenian, Slovak, Norwegian, Korean, Turkish, Romanian
<b>CAD viewer</b>	✓		Display of standardized CAD file formats on the TNC

### Accessories

<b>Electronic handwheels</b>	<ul style="list-style-type: none"> <li>• <b>HR 510/HR 520:</b> portable handwheels</li> <li>• <b>HR 550:</b> portable wireless handwheel</li> <li>• <b>HR 130:</b> panel-mounted handwheel</li> </ul>
<b>Workpiece measurement</b>	<ul style="list-style-type: none"> <li>• <b>TS 150/TS 750:</b> compact workpiece touch probes with cable connection</li> <li>• <b>TS 260:</b> workpiece touch probe with cable connection</li> <li>• <b>TS 248:</b> workpiece touch probe with reduced deflection force</li> <li>• <b>TS 460:</b> workpiece touch probe with radio or infrared transmission</li> <li>• <b>TS 642/TS 740:</b> workpiece touch probes with infrared transmission</li> </ul>
<b>Tool measurement</b>	<ul style="list-style-type: none"> <li>• <b>TT 160:</b> triggering tool touch probe</li> <li>• <b>TT 460:</b> triggering tool touch probe with radio or infrared transmission</li> </ul>
<b>Programming station</b>	<ul style="list-style-type: none"> <li>Control software for PCs for programming, archiving, and training</li> <li>• Single-station license with original control keyboard/with operation via virtual keyboard</li> <li>• Network license with operation via virtual keyboard</li> <li>• Demo version (operated with PC keyboard—free of charge)</li> </ul>
<b>Software for PCs</b>	<ul style="list-style-type: none"> <li>• <b>TeleService:</b> software for remote diagnostics, monitoring, and operation</li> <li>• <b>CycleDesign:</b> software for creating your own cycle structure</li> <li>• <b>TNCremo:</b> software for data transfer—free of charge</li> <li>• <b>State Monitor:</b> software for recording, evaluating, and visualizing machine data</li> </ul>



# Overview

## Options

Option number	Option	With NC software 34059x- or later	ID	Comment
<b>0</b> <b>1</b> <b>2</b> <b>3</b> <b>4</b> <b>5</b> <b>6</b> <b>7</b>	Additional Axis 1 Additional Axis 2 Additional Axis 3 Additional Axis 4 Additional Axis 5 Additional Axis 6 Additional Axis 7 Additional Axis 8	01	354540-01 353904-01 353905-01 367867-01 367858-01 370291-01 370292-01 370293-01	One to eight additional control loops
<b>8</b>	Advanced Function Set 1	01	617920-01	<b>Rotary table machining</b> <ul style="list-style-type: none"> <li>• Programming of contours for machining on a cylinder as if in two axes</li> <li>• Feed rate in mm/min</li> </ul> <b>Interpolation:</b> circular in 3 axes with tilted working plane <b>Coordinate transformation:</b> tilting the working plane, PLANE function
<b>9</b>	Advanced Function Set 2	01	617921-01	<b>Interpolation:</b> straight line in more than four axes (export license required) <b>3D machining</b> <ul style="list-style-type: none"> <li>• 3D tool compensation via surface normal vectors</li> <li>• Alteration of the swivel head angle via the electronic handwheel during program run without changing the position of the tool center point (TCPM = Tool Center Point Management)</li> <li>• Keeping the tool perpendicular to the contour</li> <li>• Tool radius compensation perpendicular to the tool direction</li> <li>• Manual traverse in the active tool-axis system</li> </ul>
<b>18</b>	HEIDENHAIN DNC	01	526451-01	Communication with external PC applications over COM component
<b>40</b>	DCM Collision	02	526452-01	Dynamic Collision Monitoring (DCM)
<b>42</b>	CAD Import	08	526450-01	Import of contours from 2D and 3D models (e.g., STEP, IGES, DXF)
<b>44</b>	Global PGM Settings	08	576057-01	Global program settings
<b>45</b>	Adaptive Feed Control (AFC)	02	579648-01	Adaptive feed control
<b>46</b>	Python OEM Process	01	579650-01	Python application on the TNC
<b>48</b>	KinematicsOpt	01	630916-01	Touch-probe cycles for the automatic measurement of rotary axes
<b>49</b>	Double-Speed Axes	01	632223-01	Short control-loop cycle times for a direct drive motor
<b>50</b>	Turning	01	634608-01	Turning functions: <ul style="list-style-type: none"> <li>• Tool management for turning</li> <li>• Tool radius compensation</li> <li>• Switching between milling and turning mode</li> <li>• Turning-specific contour elements</li> <li>• Package of turning cycles</li> <li>• Hobbing</li> </ul>
<b>52</b>	KinematicsComp	05	661879-01	Compensation of spatial errors for rotary and linear axes (export license required)

Option number	Option	With NC software 34059x- or later	ID	Comment
<b>56–61</b>	HEIDENHAIN OPC UA NC Server	10	1291434-01 to -06	Reliable and stable interface for the connection of state-of-the-art industry applications; uncomplicated thanks to the use of standardized designs. Each of the six SIK options enables an incoming OPC UA connection via an application certificate
<b>77</b>	4 Additional Axes	01	634613-01	4 additional control loops
<b>78</b>	8 Additional Axes	01	634614-01	8 additional control loops
<b>92</b>	3D-Tool Comp	07	679678-01	3D radius compensation based on the contact angle (only with the Advanced Function Set 2 software option)
<b>93</b>	Extended Tool Management	01	676938-01	Extended tool management
<b>96</b>	Advanced Spindle Interpolation	05	751653-01	Cycles for interpolation turning
<b>101–130</b>	OEM options	02	579651-01 to -30	Options of the machine manufacturer
<b>131</b>	Spindle Synchronism	05	806270-01	Synchronization of two or more spindles
<b>133</b>	Remote Desk. Manager	01	894423-01	Display and remote operation of external computer units (e.g., a Windows PC)
<b>135</b>	Synchronizing Functions	04	1085731-01	Advanced synchronization of axes and spindles
<b>136</b>	Visual Setup Control	06	1099457-01	VSC: camera-based monitoring of the setup situation
<b>137</b>	State Reporting	09	1232242-01	Operating states provided over the State Reporting Interface (SRI)
<b>141</b>	Cross Talk Comp.	02	800542-01	CTC: compensation of axes coupling
<b>142</b>	Pos. Adapt. Control	02	800544-01	PAC: position-dependent adaptation of the control parameters
<b>143</b>	Load Adapt. Control	02	800545-01	LAC: load-dependent adaptation of the control parameters
<b>144</b>	Motion Adapt. Control	02	800546-01	MAC: motion-dependent adaptation of the control parameters
<b>145</b>	Active Chatter Control	02	800547-01	ACC: active suppression of chatter during heavy machining
<b>146</b>	Machine Vibration Control	04	800548-01	MVC: better surfaces. The following functions are part of MVC: <ul style="list-style-type: none"> <li>• Active Vibration Damping (AVD)</li> <li>• Frequency Shaping Control (FSC)</li> </ul>
<b>154</b>	Batch Process Manager	08	1219521-01	Depiction of pallet management in a clear format
<b>155</b>	Component Monitoring	09	1226833-01	Function for the monitoring of components for overload and wear

Option number	Option	With NC software 34059x- or later	ID	Comment
156	Grinding	10	1237232-01	Grinding functions: <ul style="list-style-type: none"> <li>• Jig grinding</li> <li>• Ability to switch between normal operation and dressing mode</li> <li>• Reciprocating stroke</li> <li>• Grinding cycles</li> <li>• Tool management for grinding and dressing</li> </ul>
157	Gear Cutting	09	1237235-01	Machining of gear teeth
158	Advanced Function Set Turning	09	1237237-01	Advanced turning functions
160	Integrated FS: Basic	10	1249928-01	Gen 3 exclusive: <ul style="list-style-type: none"> <li>• Enables 4 safe control loops</li> <li>• Enables functional safety (FS)</li> </ul>
161	Integrated FS: Full	10	1249929-01	Gen 3 exclusive: <ul style="list-style-type: none"> <li>• Enables the maximum number of safe control loops</li> <li>• Enables functional safety (FS)</li> </ul>
162	Add. FS Ctrl. Loop 1	10	1249930-01	Only for Gen 3: additional safe control loop 1
163	Add. FS Ctrl. Loop 2	10	1249931-01	Only for Gen 3: additional safe control loop 2
164	Add. FS Ctrl. Loop 3	10	1249932-01	Only for Gen 3: additional safe control loop 3
165	Add. FS Ctrl. Loop 4	10	1249933-01	Only for Gen 3: additional safe control loop 4
166	Add. FS Ctrl. Loop 5	10	1249934-01	Only for Gen 3: additional safe control loop 5
167	Optimized Contour Milling (OCM)	10	1289547-01	Optimization of roughing processes
169	Add. FS Full	11	1319091-01	Only for Gen 3: enables remaining control loops (as with option 161). Before being able to activate and use this function, you must activate options 160 and 162 through 166

Specifications	Standard	Option	
<b>Components</b>	✓ ✓ ✓		MC main computer CC or UxC controller unit Multi-touch operation 15-inch screen (1024 x 768 pixels) 19-inch screen (portrait, 1024x1280 pixels) 19-inch screen (1280x1024 pixels) 24-inch screen (1920x1080 pixels) TE operating panel (suitable for 19-inch or 24-inch screen)
<b>Operating system</b>	✓		HEROS 5 real-time operating system for machine control
<b>Memory</b>	✓ ✓		Main memory: 4 GB RAM NC program memory: 21.4 GB (with 32 GB SSDR) 189 GB (with 240 GB SSDR)
<b>Input resolution and display step</b>	✓ ✓		Linear axes: down to 0.01 µm Rotary axes: down to 0.00001°
<b>Input range</b>	✓		Maximum: 999 999 999 mm or 999 999 999°
<b>Interpolation</b>	✓ ✓ ✓	9 8	Linear in four axes Linear in up to 6 axes (export license required) Circular in two axes Circular in three axes with tilted working plane Helical: superimposition of circular and straight paths
<b>Block processing time</b>	✓		0.5 ms (3D straight line without radius compensation)
<b>Axis feedback control</b>	✓ ✓ ✓ ✓		Position-loop resolution: signal period of the position encoder/4096 Position controller cycle time: 200 µs (100 µs with option 49) Speed controller cycle time: 200 µs (100 µs with option 49) Current controller cycle time: minimum 100 µs (minimum 50 µs with option 49)
<b>Spindle speed</b>	✓		Maximum: 100 000 rpm (with two pole pairs)
<b>Error compensation</b>	✓ ✓		Linear and nonlinear axis error, backlash, reversal peaks during circular movements, reversal error, thermal expansion Static friction, sliding friction
<b>Data interfaces</b>	✓ ✓ ✓ ✓	18 56–61	One each: RS-232-C/V.24 max. 115 kbit/s Extended data interface with LSV2 protocol for remote operation of the TNC over the data interface with the TNCremo or TNCremoPlus software from HEIDENHAIN 2 x Gigabit Ethernet interface 1000BASE-T 4 x USB (3.0) HEIDENHAIN-DNC for communication between a Windows application and the TNC (DCOM interface) HEIDENHAIN OPC UA NC Server Reliable and stable interface for the connection of state-of-the-art industrial applications
<b>Diagnostics</b>	✓		Fast and simple troubleshooting through integrated diagnostic aids
<b>Ambient temperature</b>	✓ ✓		Operation: +5 °C to +40 °C Storage: –20 °C to +60 °C



# Overview

## Comparison of controls

Comparison of controls	TNC 620 NC SW 81760x-08	TNC 640 NC SW 34059x-11
Area of application	Standard milling	High-end milling / turning / grinding
Simple machining centers (up to 8 control loops, max. 2 of which are spindles)	✓	✓
Machine tools/machining centers (up to 24 control loops, max. 4 of which are spindles)	–	✓
Milling / turning / grinding operations (up to 18 control loops + 2 spindles)	–	Option
<b>Program entry</b>		
HEIDENHAIN Klartext	✓	✓
ISO programming	✓	✓
CAD Import	Option	Option
CAD viewer	✓	✓
FK free contour programming	Option	✓
Extended milling and drilling cycles	Option	✓
OCM cycles	Option	Option
Turning cycles	–	Option
Functions for grinding	–	Option
<b>NC program memory</b>	30 GB (CompactFlash CFR)	21.4 GB (with 32 GB SSDR) 189 GB (with 240 GB SSDR)
<b>5-axis and high-speed machining</b>	Option	Option
<b>Block processing time</b>	1.5 ms	0.5 ms
<b>Input resolution and display step (standard)</b>	0.01 µm	0.01 µm
<b>Screen and keyboard</b>	15-inch touchscreen/ 19-inch touchscreen (portrait)	15-inch / 19-inch / 24-inch touchscreen
<b>Touchscreen</b>	✓	✓
<b>Adaptive Feed Control (AFC)</b>	–	Option

Comparison of controls	TNC 620 NC SW 81760x-08	TNC 640 NC SW 34059x-11
Area of application	Standard milling	High-end milling / turning / grinding
<b>Active Chatter Control (ACC)</b>	Option	Option
<b>Dynamic Collision Monitoring (DCM)</b>	–	Option
Fixture monitoring	–	Option
<b>KinematicsOpt</b>	Option	Option
<b>KinematicsComp</b>	–	Option
<b>Touch probe cycles</b>	Option	✓
<b>Pallet Management</b>	Option	✓
<b>Parallel axis function</b>	✓	✓

- ✓ Standard
- Not available

# 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](http://www.heidenhain.de)  
For complete and further addresses see [www.heidenhain.com](http://www.heidenhain.com)

<b>DE</b>	<b>HEIDENHAIN Vertrieb Deutschland</b> 83301 Traunreut, Deutschland ☎ 08669 31-3132 FAX 08669 32-3132 E-Mail: hd@heidenhain.de	<b>DK</b>	Denmark → SE	<b>NZ</b>	<b>Llama ENGINEERING Ltd</b> 5012 Wellington, New Zealand E-mail: info@llamaengineering.co.nz
	<b>HEIDENHAIN Technisches Büro Nord</b> 12681 Berlin, Deutschland ☎ 030 54705-240	<b>ES</b>	<b>FARRESA ELECTRONICA S.A.</b> 08028 Barcelona, Spain www.farresa.es	<b>PH</b>	<b>MACHINEBANKS' CORPORATION</b> Quezon City, Philippines 1113 E-mail: info@machinebanks.com
	<b>HEIDENHAIN Technisches Büro Mitte</b> 07751 Jena, Deutschland ☎ 03641 4728-250	<b>FI</b>	<b>HEIDENHAIN Scandinavia AB</b> 01740 Vantaa, Finland www.heidenhain.fi	<b>PL</b>	<b>APS</b> 02-384 Warszawa, Poland www.heidenhain.pl
	<b>HEIDENHAIN Technisches Büro West</b> 44379 Dortmund, Deutschland ☎ 0231 618083-0	<b>FR</b>	<b>HEIDENHAIN FRANCE sarl</b> 92310 Sèvres, France www.heidenhain.fr	<b>PT</b>	<b>FARRESA ELECTRÓNICA, LDA.</b> 4470 - 177 Maia, Portugal www.farresa.pt
	<b>HEIDENHAIN Technisches Büro Südwest</b> 70771 Leinfelden-Echterdingen, Deutschland ☎ 0711 993395-0	<b>GB</b>	<b>HEIDENHAIN (G.B.) Limited</b> Burgess Hill RH15 9RD, United Kingdom www.heidenhain.co.uk	<b>RO</b>	<b>HEIDENHAIN Reprezentantă Romania</b> Braşov, 500407, Romania www.heidenhain.ro
	<b>HEIDENHAIN Technisches Büro Südost</b> 83301 Traunreut, Deutschland ☎ 08669 31-1337	<b>GR</b>	<b>MB Milionis Vassilis</b> 17341 Athens, Greece www.heidenhain.gr	<b>RS</b>	Serbia → BG
		<b>HR</b>	Croatia → SL	<b>RU</b>	<b>GERTNER Service</b> 119002 Moscow, Russian Federation www.heidenhain.ru
		<b>HU</b>	<b>HEIDENHAIN Kereskedelmi Képviselet</b> 1239 Budapest, Hungary www.heidenhain.hu	<b>SE</b>	<b>HEIDENHAIN Scandinavia AB</b> 12739 Skärholmen, Sweden www.heidenhain.se
<b>AR</b>	<b>NAKASE SRL.</b> B1653AOX Villa Ballester, Argentina www.heidenhain.com.ar	<b>ID</b>	<b>PT Servitama Era Toolsindo</b> Jakarta 13930, Indonesia E-mail: ptset@group.gts.co.id	<b>SG</b>	<b>HEIDENHAIN PACIFIC PTE LTD</b> Singapore 408593 www.heidenhain.com.sg
<b>AT</b>	<b>HEIDENHAIN Techn. Büro Österreich</b> 83301 Traunreut, Germany www.heidenhain.de	<b>IL</b>	<b>NEUMO VARGUS MARKETING LTD.</b> Holon, 5885948, Israel E-mail: neumo@neumo-vargus.co.il	<b>SK</b>	<b>KOPRETINA TN s.r.o.</b> 91101 Trenčin, Slovakia www.kopretina.sk
<b>AU</b>	<b>FCR MOTION TECHNOLOGY PTY LTD</b> Ravenhall Victoria 3023, Australia E-mail: sales@fcrmotion.com	<b>IN</b>	<b>HEIDENHAIN Optics &amp; Electronics India Private Limited</b> Chetpet, Chennai 600 031, India www.heidenhain.in	<b>SL</b>	<b>NAVO d.o.o.</b> 2000 Maribor, Slovenia www.heidenhain.si
<b>BE</b>	<b>HEIDENHAIN NV</b> 1760 Roosdaal, Belgium www.heidenhain.be	<b>IT</b>	<b>HEIDENHAIN ITALIANA S.r.l.</b> 20128 Milano, Italy www.heidenhain.it	<b>TH</b>	<b>HEIDENHAIN (THAILAND) LTD</b> Bangkok 10250, Thailand www.heidenhain.co.th
<b>BG</b>	<b>ESD Bulgaria Ltd.</b> Sofia 1172, Bulgaria www.esd.bg	<b>JP</b>	<b>HEIDENHAIN K.K.</b> Tokyo 102-0083, Japan www.heidenhain.co.jp	<b>TR</b>	<b>T&amp;M Mühendislik San. ve Tic. LTD. ŞTİ.</b> 34775 Y. Dudullu – Umraniye-Istanbul, Turkey www.heidenhain.com.tr
<b>BR</b>	<b>HEIDENHAIN Brasil Ltda.</b> 04763-070 – São Paulo – SP, Brazil www.heidenhain.com.br	<b>KR</b>	<b>HEIDENHAIN Korea Ltd.</b> Anyang-si, Gyeonggi-do, 14087 South Korea www.heidenhain.co.kr	<b>TW</b>	<b>HEIDENHAIN CO., LTD.</b> Taichung 40768, Taiwan www.heidenhain.com.tw
<b>BY</b>	<b>GERTNER Service</b> 220026 Minsk, Belarus www.heidenhain.by	<b>MX</b>	<b>HEIDENHAIN CORPORATION MEXICO</b> 20290 Aguascalientes, AGS., Mexico E-mail: info@heidenhain.com	<b>UA</b>	<b>GERTNER Service</b> 02094 Kiev, Ukraine www.heidenhain.ua
<b>CA</b>	<b>HEIDENHAIN CORPORATION</b> Mississauga, Ontario L5T2N2, Canada www.heidenhain.com	<b>MY</b>	<b>ISOSERVE SDN. BHD.</b> 43200 Balakong, Selangor E-mail: sales@isoserve.com.my	<b>US</b>	<b>HEIDENHAIN CORPORATION</b> Schaumburg, IL 60173-5337, USA www.heidenhain.us
<b>CH</b>	<b>HEIDENHAIN (SCHWEIZ) AG</b> 8603 Schwerzenbach, Switzerland www.heidenhain.ch	<b>NL</b>	<b>HEIDENHAIN NEDERLAND B.V.</b> 6716 BM Ede, Netherlands www.heidenhain.nl	<b>VN</b>	<b>AMS Co. Ltd</b> HCM City, Vietnam E-mail: davidgoh@amsvn.com
<b>CN</b>	<b>DR. JOHANNES HEIDENHAIN (CHINA) Co., Ltd.</b> Beijing 101312, China www.heidenhain.com.cn	<b>NO</b>	<b>HEIDENHAIN Scandinavia AB</b> 7300 Orkanger, Norway www.heidenhain.no	<b>ZA</b>	<b>MAFEMA SALES SERVICES C.C.</b> Kyalami 1684, South Africa www.heidenhain.co.za
<b>CZ</b>	<b>HEIDENHAIN s.r.o.</b> 102 00 Praha 10, Czech Republic www.heidenhain.cz				