TNC 640

Contouring Control for Machining Centers and Milling-Turning Machines

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-18

2

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

For more than 45 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 TNC 640 user.

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 HEIDEN-HAIN. The TNC 640 provides you with optimum support with practical prompts, questions, and expressive graphical aids—for turning operations, too.

For standard machining operations and even complex applications, a wealth of practical machining and coordinate trans-formation 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 control loops. The TNC 640 is especially attractive for the following areas of application:

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
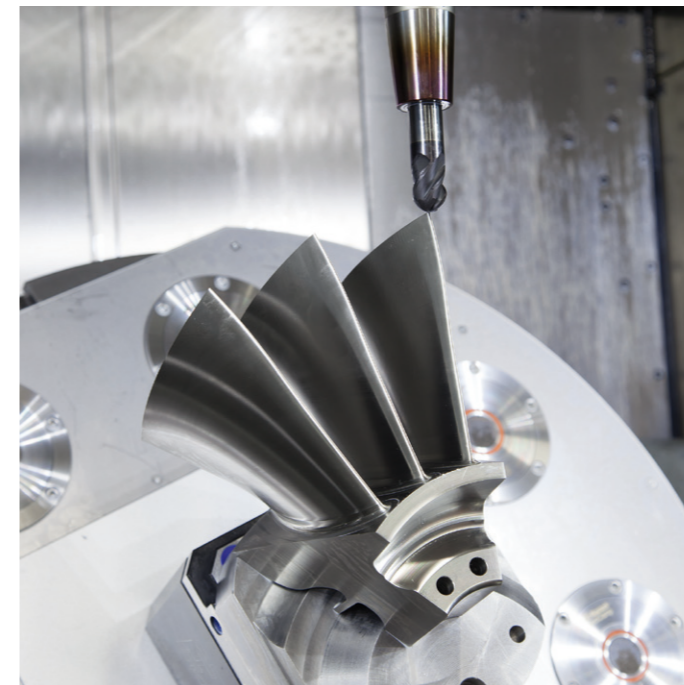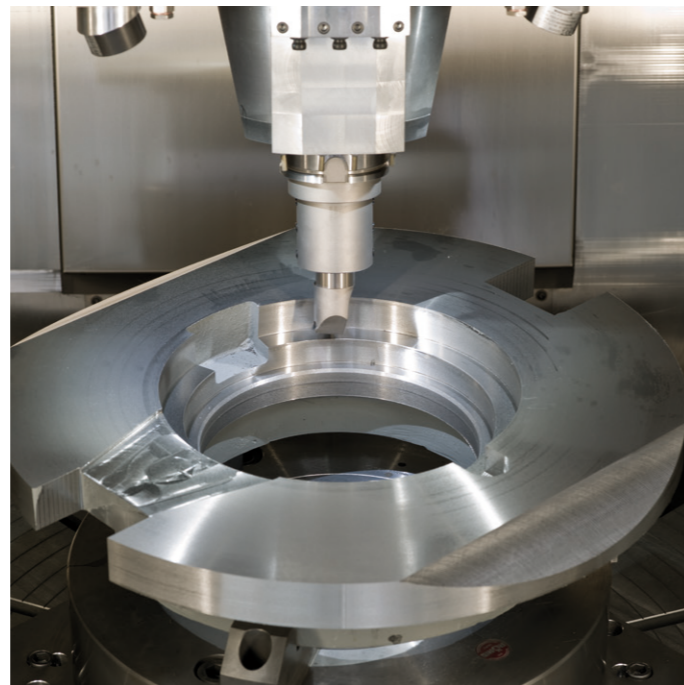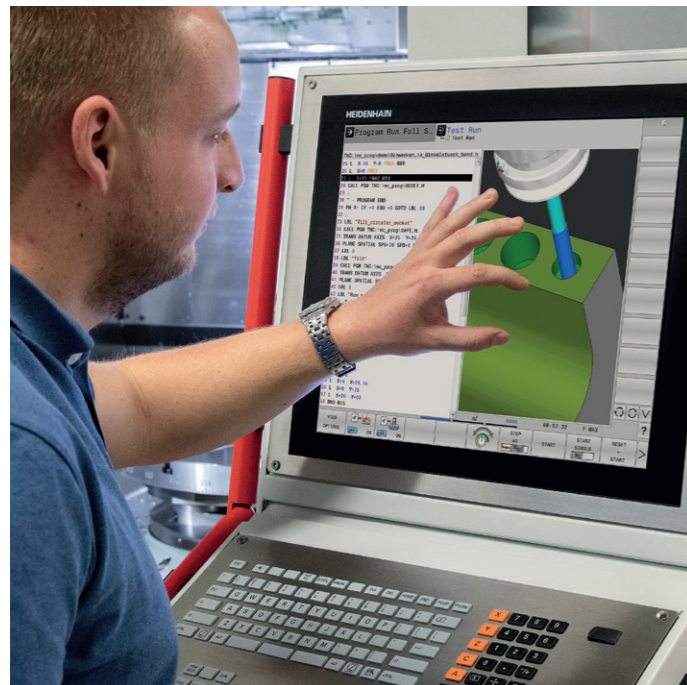
Intuitive and user-friendly
State-of-the-art multitouch operation

Monitor
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 there 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 with 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 conventional operating approach of the TNC has proven itself over many years, with users around the world using the TNC 640's dialog keys, navigation keys, and soft keys. But the touchscreen version of the TNC 640 now assists you with a particularly innovative and user-friendly method of operation. 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 the screen, simply select the touchscreen cleaning mode. This locks the screen to prevent unintended operation.

Gestures for multitouch operation
The screen of the TNC 640 can be operated with gestures like the ones you use on 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

Highly practical 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.

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.

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.

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, and 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 help you always find your way even in complex programs, the individual program elements on the TNC 640 are shown in different colors. This allows you to see at a glance which input values are editable.

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 increases data security but also promotes operating safety.
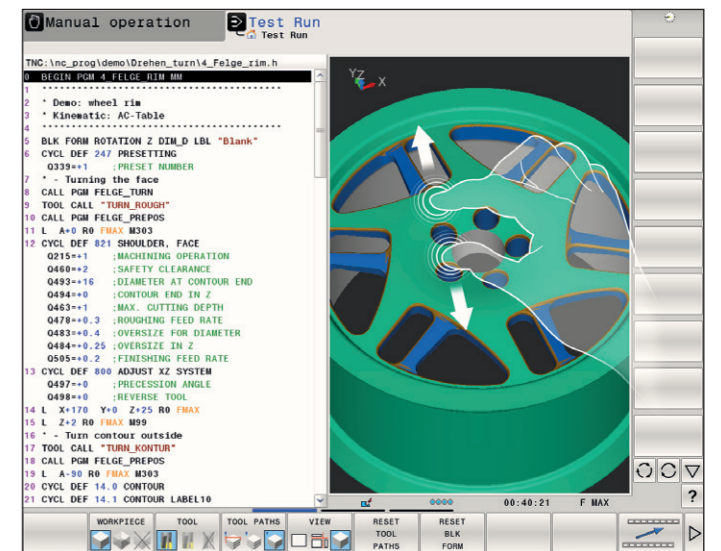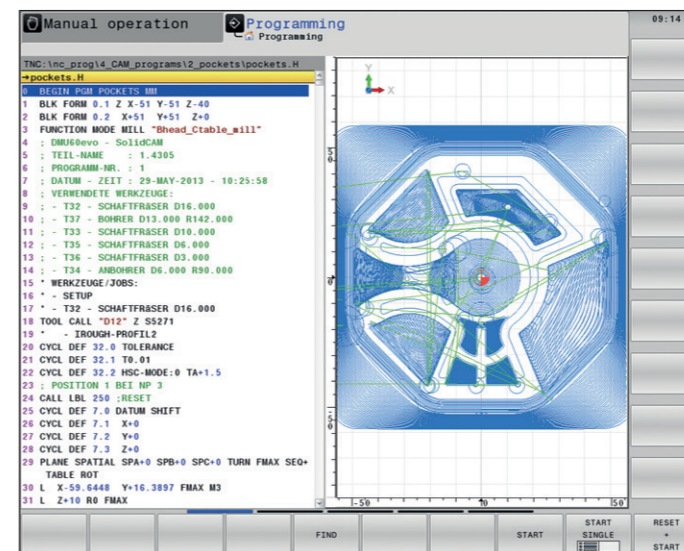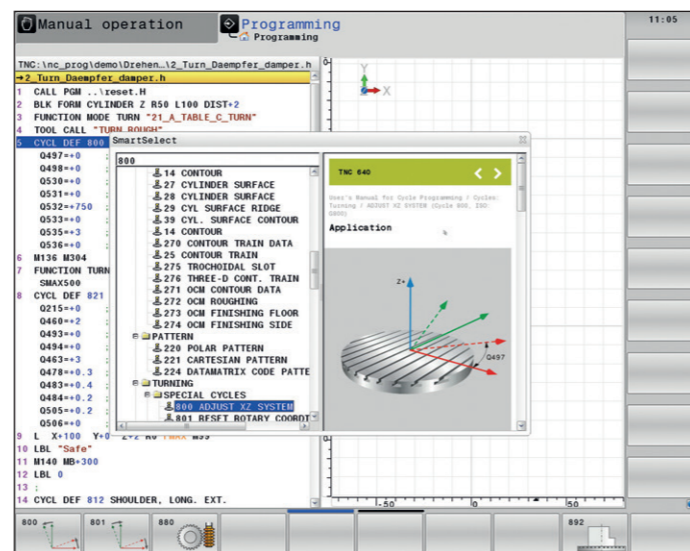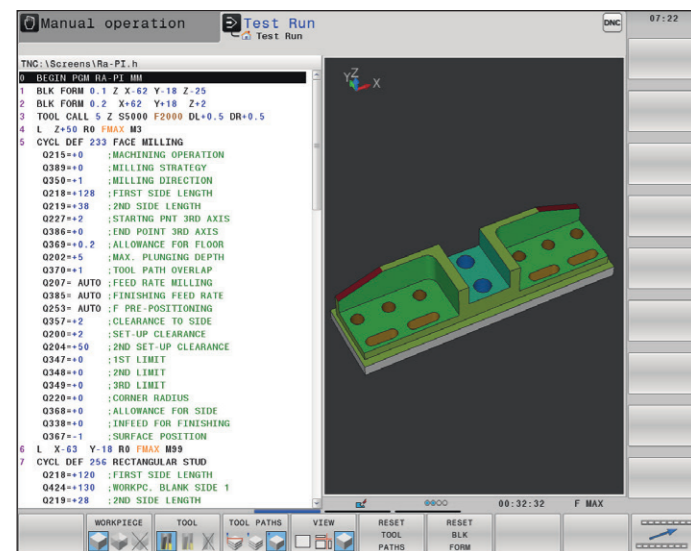
Complete part machining
Milling, turning and grinding on one machine (option)

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 it, 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 gives you powerful functions for easily switching between turning, dressing, and milling modes, all from within the NC program. 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 control must be adapted for these functions by the machine manufacturer.

Programming as usual
Turning operations can be programmed as always using the convenient HEIDENHAIN Klartext 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 software option.

Milling, turning and grinding cycles
HEIDENHAIN controls have always been known for their comprehensive and technologically advanced package of cycles. Frequently recurring, multi-step machining operations are stored as cycles in the TNC 640 as well. 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 numerous turning cycles for roughing, finishing, recessing, thread turning, 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 (option)
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.

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 motions 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.
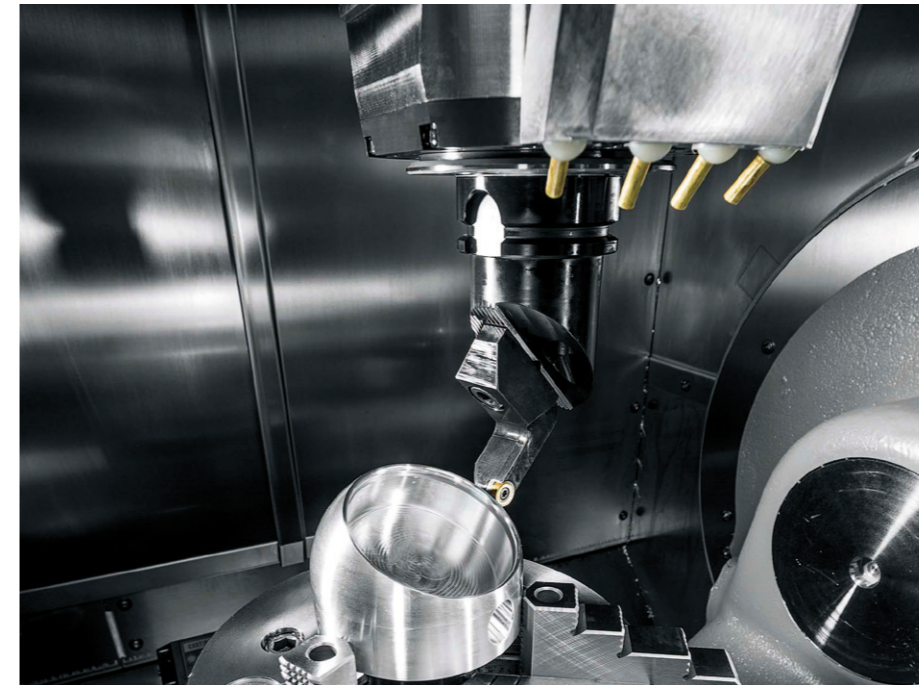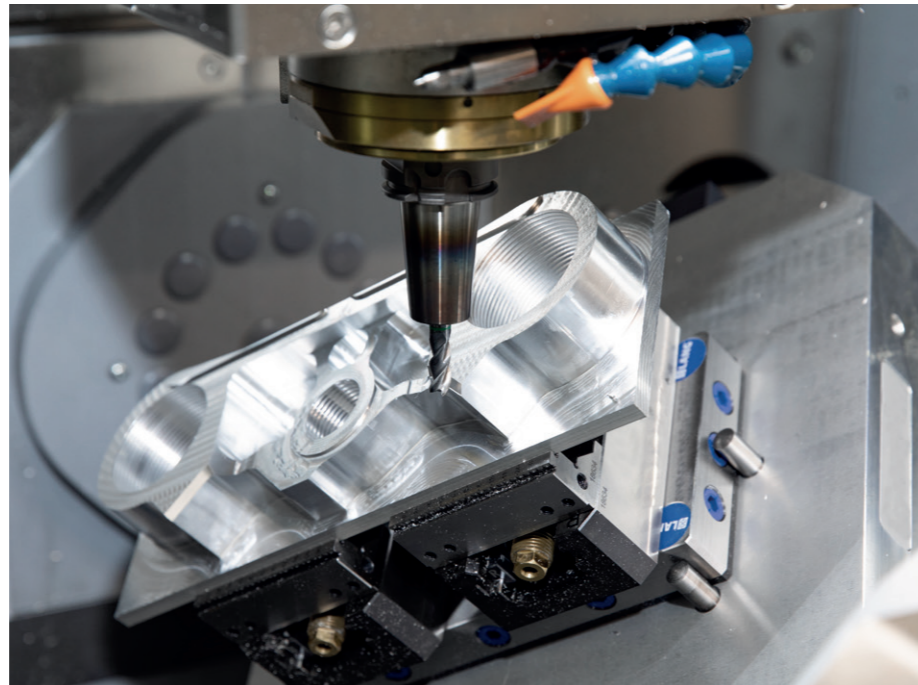
Fast and reliable machining at high contour fidelity
Dynamic Precision

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 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

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

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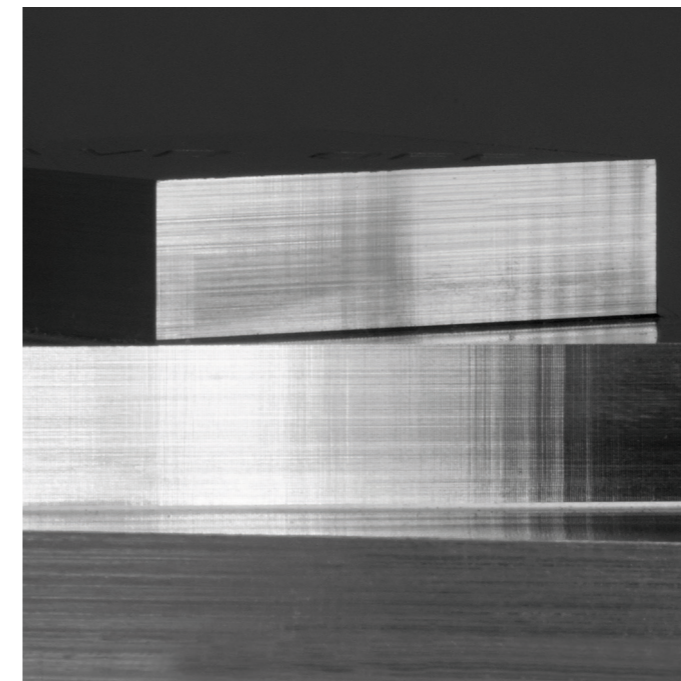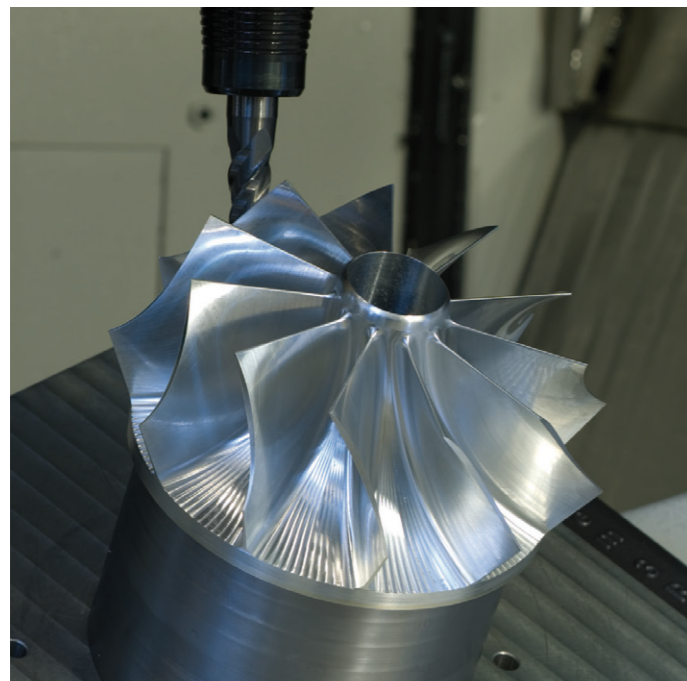
Vibration can significantly impair surface quality.
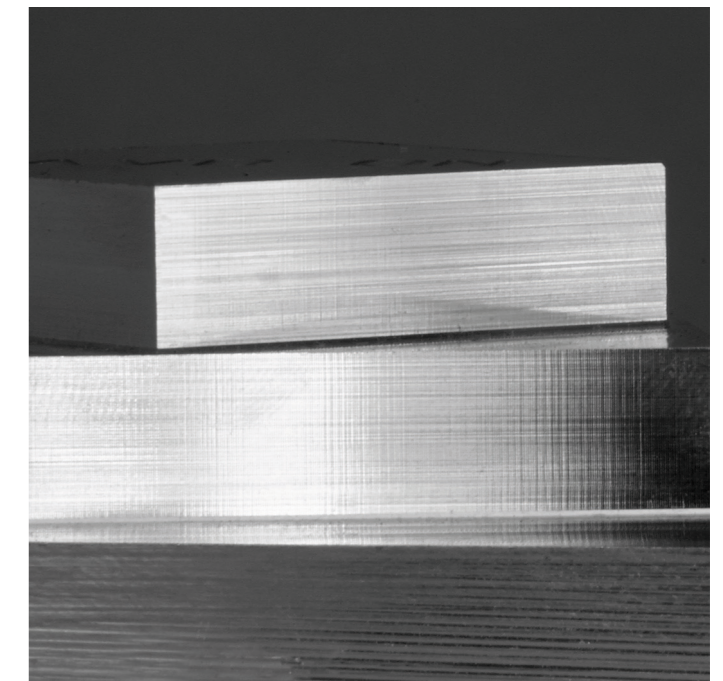
Visibly superior surface quality can be attained with MVC.

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 then automatically adapts the machining operation to the selected tolerance. 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 fully 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 100 000 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.
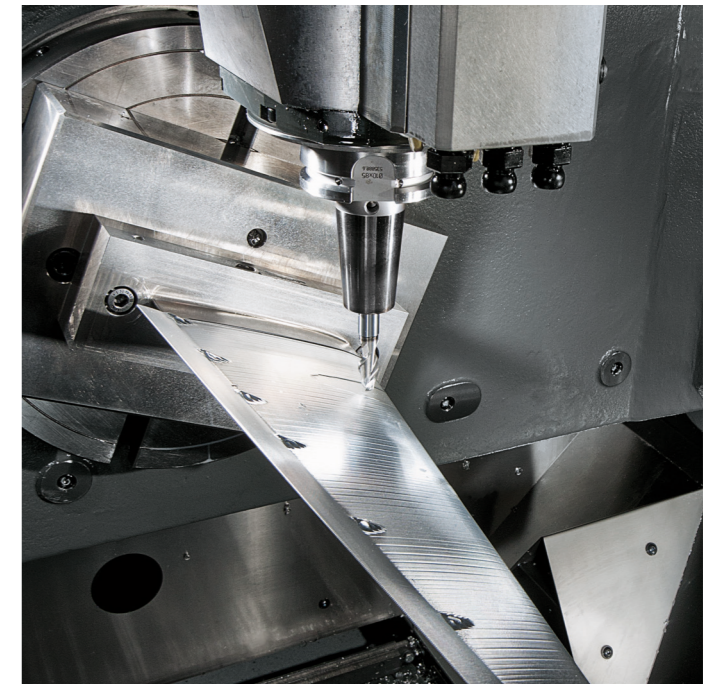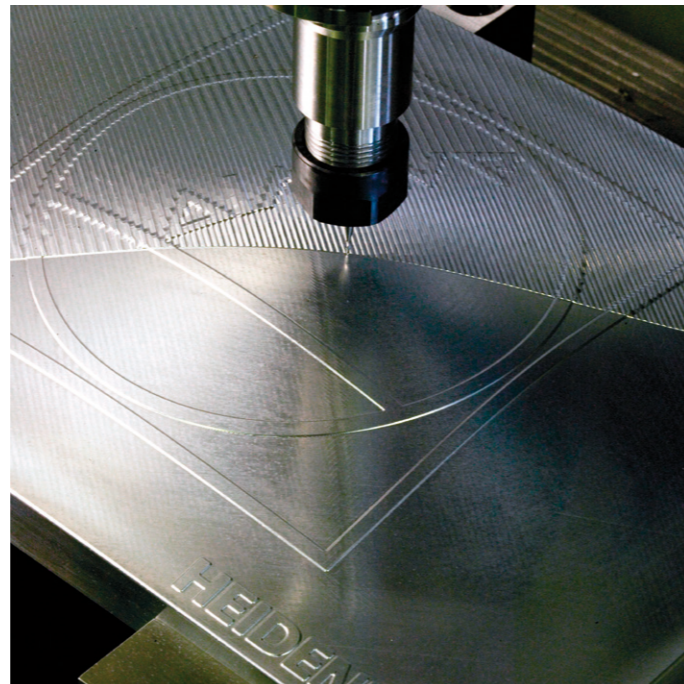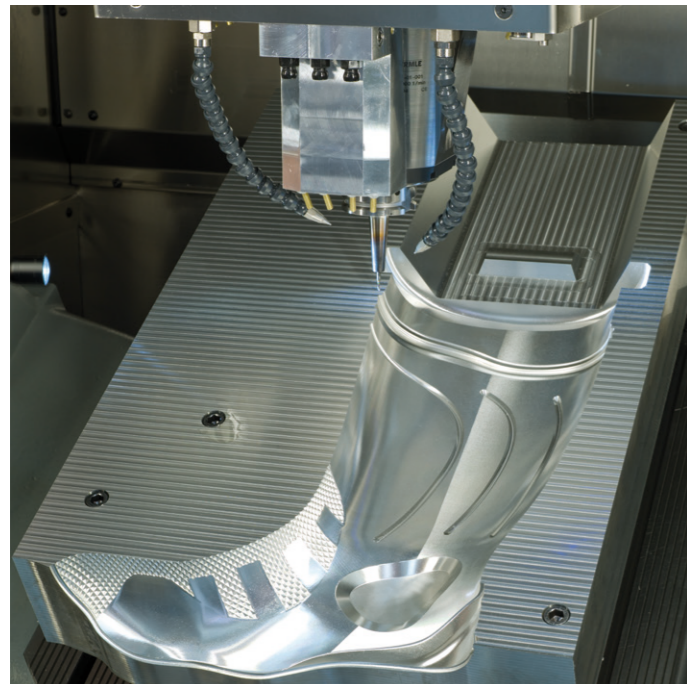
Compensating for tool form error

3D-ToolComp is a powerful software option for three-dimensional tool radius compensation. 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 optimal procedure is to measure the tool with a laser system and a special cycle, automatically creating a compensation table ready for immediate use by the TNC 640. 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

With Cycle 444 (3D Probing) you can 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. The cycle then compensates for the switching behavior of the individual touch probe in all directions. The 3D-ToolComp software option is required for 3D calibration and for Cycle 444.

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 5-axis machining on machines with three linear axes and two additional tilting axes,* the tool is always kept perpendicular to the surface of the workpiece or is titled at a specific angle relative to it (inclined tool machining).

Regardless of what type of 5-axis programs you wish to run, the TNC 640 performs all of the 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 control 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 movies 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: Very high axis feed rates can result from large compensating motions during machining near the center of tilting.

- Contouring feed rate of the axes programmed in the respective NC block: The feed rate is usually lower, but you attain better surface quality during 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.
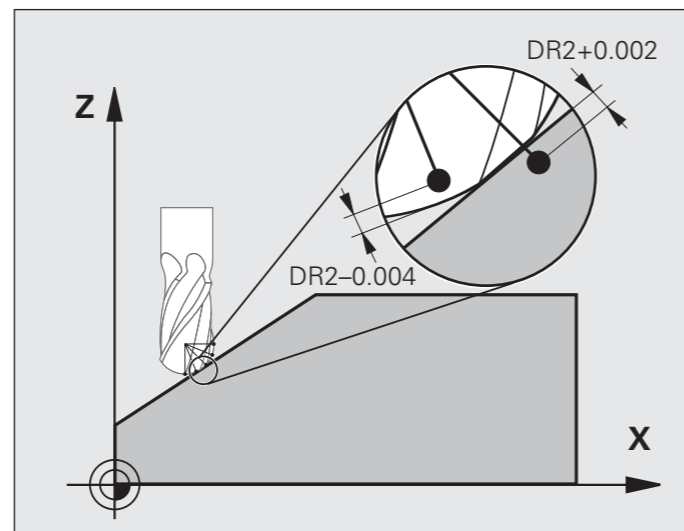
DR2+0.002

DR2−0.004
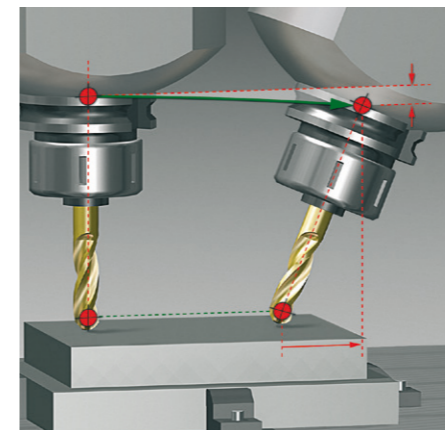
Face milling
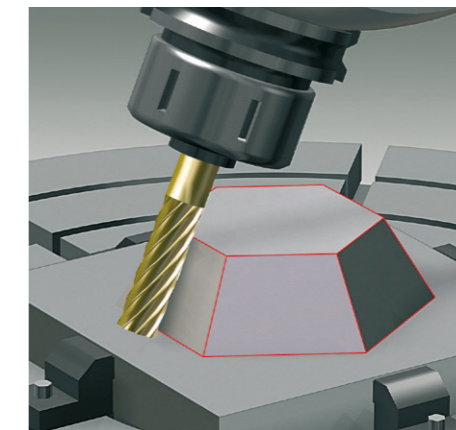
Peripheral milling

Inclined-tool machining

TNC 640-controlled swivel head and rotary table
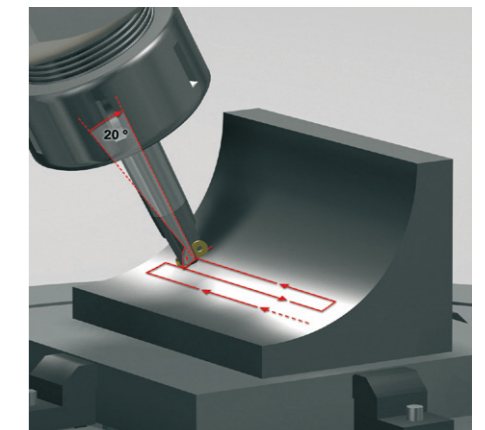
Many five-axis operations that initially appear to be highly complex can be broken down into common 2D movements that have been tilted about one or more rotary axes or wrapped around a cylindrical surface. In order for you to create and edit such programs quickly and easily without a CAM system, the TNC assists 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 the working plane (e.g., the X/Y plane); the machine tool then performs the operation in the 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.
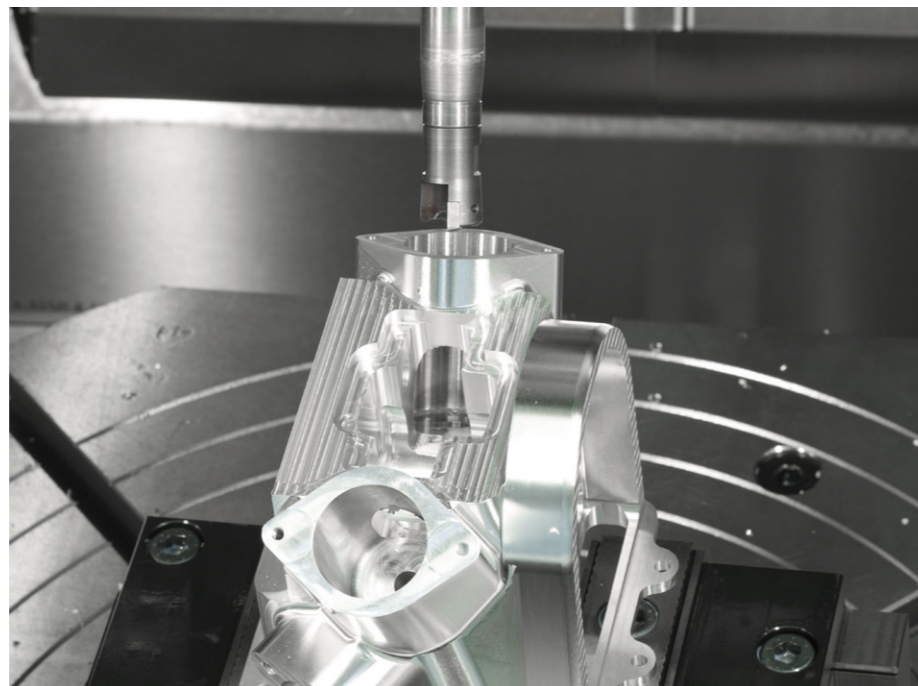
Cylinder surface machining*
The programming of contours made up of straight lines and circles on cylindrical surfaces for rotary tables is no problem with the TNC 640. You simply program the contour in the plane of an unrolled cylinder surface. The TNC 640 then performs the machining operation on the 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

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 axes and tables in mm/min*
By default, rotary axis feed rates are programmed in degrees/min, but the TNC 640 can also interpret these feed rates in mm/min. The feed rate at the contour is then independent of the distance of the tool center point from the center of the rotary axis.

* The control must be adapted for these functions by the machine manufacturer.

* The control must be adapted for these functions by the machine manufacturer.
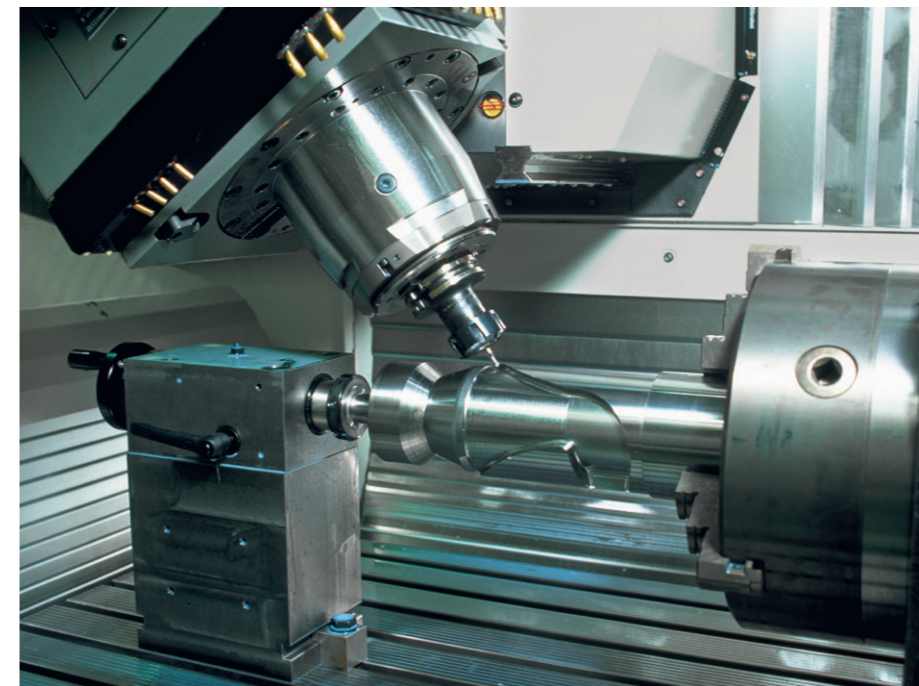
Intelligent machining
Dynamic collision monitoring (DCM, option)

The complex machine movements and normally high traversing speeds of five-axis machining make axis movements difficult to predict. This makes collision monitoring a valuable function that relieves the user 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 DCM* software option of the TNC 640 provides the machine user with peace of mind. The control interrupts the machining process whenever a collision is imminent, thus increasing safety for both the machine and its user. This, in turn, helps prevent machine damage and any resulting costly downtime. Unattended shifts also become more reliable.

DCM even works in both automatic mode and manual operation. If, for example, the machine user is on a collision course during setup, the TNC 640 detects this, stops axis movement, and issues an error message.

* The control must be adapted for these functions by the machine manufacturer.

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 enhanced checking for collisions between the workpiece and the non-cutting part of the tool or the tool holder.

Of course, the TNC 640 also shows the user 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.

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

A 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.

probe clamped to the machine table for purposes of tool measurement will never collide with the machine cabin. The machine manufacturer can therefore limit the machine elements that 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.
- The collision of machine parts (for example, the swivel head) with the workpiece cannot be detected.
- DCM cannot be used in servo-lag operation (no feedforward).
- DCM cannot be used for eccentric turning.

dynamic + efficiency
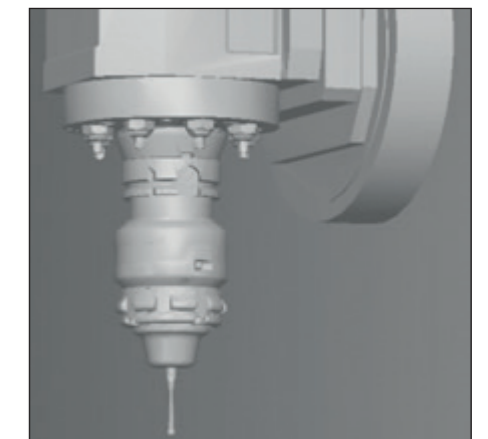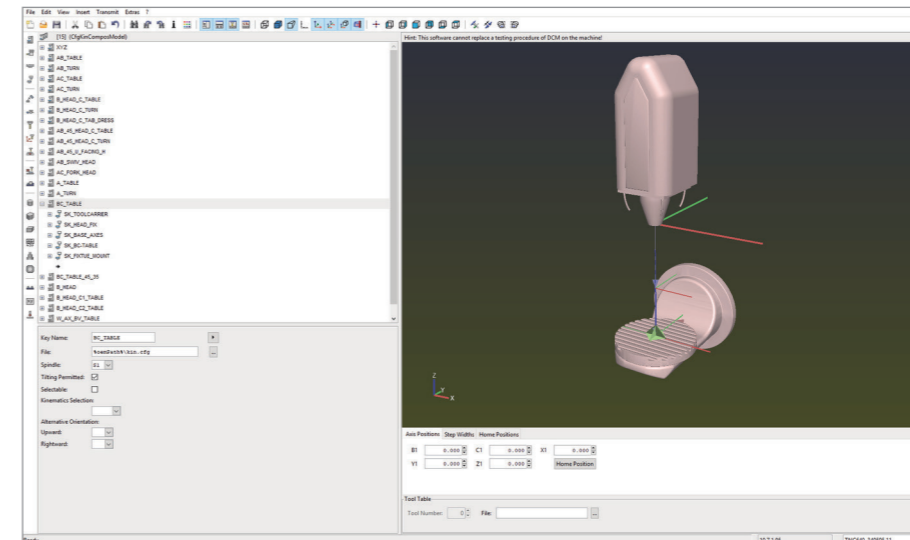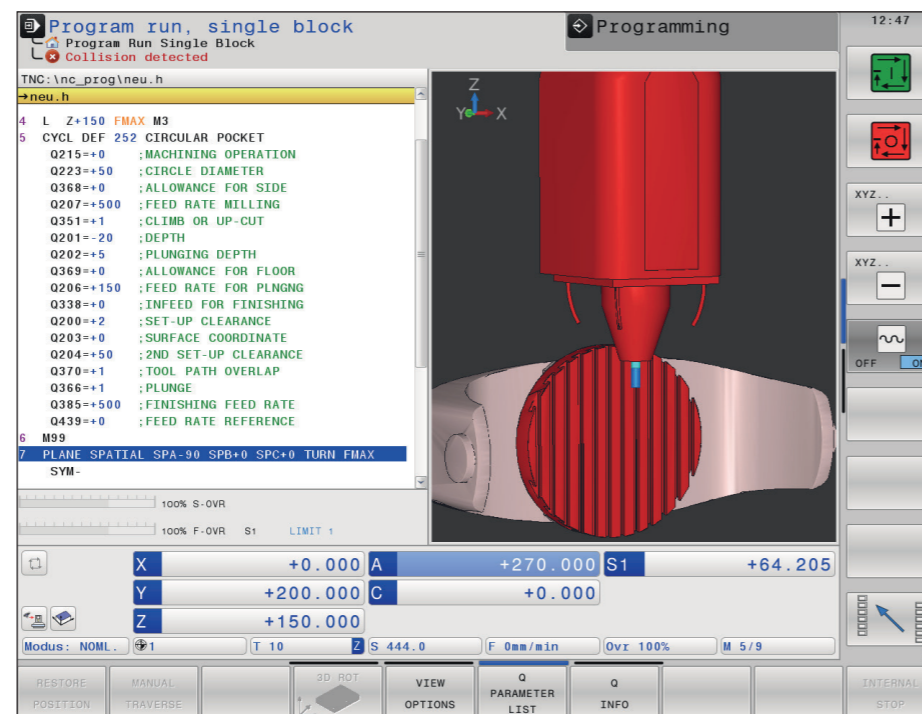
dynamic + efficiency

With the concept of Dynamic Efficiency, HEIDENHAIN offers innovative TNC functions that help the user to make heavy machining and roughing more efficient while also enhancing its process reliability. These software functions not only assist the user but also make the manufacturing process itself faster, more stable, and more predictable—in short, more efficient. By permitting higher removal rates, Dynamic Efficiency boosts productivity without requiring users to 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)** reduces chatter tendencies and thus permits higher feed rates and greater infeeds.
- **Adaptive Feed Control (AFC)** controls the feed rate depending on the machining situation.
- **Trochoidal milling** is a function for tool- and machine-friendly roughing of slots and pockets
- **Optimized Contour Milling (OCM)** allows pockets and islands of any shape to be machined with low tool wear using the highly efficient trochoidal milling method.

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 are also of great value under changing machining conditions, such as during interrupted cuts, various plunging processes, or simple roughing operations. In practice, the removal rate can be increased by 20 to 25 percent.

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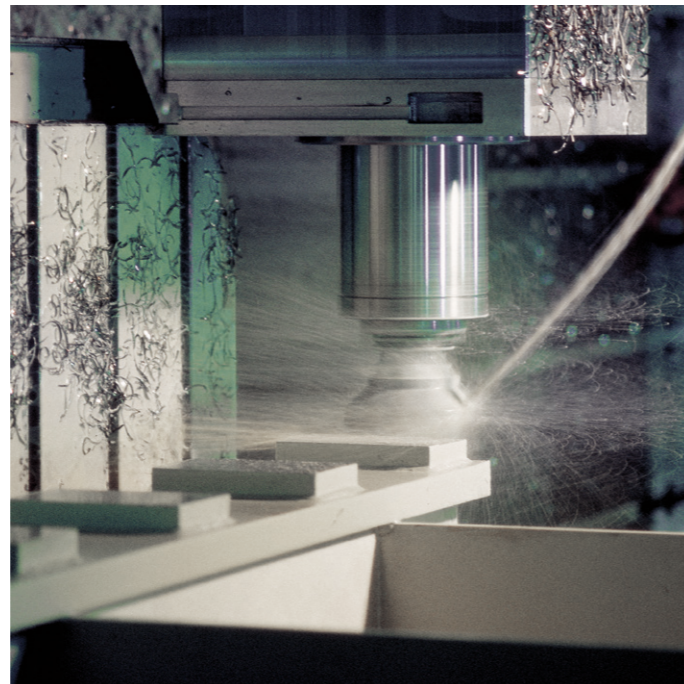
Heavy machining without ACC (upper image) and with ACC (lower image)

dynamic + efficiency

dynamic + efficiency

HEIDENHAIN controls have always allowed the user 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 user 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 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.
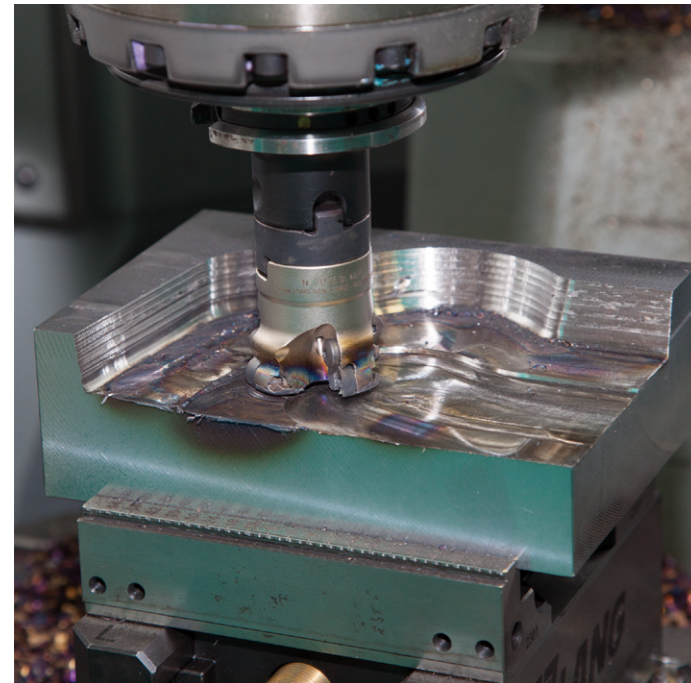
Reduced machining time
AFC regulates the feed rate of the TNC based on the given spindle power. In machining areas requiring less material removal, the feed rate is appropriately increased. This can significantly reduce the machining time.
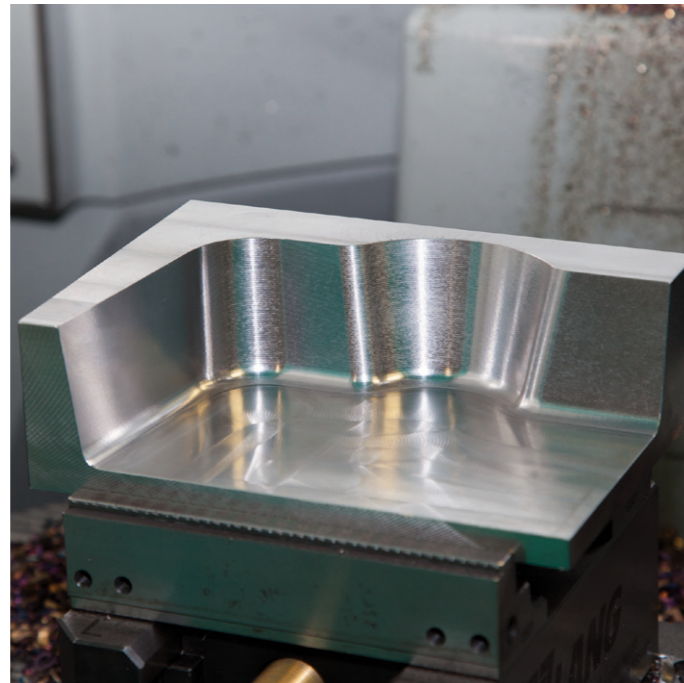
Gentler on the machine
When the maximum teach-in power is exceeded, the feed rate is reduced down to the reference spindle power. As a result, the machine undergoes less strain, and the main spindle is protected from overloading.

[*] The control must be adapted for these functions by the machine manufacturer.

The ability to machine any slot at high efficiency from start to finish is the key benefit of trochoidal milling. 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 damage resulting from a broken indexable insert

Fully machined workpiece protected by AFC
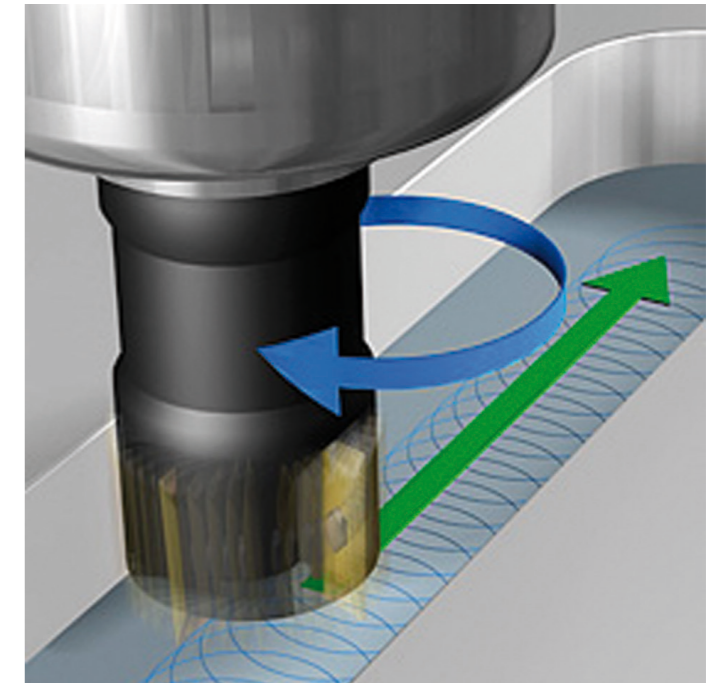
dynamic + efficiency

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 sizable portion of the total machining time.

Process reliability and maximum removal rates require optimal adaptation of the cutting data to the characteristics of the tool and workpiece material. Optimized Contour Milling (OCM) gives you a cutting data calculator that draws on a comprehensive integral materials database. You can adapt the automatically calculated cutting values with regard to the mechanical and thermal load on the tool. Tool service life can be managed with process reliability even at the highest possible removal rates.

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. OCM takes void areas into account, which significantly reduces machining times (with NC software version 16 or later).

Advantages of OCM over conventional machining:
- Reduced thermal load on the tool
- Superior chip removal
- Consistent tool contact conditions (higher cutting parameters and higher material removal rate)

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

OCM 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. 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 the tool wear were reduced by a factor of three.

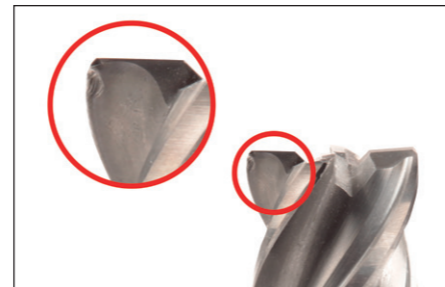| Conventional machining |
|---|
| *S5000, F1200, a_p:* 5.5 mm |
| *Overlap factor:* 5 mm |
| *Machining time:* 21 min 35 s |

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

Machining without OCM: Tool after two parts

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
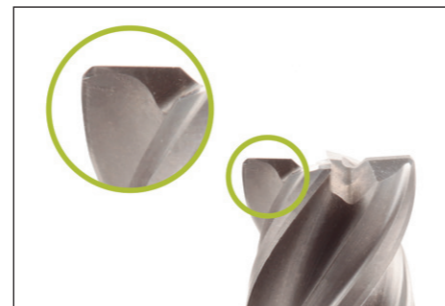
Machining with OCM: Tool after six parts
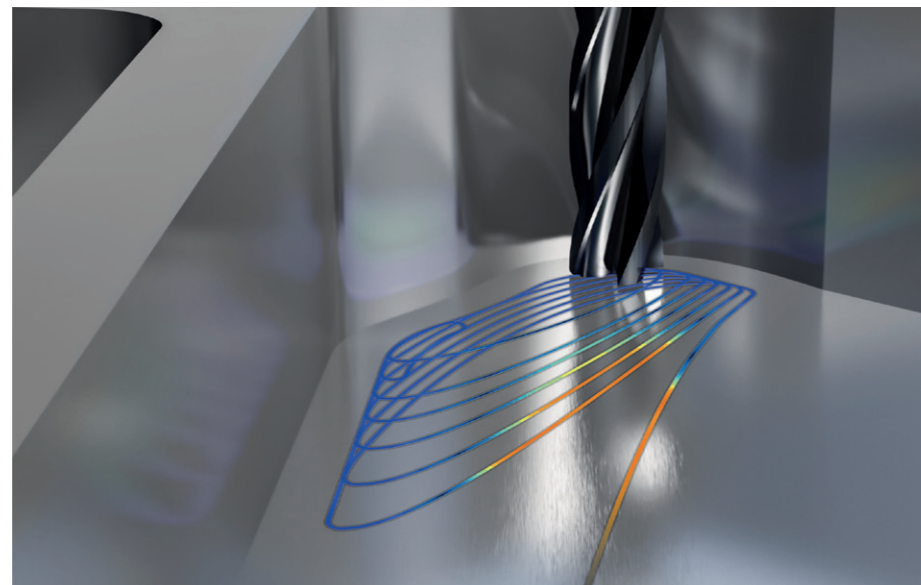
The Global Program Settings option is used especially for the production of large molds. It is available in the Program Run operating modes and in MDI mode. This option allows you to perform a variety of coordinate transformations and settings, and then apply them globally to a selected NC program without actually modifying it.

You can even change the global program settings mid-program during a program stop using a handy fillable form. Upon restarting, the TNC 640 then moves as needed to a new position in accordance with a positioning logic that you can define.

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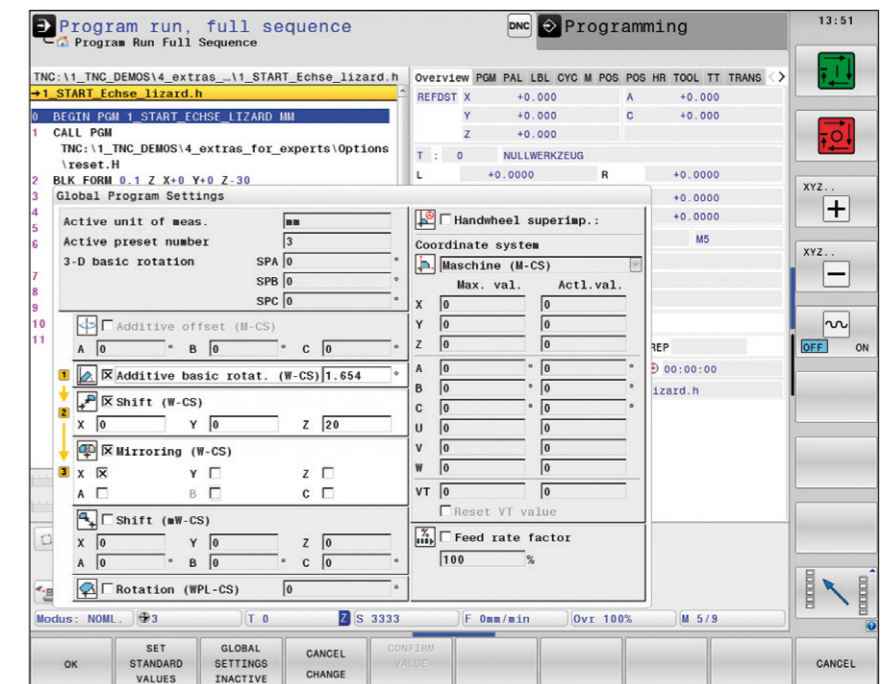
The requirements gap between machining centers and typical tool and moldmaking machines is becoming ever narrower. Of course, the TNC 640 is also very capable of controlling automated manufacturing processes. It features the necessary functionality to initiate the proper machining operation in concatenated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

Inspecting workpieces for completed machining and dimensional accuracy

The TNC 640 features a number of measuring cycles for checking the geometry of the machined workpieces. This requires inserting a touch probe from HEIDENHAIN into the spindle in place of the tool. This 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
- Inspect workpiece geometry and sort parts
- Log measured data
- Ascertain the machining trend
- Check tolerances 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, 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 p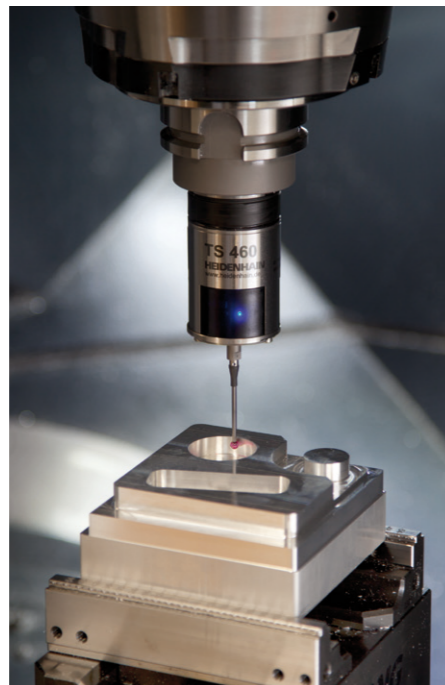reventing scrap and rework. If the measured deviations lie outside the specified tolerances or if the monitored tool life has been exceeded, the TNC 640 locks the tool and automatically inserts a replacement tool.

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. A pallet counter function lets you easily define the desired quantity to be produced.

Batch Process Manager (option)

Batch Process Manager is a powerful function for pallet machining and series production. Its straightforward user interface helps you plan your production process and gather important information on 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 preset, tool, and program

During program run 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 the Collision Monitoring software option).

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 assists 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.
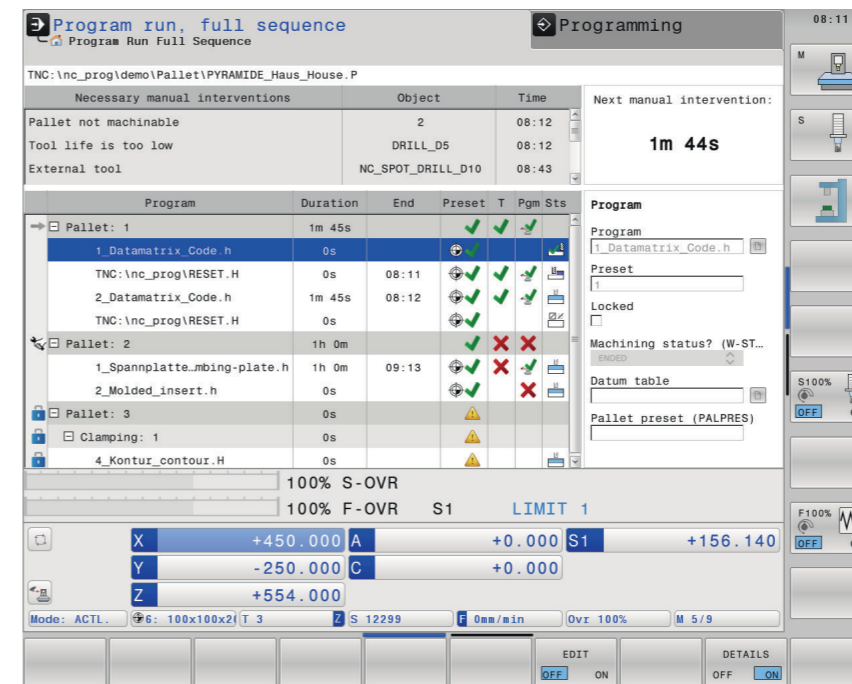
Tool management*

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 tool manager, the control now shows only the required input fields based on the selected tool type. Any data can also be graphically depicted. Extended tool management adds the "Tooling list" and the "T usage order". These tables enable you to detect the tool requirements in time and thus prevent interruptions during program run.

Component monitoring (option)

Overload often leads to damaged machine components, and therefore to machine downtime. For example, the spindle bearing is often subject to a very high load in machining strategies that have been optimized for maximum efficiency, and can therefore sustain undetected damage. Component Monitoring warns you about these dangers, and can even stop the machine tool if necessary. Thanks to continuous monitoring of the bearing load and visualization of these values, the machining processes can be optimized correspondingly.

However, the production quality of a machine tool is influenced not only by overload. Components that are constantly under load, such as guideways or the recirculating ball screw, are subject to wear and thus have an influence on the finished product. TNC 640 uses the Component Monitoring software option to measure and document the momentary machine status.

The machine manufacturer can read and evaluate these data, and react using predictive maintenance, thereby avoiding unplanned machine downtimes. The MONITORING HEATMAP function within the NC program lets you color the concurrent material removal simulation in the workpiece view with the status of a monitoring task. That way you see in the workpiece view where a component is subject to an excessive load.

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 assist 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.

The TNC 640 supports L-shaped styli in the probing cycles, which are used to quickly and easily probe undercuts.

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. 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

The HEIDENHAIN touch probes and probing functions of the TNC 640 reduce the need for tedious manual alignment of the workpiece:
- 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 you a comprehensive package of cycles for setting up the machine tool:
- Manual, automatic, and semiautomatic cycles for presetting and for compensating for misalignments in two or three dimensions
- Automatic probing cycle for repeating a measurement along a line

* The control must be adapted for these functions by the machine manufacturer.

Workpiece misalignment compensated for by rotating the coordinate system or rotating the table
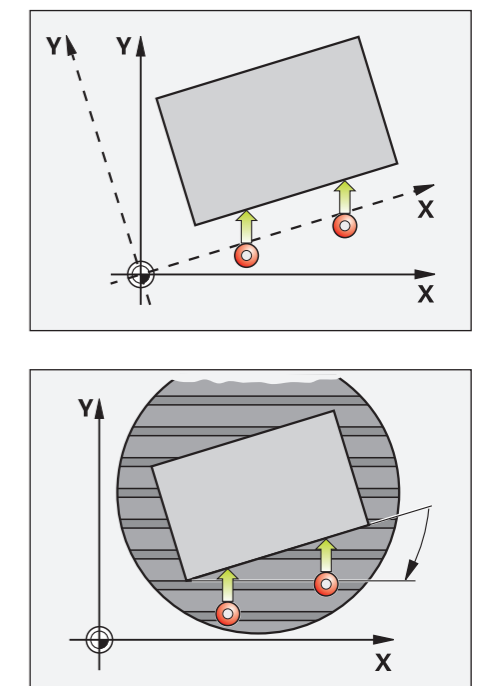
Setting presets

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

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

Individual machining steps

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 workshop oriented, which means that they were conceived for 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.

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 can present different views 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 tinie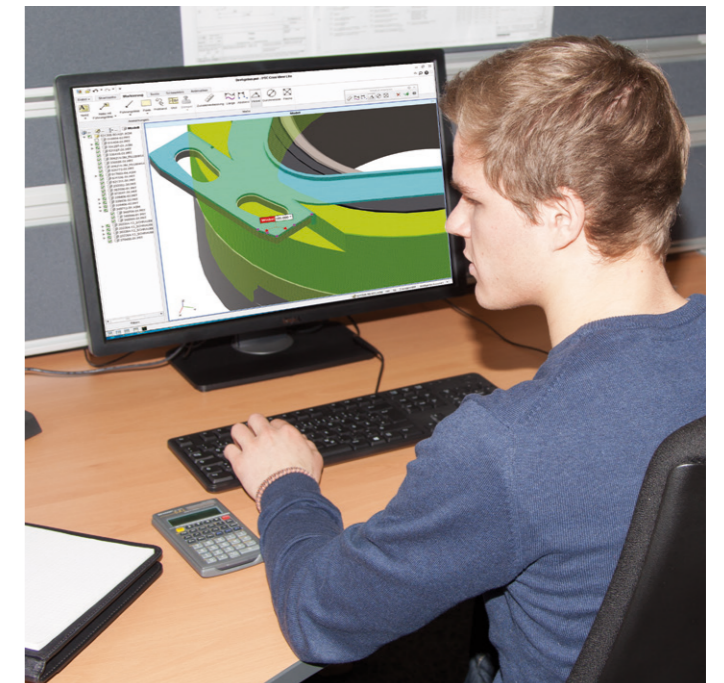st details. The 3D line graphics are especially helpful in verifying programs created offline, allowing you to avoid undesired machining marks on the workpiece, such as those that occur when the postprocessor outputs points incorrectly. The 3D view of the TNC also provides a measuring function. When 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.
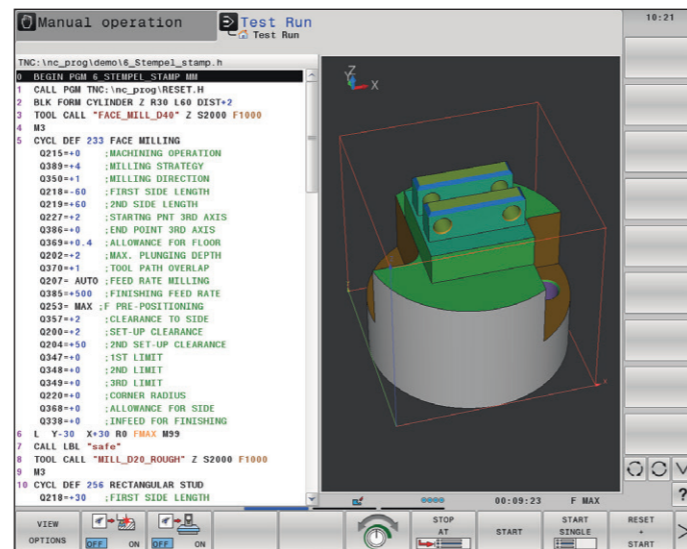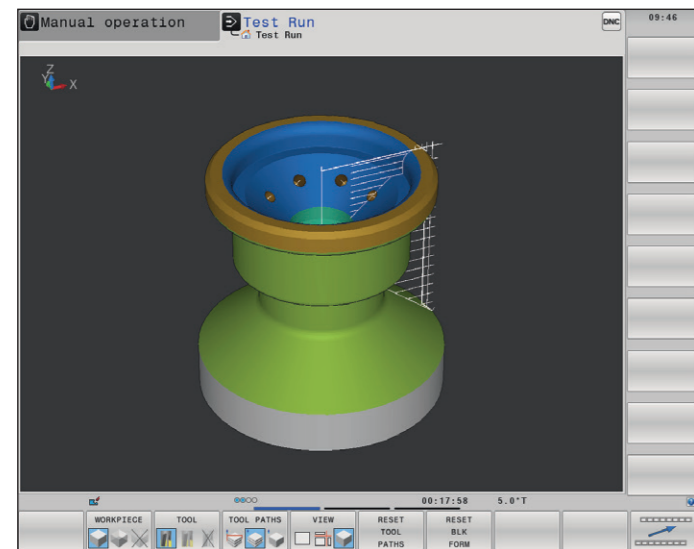
Programming 2D contours

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 select the "linear traverse" path function. 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. Functions 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.
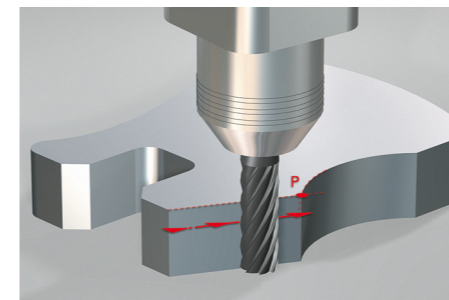
Straight line defined by its end point

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

Chamfer defined by the corner point and chamfer length

Circular path with seamless (tangential) transition from the preceding contour element, defined by its end point
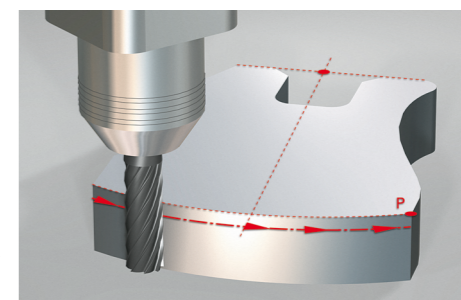
Circular path defined by its center, end point, and rotational direction
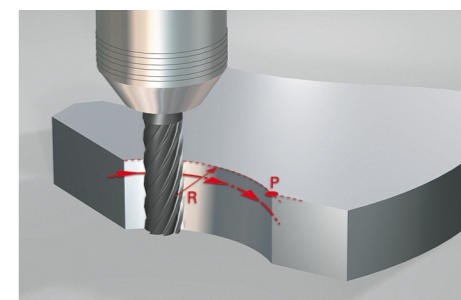
Circular path defined by its radius, end point, and rotational direction

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 allow for more than one mathematical solution, then the TNC 640 programming graphics helpfully render the possible variants to choose from.
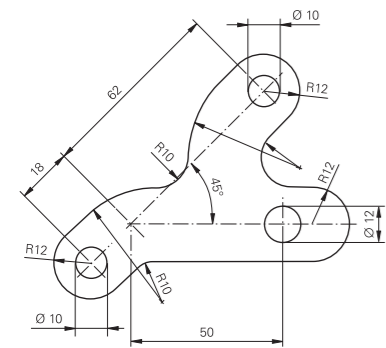
Programming machining patterns with ease and flexibility

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.
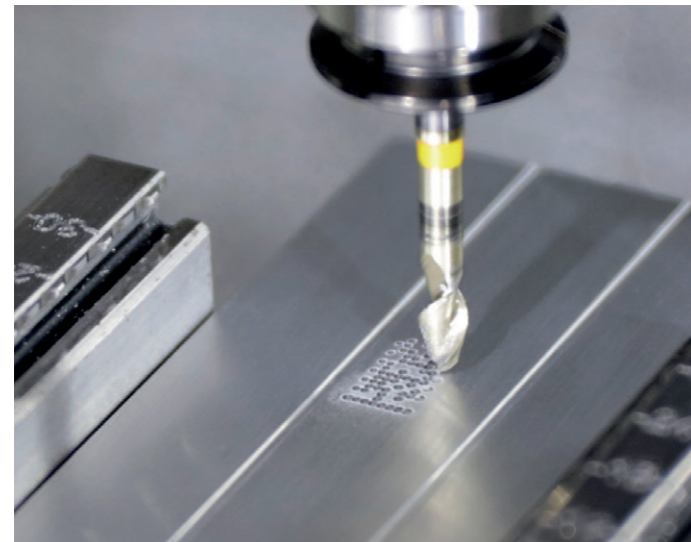
Extensive machining cycles for milling and drilling

The TNC 640 offers you a package covering a broad range of cycles, so you'll surely find the right one for each task. The cycles are divided into groups based on the machining technologies and strategies, ensuring that you always maintain a clear overview. You program them using conversational guidance, assisted by graphics that clearly illustrate all required input parameters.

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 cycles (Optimized Contour Milling). These functions refer to machining cycles for pilot drilling, roughing, and finishing in which the contour or subcontours are defined in subprograms. As a result, a single contour description can be used for various operations 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. You can also program void areas that are excluded from machining. This significantly reduces machining times for cast parts or pre-machined workpieces.
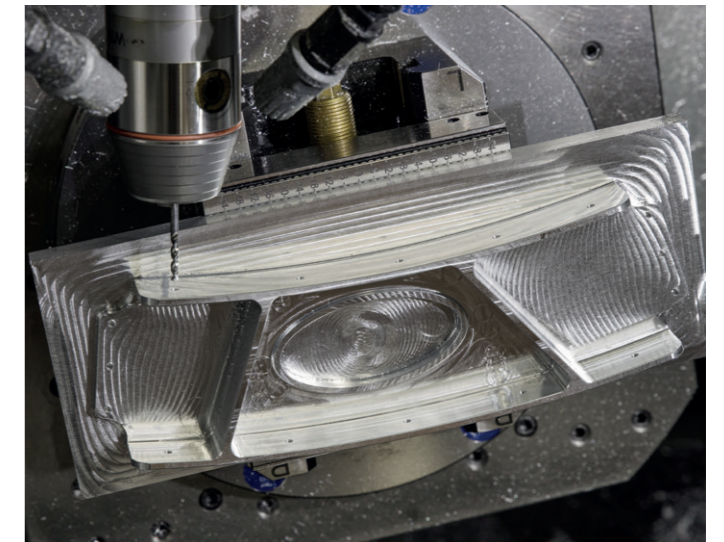
During roughing, a finishing allowance on the wall and floor surfaces is also taken into account by the TNC 640. When roughing with different tools, the control identifies residual material to be removed by smaller tools. A separate cycle is used for finishing to the final dimensions.

OEM cycles

As original equipment manufacturers (OEMs), machine manufacturers can contribute their special manufacturing know-how by designing additional fixed cycles and saving them in the TNC 640. However, the 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 menu key structure of the cycles as desired.

3D machining with parametric programming
Simple, mathematically easy-to-describe 3D géometries 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.

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

In skiving the cutting action is performed through axial feeding of the tool combined with a rolling motion. Skiving is suited for the production of straight or oblique internal and external gear teeth. Significant advantages can be gained by skiving particularly during the production of internal gear teeth, compared with traditional gear shaping. Additionally, the continuous cutting process makes excellent surface definition possible while at the same time reducing process times. With the Skiving cycle you have the possibility of creating a profile program for a custom tooth flank line. For example, you can produce crowning of the tooth flank.

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.

* Requires the Gear Cutting software option (with turning mode, the Turning software option as well)

The TNC 640 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.

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.

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.
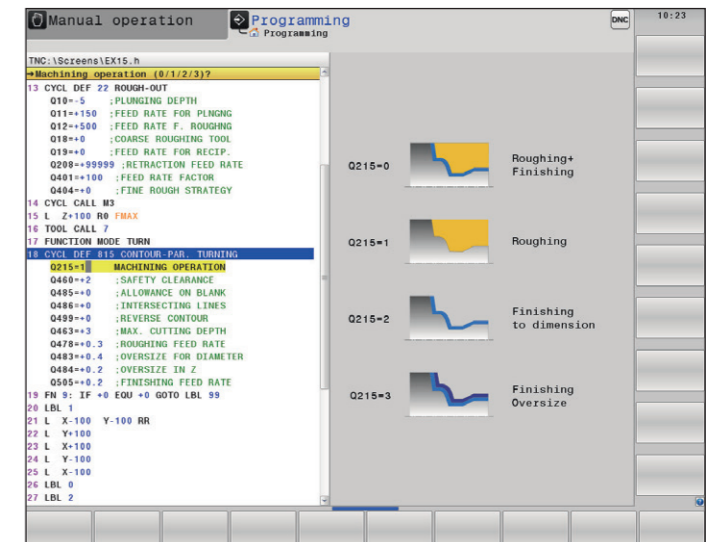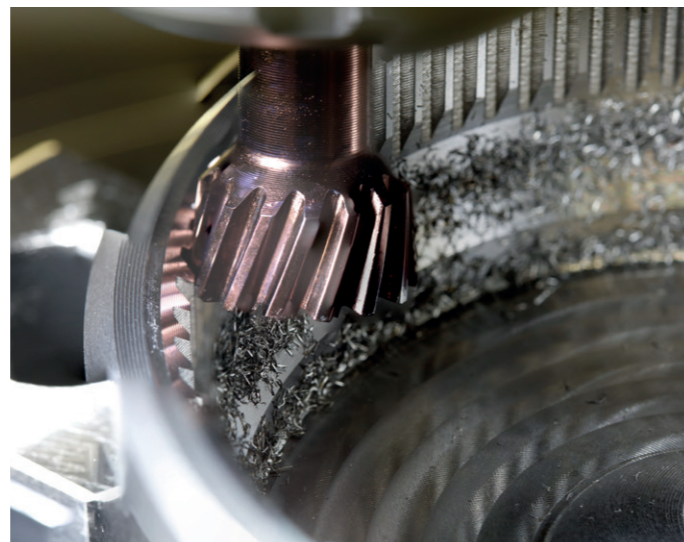
Simultaneous machining

The simultaneous roughing cycle and the simultaneous finishing cycle both adapt the inclination angle of the tool to the workpiece contour while turning. The complex tool movements are monitored in order to avoid collisions between the tool or 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 user 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 any eccentricity with corrective movements of the linear axis coupled with the rotating spindle.

Machining with FreeTurn tools

The TNC 640 supports efficient outside turning operations with FreeTurn tools. FreeTurn tools have different cutting edges for different machining tasks, such as for roughing and finishing. A rotation of the tool axis switches the cutting edges. This reduces the number of tool changes and thus machining times.

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 assists the user during every process, including grinding and dressing. The TNC 640 is therefore the ideal 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 to "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 control must be adapted for these functions by the machine manufacturer.

Reusing programmed contour elements
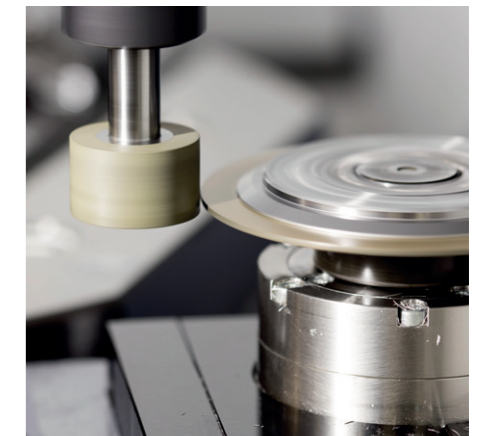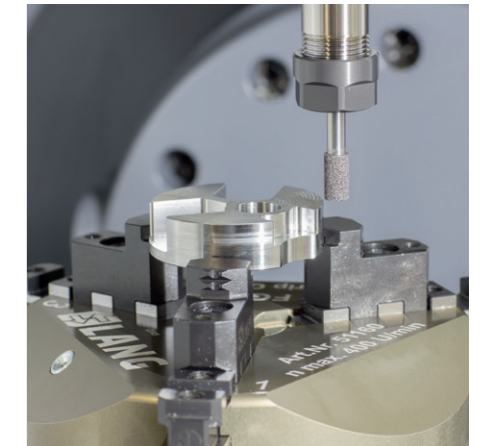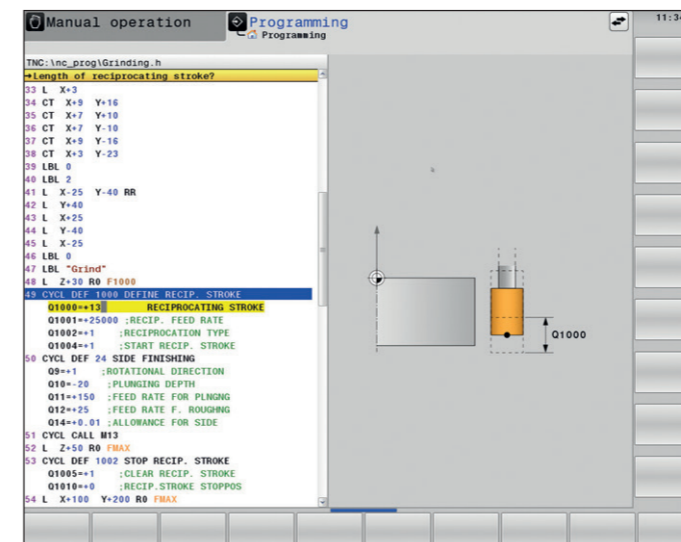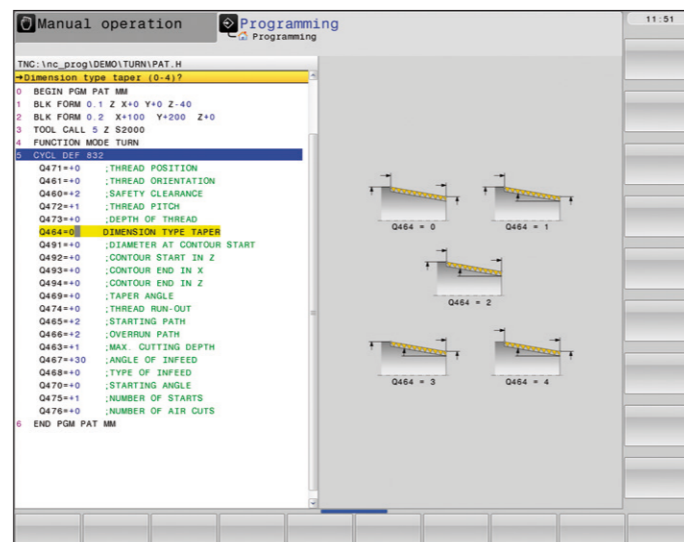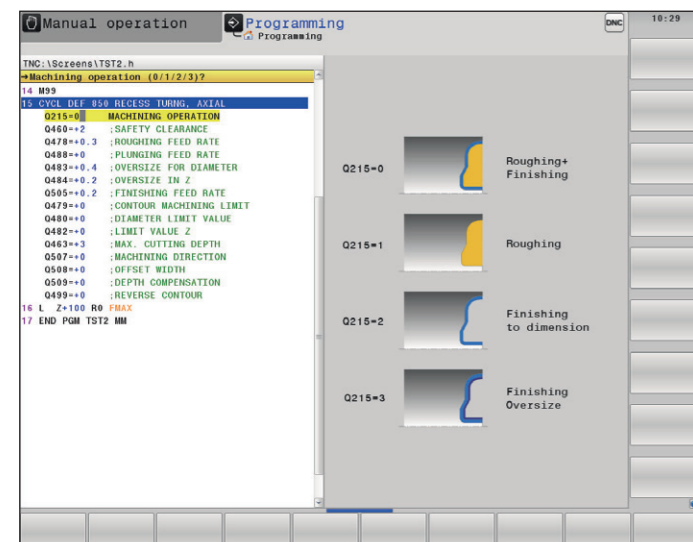
Coordinate transformations

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 directly on the touchscreen 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 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)
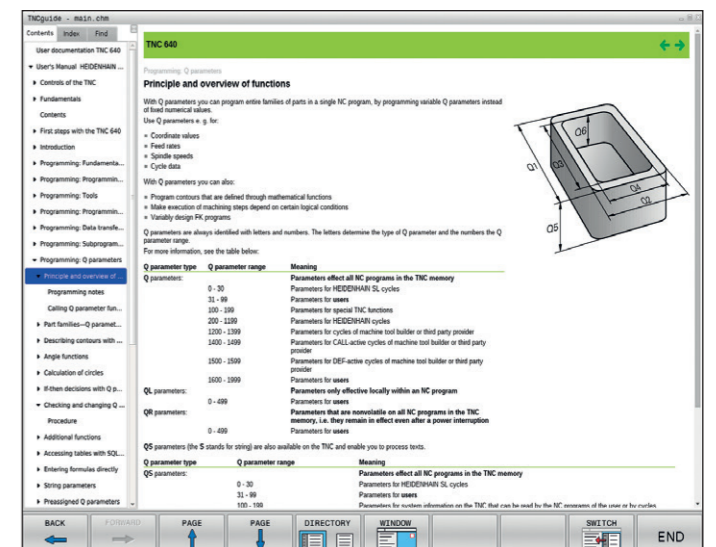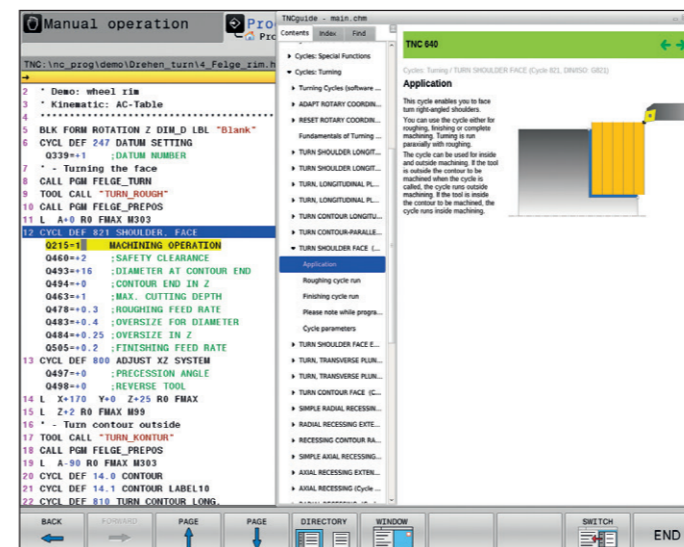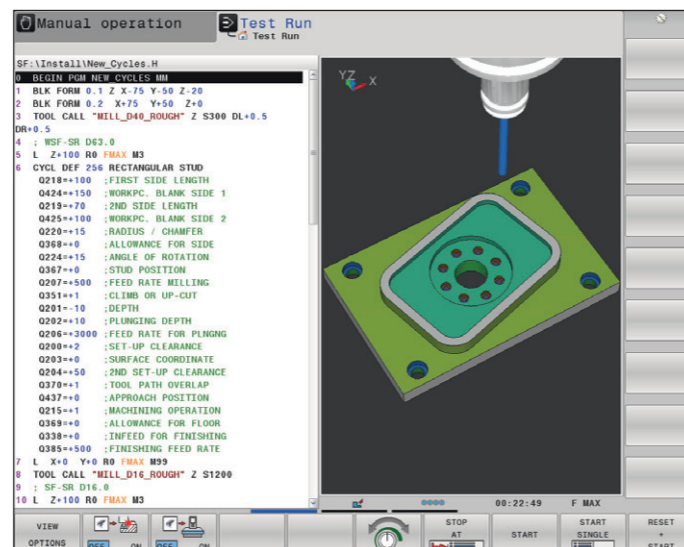
TNCguide integrated into the control, e.g., on the TNC 640 ...

… or on the programming station.

Open for communication
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. CAD Viewer then displays the coordinates of the elements in a window. CAD Viewer can depict the following file formats:
- STEP files (.STP and .STEP)
- IGES files (.IGS and .IGES)
- DXF files (.DXF)
- STL files (.STL)

CAD Import (option)

Why program complex contours if you already have the drawing as a DXF, STEP, STL 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: Start by selecting any element. 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. In this manner, you can define extensive contours with just a few steps. You can then easily copy the selected contour via the clipboard to an existing Klartext program. The contours are output as blocks with linear and circular commands.

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. 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 select 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, including in 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), you can then select the correct intersection.

Generating STL files (option)

The CAD Model Optimizer software option makes it possible to generate STL files from 3D models. The TNC 640 covers the 3D model displayed in CAD Viewer with a mesh of triangles. This simplifies the original model and removes errors, such as small holes in a solid or self-intersections of a surface. The TNC 640 then generates an STL file that you can use for various functions of the control. For example, you can then easily repair faulty files of fixtures or tool holders.
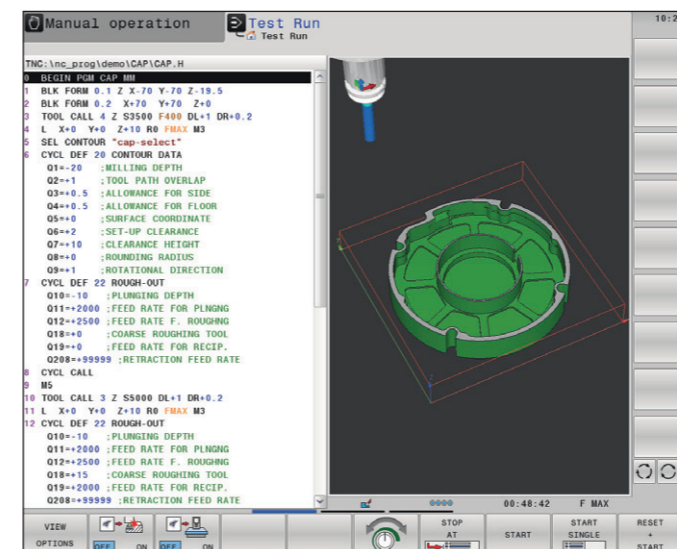
Part program on the basis of the imported DXF file
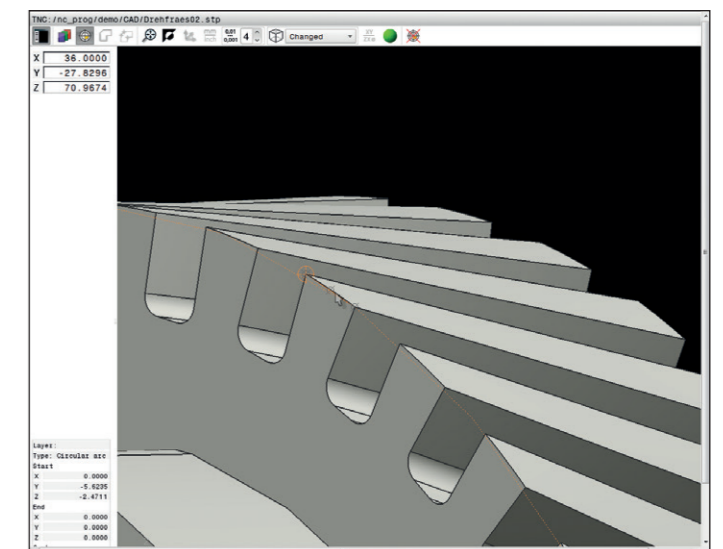
Display of a 3D model in CAD Viewer

connected + machining

Having a 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 users during all shifts. Cost-effective manufacturing therefore requires an efficient process chain and a networked control.

With its Connected Machining package of functions, the TNC 640 control integrates itself flexibly into your process chain and helps you optimize the transfer of knowledge within your company. So let your shop benefit from all of the information available in your company. Connected Machining makes fully digital job management possible in networked manufacturing. You thus benefit from the following:
- Ease of data usage
- Time-saving procedures
- Transparent processes

The networked TNC 640
With the functions of Connected Machining, integrate the TNC 640 into your corporate network, and connect the shop floor via the control to PCs, programming stations, and other data storage media in the following areas:
- Design
- Programming
- Simulation
- Production planning
- Production

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

Standard range of functions
In order to be able to use the data that you have transmitted to the control via your network, the TNC 640 offers several interesting applications, even in its standard range of functions. CAD Viewer, a PDF viewer and the Mozilla Firefox web browser enable the simplest form of Connected Machining: access to manufacturing process data right at the control. In this case, operating web-based documentation systems or ERP systems is just as feasible as accessing your e-mail inbox. For example, the following file formats can also be opened directly on the TNC 640:
- Text and PDF files
- Graphic files with the extensions .gif, .bmp, .jpg or .png
- Spreadsheet files with the extensions .xls, .xlsx, .odv or .csv
- html files: .htm, .html, .chm and several more

Data transmission
Another solution for full digital job management with Connected Machining is the free-of-charge TNCremo PC software. With it, you can transfer remotely stored part programs and pallet tables in both directions over the Ethernet.

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

Job-related data
With the Remote Desktop Manager software option, you can operate a Windows PC from the TNC 640. On the control, you gain direct access to IT 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. This PC can be a computer in the local network or an industrial PC (IPC) in the machine's electrical cabinet.

Detailed data for the optimal organization of the production process
HEIDENHAIN DNC enables the connection of existing machine tools to Windows-based industrial applications, meaning that even older TNC controls, like the TNC 426/430 or the iTNC 530, can be connected to modern inventory management systems and production-activity control systems. Use RemoTools SDK to connect your applications to TNC controls, or purchase an application that is DNC-compatible.

Appropriate monitoring and controlling of applications
Efficient and secure digital communication in a machine-tool context requires standardized components, application-relevant information models and adherence to current IT security guidelines. OPC UA NC Server offers an interface based on OPC UA for HEIDENHAIN controls. This internationally standardized and widespread communication protocol makes it fast and easy to connect machines to your production IT network.

You also save time when integrating new functions because the application-based presentation of information considerably reduces programming and configuration effort.
- *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
- *Expandable by the machine manufacturer:* The machine manufacturer can also enhance the OPC UA NC Server, giving you access to additional sensors, machine subsystems, or values from PLC programs.
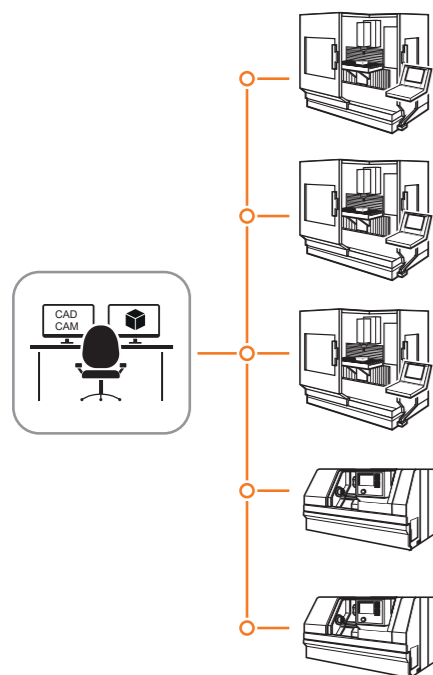
StateMonitor collects and visualizes the statuses of production machines. By evaluating important data such as the current machine status, machine messages, override positions, and utilization history, StateMonitor provides in-depth information about the machine's degree of utilization. Based on the data it collects, StateMonitor also reveals machine-specific optimization potential. The user can enter comments about machine downtimes and setup times in order to uncover machine-specific and organizational optimization potential. Through its messenger function, StateMonitor notifies the appropriate employee by e- mail based on customizable combinations of 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 number of parts 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.

Access over a web browser from various terminals

Real-time notifications via e-mail

Keep an eye on your machines

Collect and visualize job data

Analyze production data in detail

StateMonitor

Enhance your tool management

Plan and log on-demand maintenance

Forwarding to an external SQL database for the processing of machine data in an MES or ERP system

Connect your machines via the HEIDENHAIN DNC, OPC UA, MTConnect, Modbus TCP and FOCAS interfaces.

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 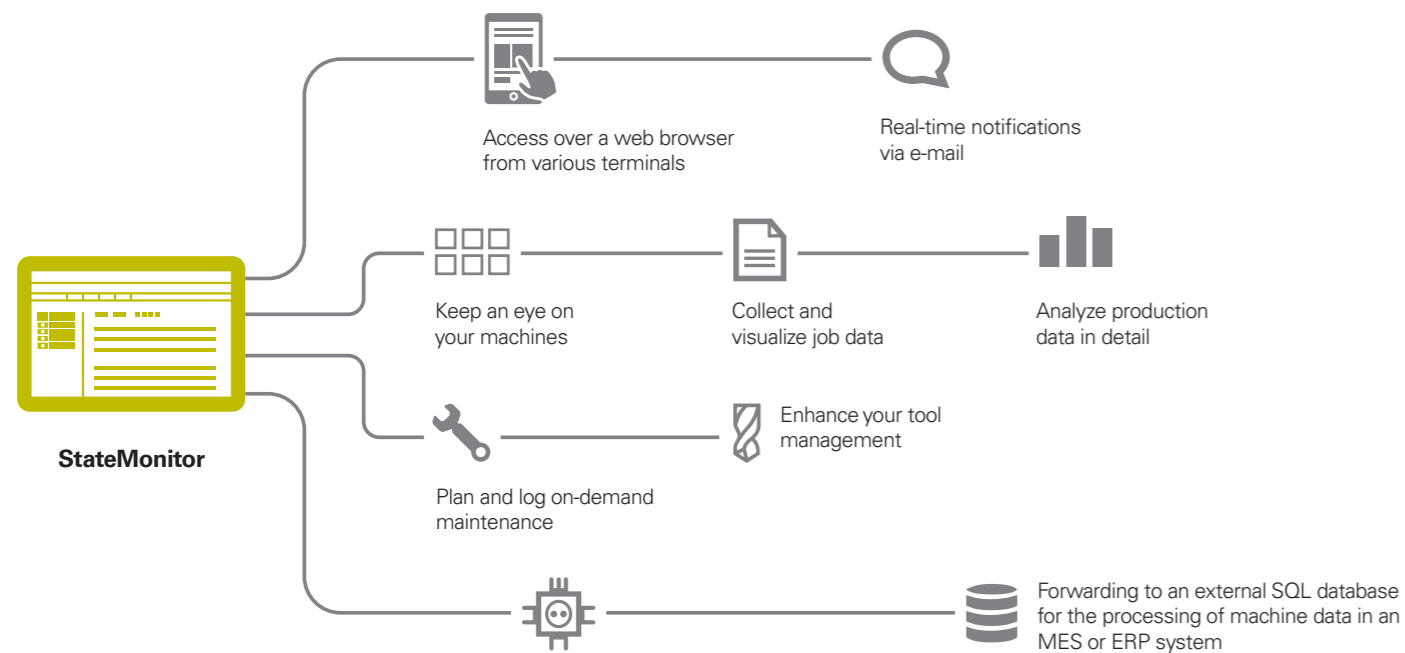station lets you program exactly as you would at the machine, but away from all the noise and distractions of the shop floor.

Creating programs
The programming, testing, and optimizing of HEIDENHAIN Klartext or ISO programs with the programming station substantially reduces machine idle times. This is easy to get used to, since every keystroke feels the same. With the programming station, you program using the same keyboard layout that you use at 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
Because the TNC 640 programming station is based on the same software as the TNC 640 control, it is ideally suited 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.

By being programmable in either Klartext or ISO, the TNC 640 programming station is also well suited for TNC programming training in school settings.

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 functions 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.

Further information:

Comprehensive descriptions of the programming station and a free demo version are available on the Internet at *www.heidenhain.com/programming-stations*.

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 to the control either by cable or by radio or infrared transmission.

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.
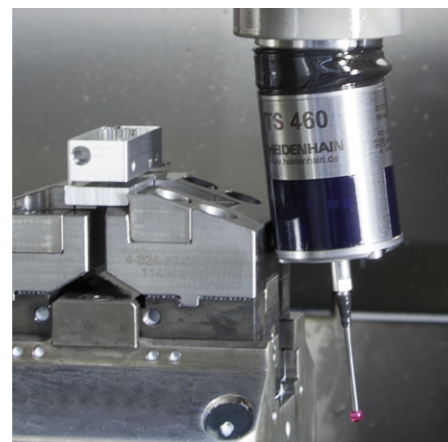
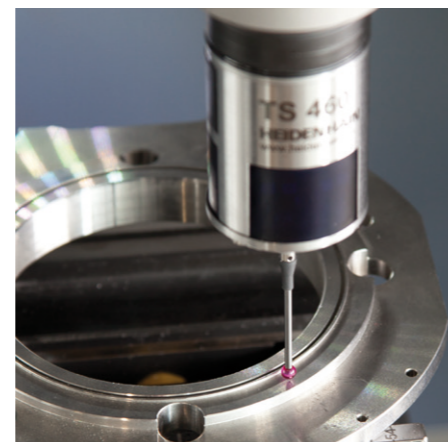* The control must be adapted for these functions by the machine manufacturer.

Cable-bound touch probes
For machines requiring manual tool changes, as well as for grinding machines and lathes:
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 switch in taper shank
• Infrared transmission

TS 760
• High scanning accuracy
• High reproducibility
• Low probing forces
• Radio and infrared transmission

Transceiver
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

TS 460

Further information:
You can find detailed descriptions of workpiece touch probes on the Internet at *www.heidenhain.com* or in the *Touch Probes for Machine Tools* brochure.

TS 460 with collision protection

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 to 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 tool touch probes on the Internet at *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)

Positioning with an electronic handwheel
Fine motion control of axes

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 KinematicsOpt software option for TNC controls is a key ingredient in helping you meet these high demands. First, a cycle automatically measures your machine's rotary axes with a HEIDENHAIN touch probe. 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 reason, HEIDENHAIN offers calibration spheres with highly rigid holders available in various lengths.

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.

Panel-mounted handwheel
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.

Portable handwheels
The HR 510, HR 520, and HR 550 portable handwheels are particularly helpful when you work in close proximity to the machine's working space. The axis keys and certain function keys are integrated in the housing. This lets you set up the machine or switch between the axes you want to move from wherever you happen to be standing with the handwheel. 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
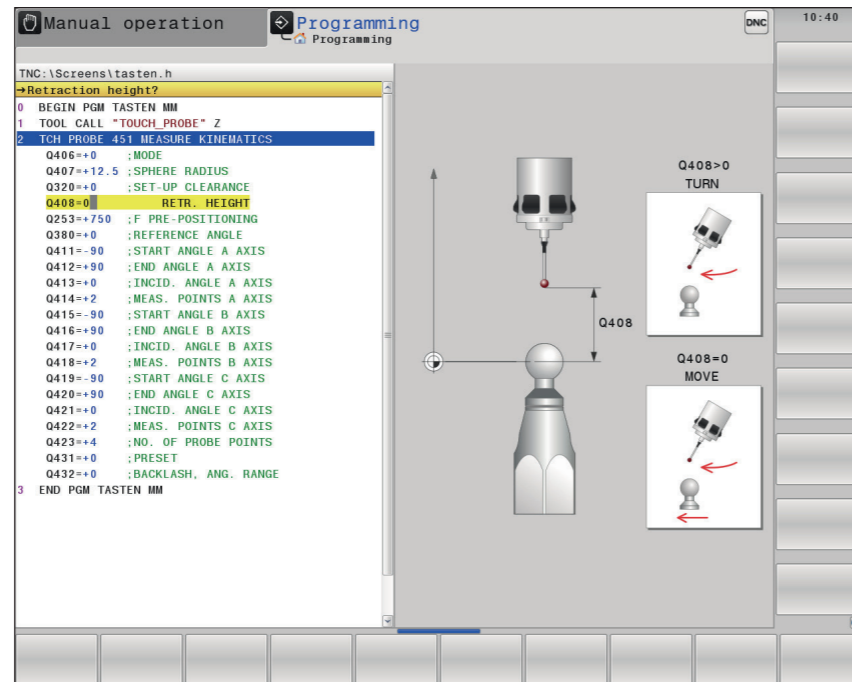
HR 550

Overview
User functions

User function	Standard	Option	
Short description	✓		Basic version: 3 axes plus closed-loop spindle
		0-7 77 78 }	A total of 14 additional NC axes or 13 additional NC axes plus second spindle
	✓		Digital current and speed control
Program entry	✓ ✓		HEIDENHAIN Klartext format
			ISO programming
		42	Importing of contours or machining positions from CAD files (STP, IGS, DXF) and saving them as a Klartext contour program or Klartext point table
Position feedback	✓ ✓ ✓		Nominal positions for lines and arcs in Cartesian coordinates or polar coordinates
			Incremental or absolute dimensions
			Display and entry in mm or inches
Tool compensation	✓ ✓		Tool radius in the working plane, and tool length
			Radius compensated contour look ahead for up to 99 blocks (M120)
		9	Three-dimensional tool-radius compensation for changing tool data without having to recalculate an existing program
Tool tables	✓		Multiple tool tables with any number of tools
Cutting data	✓		Automatic calculation of spindle speed, cutting speed, feed per tooth, and feed per revolution
		167	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
Constant contour speed	✓ ✓		With respect to the path of the tool center
			With respect to the cutting edge
Parallel operation	✓		Program creation with graphical support while another program is running
3D machining	✓		Motion control with highly smoothed jerk
		9	3D tool compensation via surface-normal vectors
		9	Changing the swivel-head angle with the electronic handwheel during program run without affecting the tool tip position (TCPM = Tool Center Point Management)
		9	Keeping the tool perpendicular to the contour
		9	Tool radius compensation perpendicular to the tool direction
		9	Manual traverse in the active tool-axis system
		92	3D radius compensation based on the tool's contact angle
Rotary table machining		8	Programming of cylindrical contours as if in two axes
		8	Feed rate in mm/min
Fixture monitoring		40	Integration of 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	
Turning operations		50/158	Program-controlled switchover between milling and turning
		50/158	Constant cutting speed
		50/158	Cutter radius compensation
		50/158	Cycles for roughing, finishing, recessing, thread cutting, and recess turning
		50/158	Blank-form updating in contour cycles
		50/158	Turning-specific contour elements for recesses and undercuts
		50/158	Orientation of the turning tool for external turning / internal turning
		50/158	Inclined turning
		50/158	Speed limiting
		50/158	Eccentric turning (additionally required: the Synchronizing Functions software option)
		158	Simultaneous turning operations
Contour elements	✓		Straight line
	✓		Chamfer
	✓		Circular path
	✓		Circle center
	✓		Circle radius
	✓		Tangentially connected arc
	✓		Corner rounding
		50/158	Recess
		50/158	Undercut
Contour approach and departure	✓ ✓		Approach and departure on a straight line: tangential or perpendicular
			Approach and departure on an arc
Adaptive feed control		45	AFC adapts the contouring feed rate to the current spindle power
Collision monitoring		40	Dynamic collision monitoring (DCM)
		40	Graphic depiction of the active collision objects
		40	Tool carrier monitoring
Free contour programming	✓		FK free contour programming in HEIDENHAIN Klartext format with graphical support for workpiece drawings not dimensioned for NC
Program jumps	✓		Subprograms
	✓		Program section repeat
	✓		Any desired program as subprogram

User function	Standard	Option	
Machining cycles	✓		Drilling, tapping (with or without floating tap holder), rectangular and circular pockets
	✓		Peck drilling, reaming, boring, counterboring, centering
		50/158	Area clearance cycles, longitudinal and transverse, paraxial and contour-parallel
		50/158	Recessing cycles, radial/axial
		50/158	Radial/axial recess turning cycles (combined recessing and roughing motion)
	✓		Milling internal and external threads
		50/158	Turning of internal and external threads
		50/158	Hobbing (additionally required: the Spindle Synchronism software option)
		96	Interpolation turning
		156	Function for grinding operation
		157	Gear-system production
	✓		Clearing level and oblique surfaces
	✓		Complete machining of rectangular and circular pockets, rectangular and circular studs
	✓		Multi-operation machining of straight and circular slots
	✓		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
	✓		OEM cycles (special cycles developed by the machine manufacturer) can be integrated
		167	Optimized Contour Milling (OCM) cycles: optimization of roughing processes
Coordinate transformation	✓		Shifting, rotating, mirroring, scaling (axis-specific)
		8	Tilting the working plane, PLANE function
		44	Manually definable: shifts, rotations, and handwheel superimpositioning can be manually defined via global program settings
Q parameters Programming with variables	✓		Mathematical functions $=, +, -, *, /, \sin \alpha, \cos \alpha, \tan \alpha$, arc sin, arc cos, arc tan, a^n, e^n, In, log, $\sqrt{a}$, $\sqrt{a^2 + b^2}$
	✓		Logical operations $(=, \neq, <, >)$
	✓		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
Programming aids	✓		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
	✓		Graphic support for the programming of cycles
	✓		Comment and structure blocks in the NC program
Teach-In	✓		Actual positions can be transferred directly to the NC program
Program verification graphics 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 enlargement
Programming graphics	✓		In the Programming mode, the contours of entered NC blocks are rendered (2D pencil-trace graphics), even while another program is running
Program-run graphics Display modes	✓		Real-time graphical simulation during execution of the milling program
	✓		Plan view / view in three planes / 3D view
Machining time	✓		Calculation of machining time in the Test Run mode of operation
	✓		Display of the current machining time in the program run

User function	Standard	Option	
Returning to the contour	✓		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
Preset management	✓		One table for saving any reference points (presets)
Datum tables	✓		Multiple datum tables for storing workpiece-specific datums
Pallet tables	✓		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
Touch probe cycles	✓		Touch probe calibration
	✓		Manual or automatic compensation of workpiece misalignment
	✓		Manual or automatic preset setting
	✓		Automatic tool and workpiece measurement
		48	KinematicsOpt: automatic measurement and optimization of the machine's kinematic model
		50/158	Cycle for measurement of turning tools
Parallel secondary axes	✓		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
Conversational languages	✓		English, German, Czech, French, Italian, Spanish, Portuguese, Swedish, Danish, Finnish, Dutch, Polish, Hungarian, Russian (Cyrillic), Chinese (traditional, simplified), Slovenian, Slovak, Norwegian, Korean, Turkish, Romanian
CAD Viewer	✓		Display of standardized CAD file formats on the TNC

Accessories	
Electronic handwheels	• **HR 510/HR 520** portable handwheels • **HR 550** portable wireless handwheel • **HR 130** panel-mounted handwheel
Workpiece measurement	• **TS 260** workpiece touch probe with cable connection • **TS 460/TS 760** workpiece touch probes with radio or infrared transmission • **TS 642** workpiece touch probe with infrared transmission
Tool measurement	• **TT 160** triggering tool touch probe • **TT 460** triggering tool touch probe with radio or infrared transmission
Programming station	Control software for PCs for programming, archiving, and training • Single-station license with original control keyboard/with operation via virtual keyboard • Network license with operation via virtual keyboard • Demo version (operated with PC keyboard—free of charge)
Software for PCs	• **RemoteAccess** for remote diagnostics, monitoring, and operation • **CycleDesign** for creating your own cycle structure • **TNCremo** for data transfer—free of charge • **TNCremoPlus** for data transfer with live-screen function • **StateMonitor** for recording, evaluating, and visualizing machine data

Software options

Option number	Option	With NC software 34059x- or later	
0 to 7	Control Loop Qty.	01	Additional control loop
8	Adv. Function Set 1	01	**Rotary table machining** • Programming of cylindrical contours as if in two axes • Feed rate in mm/min **Interpolation:** circular in three axes with tilted working plane **Coordinate conversion:** tilting the working plane, and the PLANE function
9	Adv. Function Set 2	01	**Interpolation:** straight line in more than four axes (export license required) **3D machining** • 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
18	HEIDENHAIN DNC	01	Communication with external PC applications over COM component
40	Collision Monitoring	02	Dynamic collision monitoring (DCM): For defining machine components as collision objects. The TNC 640 monitors the defined collision objects during all machine movements. • Graphic depiction of the active collision objects (high-resolution M3D format) • Tool carrier monitoring • Fixture monitoring
42	CAD Import	08	Importing of contours from 3D and 2D models (e.g., STEP, IGES, DXF)
44	Global PGM Settings	08	Global program settings
45	Adaptive Feed Contr.	02	Adaptive feed control (AFC)
46	Python OEM Process	01	Execution of Python applications
48	KinematicsOpt	01	Touch-probe cycles for the automatic measurement of rotary axes
49	Double Speed Axes	01	Short control-loop cycle times for a direct drive motor
50	Turning	01	Turning functions: • Tool management for turning • Tool radius compensation • Switching between milling and turning mode • Turning-specific contour elements • Turning cycle package
52	KinematicsComp	05	Compensation of spatial errors for rotary and linear axes (export license required)
56 to 61	OPC UA NC Server	10	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

Option number	Option	With NC software 34059x- or later	
77	4 Additional Axes	01	Four additional control loops
78	8 Additional Axes	01	Eight additional control loops
92	3D-Tool Comp	07	3D radius compensation based on the contact angle (only with the Advanced Function Set 2 software option)
93	Ext. Tool Management	01	Expanded tool management • Tooling list (list of all tools of the NC program) • T usage sequence (sequence of all tools inserted during the program)
96	Adv. Spindle Interpol.	05	Additional function for an interpolated spindle • Interpolation turning, coupling • Interpolation turning, contour finishing
101 to 130	OEM Option	02	Options of the machine manufacturer
131	Spindle Synchronism	05	Synchronization of two or more spindles
133	Remote Desk. Manager	01	Display and remote operation of external computer units (e.g., a Windows PC)
135	Synchronizing Functions	04	Advanced synchronization of axes and spindles
141	Cross Talk Comp.	02	CTC: compensation of axis couplings
142	Position Adapt. Contr.	02	PAC: position-dependent adaptation of the control parameters
143	Load Adapt. Contr.	02	LAC: load-dependent adaptation of the control parameters
144	Motion Adapt. Contr.	02	MAC: motion-dependent adaptation of the control parameters
145	Active Chatter Contr.	02	ACC: active suppression of chatter during heavy machining
146	Machine Vibr. Contr.	04	Damping of machine oscillations to improve workpiece surfaces. The following functions are part of Machine Vibration Control (MVC): • Active Vibration Damping (AVD): active damping of vibrations in the control loop • Frequency Shaping Control (FSC): reduction of vibration inducement by means of frequency-based feedforward control
152	CAD Model Optimizer	16	Conversion and optimization of CAD models • Fixtures • Workpiece blank • Finished part
154	Batch Process Mngr.	08	Batch Process Manager for easy planning and execution of multiple production jobs
155	Component Monitoring	09	Monitoring for component overloading and wear
156	Grinding	10	Grinding functions: • Jig grinding • Ability to switch between normal operation and dressing mode • Reciprocating stroke • Grinding cycles • Tool management for grinding and dressing
157	Gear Cutting	09	Functions for the machining of gear teeth

Software options

Option number	Option	With NC software 34059x- or later	
158	Turning v2	17	Turning functions (mill-turning version 2) • Includes all functions of the Turning software option plus cycles for simultaneous roughing and finishing
160	Integrated FS: Basic	10	Gen 3 exclusive: • Enables 4 safe control loops • Enabling of functional safety (FS)
161	Integrated FS: Full	10	Gen 3 exclusive: • Enables the maximum number of safe control loops • Enabling of functional safety (FS)
162 to 166	FS Control Loop Qty.	10	Additional safe control loop 1 to 5
167	Opt. Contour Milling	10	OCM: optimize roughing processes and fully utilize milling tools with the integrated cutting data calculator
169	FS Control Loop Qty.	11	Enabling of all FS axis options or control loops. The software options Integrated FS: Basic as well as FS Control Loop Qty. (162 to 166) must already be set.

Specifications

Specifications	Standard	Option	

Components	✓
	✓
	✓
	
	✓
			MC main computer
			CC or UxC controller unit
			Multitouch operation
			    19-inch screen (portrait, 1024x1280 pixels)
			    19-inch screen (1280x1024 pixels)
			    24-inch screen (1920x1080 pixels)
			TE operating panel (suitable for 24-inch screen)

Operating system	✓		HEROS 5 real-time operating system for machine control
NC program memory	✓		• HDR: 285 GB (total capacity: 320 GB) • SSDR: 17 GB (total capacity: 32 GB)
Input resolution and display step	✓ ✓		Linear axes: down to 0.01 µm Rotary axes: down to 0.00001°
Interpolation	✓ ✓ ✓	 9 8	Linear in 4 axes Linear in up to 6 axes (export license required) Circular in 2 axes Circular in 3 axes with tilted working plane Helical: superimposition of circular and straight paths
Block processing time	✓		≤ 0.5 ms (3D straight line without radius compensation)
Axis feedback control	✓ ✓ ✓ ✓		Position-loop resolution: signal period of the position encoder/4096 Cycle time of position controller: 200 µs (100 µs with the Double Speed Axes software option) Cycle time of speed controller: 200 µs (100 µs with the Double Speed Axes software option) Cycle time of current controller: minimum 100 µs (minimum 50 µs with the Double Speed Axes software option)
Error compensation	✓ ✓		Linear and nonlinear axis error, backlash, reversal peaks during circular movements, reversal error, thermal expansion Static friction, sliding friction
Data interfaces	✓ ✓ ✓ 	 18 56-61	Network interface for remote operation of the TNC using the TNCremo or TNCremoPlus software from HEIDENHAIN 2 x Ethernet interface, 1 Gigabit USB 3.0 (and one USB 2.0 on the operating panel); the number of ports depends on the hardware used HEIDENHAIN-DNC for communication between a Windows application and the TNC (DCOM interface) OPC UA NC Server Secure and reliable interface for connecting leading-edge industrial applications
Diagnostics	✓		Fast and simple troubleshooting through integrated diagnostic aids
Ambient temperature	✓ ✓		Operation: +5 °C to +40 °C Storage: −20 °C to +60 °C

Comparison of TNC 620 and TNC 640

Control		TNC 620 NC software 81760x-18	TNC 640 NC software 34059x-18
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
Program entry			
	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

NC program memory		CFR: 7.7 GB
		(total capacity: 30 GB)
			• HDR: 285 GB
			  (total capacity: 320 GB)
			• SSDR: 17 GB
			  (total capacity: 32 GB)

5-axis and high-speed machining		Option	Option
Block processing time		1.5 ms	< 0.5 ms
Input resolution and display step (standard)		0.01 µm	0.01 µm
Screen and keyboard		15-inch touchscreen/ 19-inch touchscreen (portrait)	19-inch / 24-inch touchscreen
Touchscreen		✓	✓
Adaptive Feed Control (AFC)		–	Option

Control		TNC 620 NC software 81760x-18	TNC 640 NC software 34059x-18
Area of application		**Standard milling**	**High-end milling/turning/grinding**
Active Chatter Control (ACC)		Option	Option
Dynamic Collision Monitoring (DCM)		–	Option
	Fixture monitoring	–	Option
KinematicsOpt		Option	Option
KinematicsComp		–	Option
Touch probe cycles		Option	✓
Pallet management		✓	✓
Parallel axis function		✓	✓

✓ Standard
– Not available

HEIDENHAIN
Mastering nanometer accuracy

HEIDENHAIN
worldwide