



Options and Accessories For TNC Controls

HEIDENHAIN controls are known for their complete range of features and extensive functionality. Thanks to a series of control options and accessories, they can also be optimally adapted to the given application. This brochure provides a broad overview of the available options and the most important hardware enhancements for the current control versions. Along with a detailed description of its functionality, each option or hardware enhancement contains a list of the software versions with which it is available.

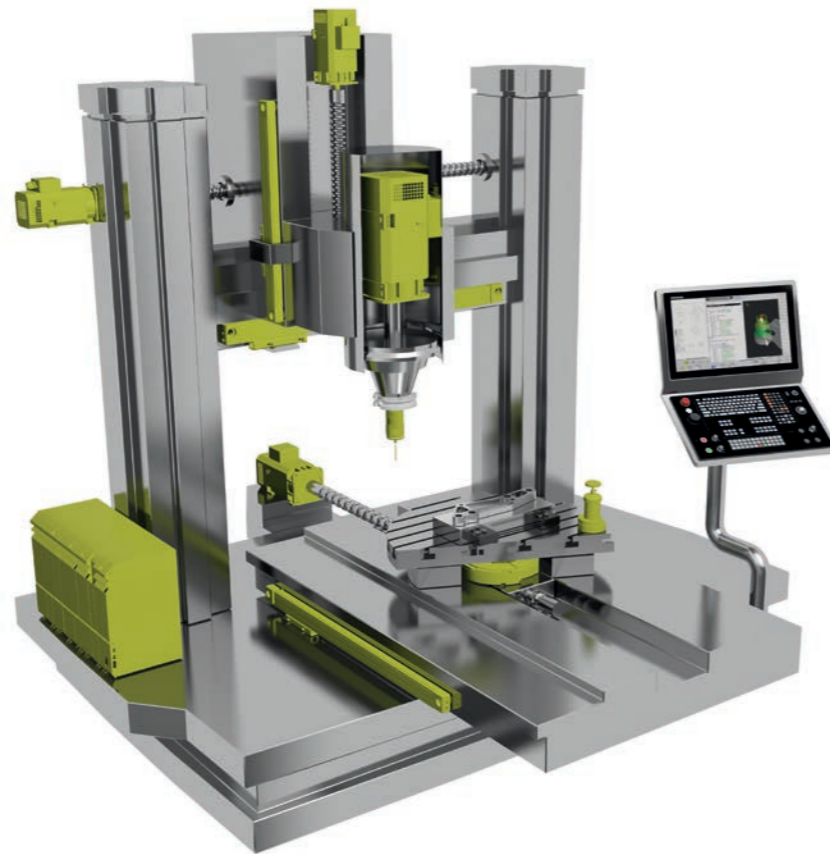
Options are functions already integrated into the control, allowing you to tailor the TNC's functionality to your specific needs at any time. Some options, however, must be adapted by the machine manufacturer. Options are conveniently enabled via a keyword.

HEIDENHAIN offers **software** as practical tools for applications external to the TNC, from supporting data transfer or creating a PLC program all the way to a complete programming station.

Hardware enhancements make work on the machine easier, faster, and more reliable. An electronic handwheel, for example, permits fine manual control, and a workpiece touch probe significantly reduces workpiece setup times.

The version schema was simplified, starting with NC software version 16:

- The publication period determines the version number.
- All control models of a publication period have the same version number.
- The version number of the programming stations corresponds to the version number of the NC software.



Overview	Overview tables	4
	Dynamic Precision	10
	Dynamic Efficiency	11
	Connected Machining	12
	Digital Shop Floor	13
Options	Programming and operation	14
	Machine accuracy	34
	Machining functions	41
	Communication	53
	Adaptation to the machine	56
Software		63
Hardware enhancements		78

Please also note the page references in the overview tables.

Overview

Option number	Option	TNC 320	TNC 620	TNC 640	TNC7	Must be adapted by the OEM	ID	Page
Machining functions								
8	Advanced Function Set 1: rotary table machining <ul style="list-style-type: none"> Programming cylindrical contours on an unrolled cylinder Feed rate in mm/min or degrees/min 	•	•	•	•	Yes	617920-01 TNC 320: 536164-01	14
	Advanced Function Set 1: coordinate transformation Tilting the working plane, PLANE function	•	•	•	•	Yes	617920-01 TNC 320: 536164-01	15
	Advanced Function Set 1: interpolation, circular in 3 axes with tilted working plane	•	•	•	•	No	617920-01 TNC 320: 536164-01	15
9	Advanced Function Set 2: interpolation, linear in 5 axes	–	•	•	•	No	617921-01	–
	Advanced Function Set 2: 5-axis simultaneous machining <ul style="list-style-type: none"> 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 	–	•	•	•	Yes	617921-01	41
17	Touch Probe Functions: probing cycles <ul style="list-style-type: none"> Workpiece misalignment compensation, preset setting Automatic tool and workpiece measurement Touch-probe input enabling for non-HEIDENHAIN systems 	✓	•	✓	✓	Yes	634063-01	16
19	Advanced Programming Features: expanded programming functions <ul style="list-style-type: none"> FK free contour programming Canned cycles Peck drilling, reaming, boring, counterboring, centering Milling internal and external threads Clearing level and oblique surfaces Multi-operation machining of straight and circular slots Complete machining of rectangular and circular pockets Circular and linear point patterns Contour train, contour pocket, including contour-parallel machining Integration of cycles created by machine manufacturer Engraving cycle: engrave text or numbers in a straight line or on an arc Contour slot with trochoidal milling 	✓	•	✓	✓	No	628252-01	18
20	Advanced Graphic Features: verification graphics, program run graphics <ul style="list-style-type: none"> Plan view Projection in three planes 3D view 	✓	•	✓	✓	No	628253-01	19
	Advanced Graphic Features: detailed 3D view	✓	•	✓	✓	No	628253-01	20

• = Available as option
– = Not available
✓ = Standard function

Option number	Option	TNC 320	TNC 620	TNC 640	TNC7	Must be adapted by the OEM	ID	Page
Machining functions								
21	Advanced Function Set 3: handwheel superimpositioning, handwheel positioning during program run	✓	•	✓	✓	Yes	628254-01	42
	Advanced Function Set 3: tool compensation, pre-calculating radius-compensated contours (LOOK AHEAD)	✓	•	✓	✓	No	628254-01	43
22	Pallet Management	–	•	✓	✓	Yes	628255-01	21
40	Dynamic Collision Monitoring (DCM)	–	–	•	•	Yes	526452-01	44
42	CAD Import: DXF converter, importing contours and machining positions from DXF files	•	•	•	•	No	526450-01	23
	CAD Import: importing contours from 3D models	•	•	•	•	No	526450-01	24
44	Global PGM Settings: global program settings	–	–	•	•	Yes	576057-01	46
45	Adaptive Feed Control (AFC)	–	–	•	•	Yes	579648-01	47
50	Turning: turning functions <ul style="list-style-type: none"> Turning tool management Tool radius compensation Switching between milling and turning mode Turning-specific contour elements Package of turning cycles 	–	–	•	•	Yes	634608-01	26
	Turning: eccentric turning	–	–	•	•	Yes	634608-01	27
92	3D-ToolComp: 3D radius compensation based on tool inclination (only in conjunction with option 9)	–	–	•	•	No	679678-01	52
93	Extended Tool Management	•	•	•	•	Yes	676938-01	29
96	Advanced Spindle Interpolation: interpolating spindle, interpolation turning	–	–	•	•	Yes	751653-01	30
131 50	Spindle Synchronism: spindle synchronism of two or more spindles <ul style="list-style-type: none"> Hobbing cycle 	–	–	•	•	Yes	806270-01	31
140	DCM v2: fixture measurement (automatically enables option 40)	–	–	–	•	Yes	1353266-01	48
145	Active Chatter Control (ACC)	–	•	•	•	Yes	800547-01	48
152	CAD Model Optimizer <ul style="list-style-type: none"> Generate correct STL files from STEP files Repair existing STL files 	–	•	•	•	No	1353918-01	25
154	Batch Process Manager: convenient interface for pallet management	–	•	•	•	Yes	1219521-01	22
156	Grinding: grinding and dressing functions	–	–	•	•	Yes	1237232-01	32

• = Available as option
– = Not available
✓ = Standard function

Overview

Option number	Option	TNC 320	TNC 620	TNC 640	TNC7	Must be adapted by the OEM	ID	Page
Machining functions								
157	Gear Cutting: function for gear manufacturing	-	-	•	•	Yes	1237235-01	33
158	Turning v2 Extended turning cycles and functions (automatically enables option 50)	-	-	•	•	Yes	1359635-01	28
159	Model Aided Setup: graphically supported setup of workpieces	-	-	-	•	No	1364052-01	17
167	Optimized Contour Milling (OCM): optimize roughing processes	-	•	•	•	Yes	1289547-01	49
Machine accuracy								
48	KinematicsOpt: touch probe cycles for automatic measurement of rotary axes	-	•	•	•	Yes	630916-01	34
52	KinematicsComp: 3D spatial compensation	-	-	•	•	Yes	661879-01	35
141	Cross Talk Compensation (CTC): compensation of position errors through axis coupling	-	•	•	•	Yes	800542-01	36
142	Position Adaptive Control (PAC): position-dependent adaptation of control parameters	-	•	•	•	Yes	800544-01	37
143	Load Adaptive Control (LAC): load-dependent adaptation of control parameters	-	•	•	•	Yes	800545-01	38
144	Motion Adaptive Control (MAC): motion-dependent adaptation of control parameters	-	•	•	•	Yes	800546-01	39
146	Machine Vibration Control (MVC): active vibration damping	-	•	•	•	Yes	800548-01	40
155	Component Monitoring: monitoring for component overload and wear	-	•	•	•	Yes	1226833-01	50
168	Process Monitoring: reference-based monitoring of the machining process	-	-	-	•	Yes	1302488-01	51
Communication								
18	HEIDENHAIN DNC: communication with external Windows applications via COM components	•	•	•	•	No	526451-01	54
56 - 61	OPC UA NC Server 1 to 6: standardized interface for access to data and functions of the control	-	•	•	•	Yes	1291434-01 - 1291434-06	53
133	Remote Desktop Manager: display and remote operation of external computers (e.g., Windows PC)	•	•	•	•	Yes	894423-01	55

• = Available as option
- = Not available
✓ = Standard function

Option number	Option	TNC 320	TNC 620	TNC 640	TNC7	Must be adapted by the OEM	ID	Page
Adaptation to the machine								
0	Additional Axis 1 to 8: additional control loops	•	•	•	•	Yes	354540-01	56
1		•	•	•	•	Yes	353904-01	
2		-	•	•	•	Yes	353905-01	
3		-	•	•	•	Yes	367867-01	
4		-	-	•	•	Yes	367868-01	
5		-	-	•	•	Yes	370291-01	
6		-	-	•	•	Yes	370292-01	
7		-	-	•	•	Yes	370293-01	
24	Gantry Axes: synchronized axes; gantry axes, tandem tables	•	•	✓	✓	Yes	634621-01	58
46	Python OEM Process: python applications; realize special functions	•	•	•	•	Yes	579650-01	59
49	Double Speed Axes: double speed control loops; short control loop cycle times for direct drive motors	-	•	•	•	Yes	632223-01	60
77	4 Additional Axes: 4 additional control loops	-	-	•	•	Yes	634613-01	56
78	8 Additional Axes: 8 additional control loops	-	-	•	•	Yes	634614-01	
101 - 130	OEM Option	-	-	•	-	Yes	579651-01 - 579651-30	61
135	Synchronizing Functions: RTC, real-time coupling function for synchronizing axes and spindles	-	-	•	•	Yes	1085731-01	62
160	Integrated FS: Basic (only for Gen 3): enabling of functional safety (FS) and four safe control loops	-	•	•	•	Yes	1249928-01	57
161	Integrated FS: Full (only for Gen 3): enabling of functional safety (FS) and maximum number of safe control loops (≥ 10)	-	•	•	•	Yes	1249929-01	
162	Add. FS Ctrl. Loop 1 (only for Gen 3): additional safe control loop 1	-	•	•	•	Yes	1249930-01	
163	Add. FS Ctrl. Loop 2 (only for Gen 3): additional safe control loop 2	-	•	•	•	Yes	1249931-01	
164	Add. FS Ctrl. Loop 3 (only for Gen 3): additional safe control loop 3	-	•	•	•	Yes	1249932-01	
165	Add. FS Ctrl. Loop 4 (only for Gen 3): additional safe control loop 4	-	•	•	•	Yes	1249933-01	
166	Add. FS Ctrl. Loop 5 (only for Gen 3): additional safe control loop 5	-	•	•	•	Yes	1249934-01	
169	Add. FS Full (only for Gen 3): enable remaining safe control loops	-	•	•	•	Yes	1319091-01	

• = Available as option
- = Not available
✓ = Standard function

Overview

Software	TNC 320	TNC 620	TNC 640	TNC7	Subject to a charge	Page
ConfigDesign : configuration of the machine parameters	•	•	•	•		–
PCtoolsUpdateManager : update manager for HEIDENHAIN software	•	•	•	•		–
PLCtext : processing and managing of language-sensitive texts	•	•	•	•		–
TNCAnalyzer : analysis and evaluation of service files	•	•	•	•		–
TNC OEMOption : key generator for enabling options for a limited time	•	•	•	•		–
TNC OEMNumber : day code generator for the machine manufacturer area	•	•	•	•		–
TNCtest : creation and execution of acceptance tests	•	•	•	•		–
RemoTools SDK : function library for the development of one's own communication applications (option 18 required)	•	•	•	•	•	54
TNCremo : data transfer software	•	•	•	•		63
TNCremoPlus : data transfer software with "live" screen	•	•	•	•	•	63
RemoteAccess : software for remote diagnostics	•	•	•	•	•	64
PLCdesign : PLC diagnostics for all HEIDENHAIN controls	•	•	•	•		65
StateMonitor	•	•	•	•	•	66
KinematicsDesign : creation of control kinematic models	•	•	•	•		68
M3D Converter : creation of high-resolution collision bodies	•	•	•	•	•	68
CycleDesign : incorporation of developed NC subprograms as cycles	•	•	•	•		69
TNCscope : recording and evaluation of, for example, axis curves of HEIDENHAIN controls	•	•	•	•		70
TNCdiag : evaluation of diagnostic information of digital motor systems	–	•	•	•		71
TNCopt : initial setup of digital control loops	–	•	•	•		72
IOconfig : configuration of, for example, handwheels	•	•	•	•		73
TNCkeygen : contains TNC OEMOption and TNC OEMNumber	•	•	•	•		74
BMXdesign : designing language-sensitive soft keys	•	•	•	•		75
Programming station	•	•	•	–		76
virtualTNC : for PC-based machine simulation	–	•	•	–	•	77

• = Available as option
 – = Not available
 ✓ = Standard function

Hardware enhancements	TNC 320	TNC 620	TNC 640	TNC7	Must be adapted by the OEM	ID	Page												
Handwheel	HR 130 TTL; panel mounted With detent W/o detent	•	•	•	•	No	540940-01 540940-03	78											
						HR 510 portable handwheel	•		•	•	•	No	1120313-xx 1119971-xx						
												HR 510FS portable handwheel	•	•	•	•	No	1119974-xx 1120311-xx	
																	HR 520 portable handwheel with display	•	•
												HR 520FS portable handwheel with display	•	•	•	•			
																	HR 550FS portable radio handwheel with display	•	•
												HRA 551FS handwheel holder for HR 550FS	•	•	•	•			
Workpiece touch probe	TS 260 with cable	•	•	•	•	Yes	738283-xx	79											
						TS 460 with radio or infrared transmission	•		•	•	•	Yes	1178530-xx						
												TS 642 with infrared transmission	•	•	•	•	Yes	653217-xx	
																	TS 760 with radio or infrared transmission	–	•
Tool touch probe	TT 160 with cable	•	•	•	•	Yes	729763-xx	80											
						TT 460 with radio or infrared transmission	•		•	•	•	•	Yes	1192582-xx					
Additional operating station	ITC 362 with 24-inch touchscreen (separate TNC keyboard required)	–	–	•	•	Yes	1346871-xx	81											
						ITC 855 with 15-inch touchscreen and ASCII keyboard	–		•	•	–	•	Yes	1370459-xx					
													ITC 860 19-inch touchscreen; separate TNC keyboard required	–	–	•	–	•	Yes
Industrial PC	IPC 306 with 8 GB RAM	–	•	•	•	Yes	1179966-01	82											
Vision system	VT 121 for tool inspection	–	–	•	•	Yes	1249466-01	83											

• = Available as option
 – = Not available
 ✓ = Standard function

The term **Dynamic Precision** refers to optional functions for HEIDENHAIN controls that reduce the dynamic error of machine tools. By improving the machine's dynamic performance and increasing TCP accuracy, these functions take milling to the limits of the technologically possible—regardless of the machine's age and load or machining position. No mechanical modifications are needed.

Attaining precise workpieces with high surface quality no longer requires working slowly. That's because, with Dynamic Precision, machine tools work **fast and accurately** at the same time.

High precision and fast machining are the basis for increased productivity. Unit cost is therefore reduced without sacrificing accuracy or surface quality. Dynamic Precision even ensures that accuracy is maintained regardless of the operating time and load. As a result, feed rates no longer need to be reduced on account of machine age or load.

The functions of Dynamic Precision are available as options for HEIDENHAIN controls. The machine manufacturer can deploy individual functions alone or in combination.

- CTC: compensation of acceleration-dependent position error at the tool center point, thus increasing accuracy during acceleration phases
- MVC: damping of machine vibrations for improved surface finishes
- PAC: position-dependent adaptation of control parameters
- LAC: load-dependent adaptation of control parameters for high accuracy, regardless of machine age and load
- MAC: motion-dependent adaptation of control parameters

The functions of Dynamic Precision are adapted to the movements and loads of the machine tool through a high clock rate in the controller unit (a component of HEIDENHAIN controls).

Because Dynamic Precision is made up of software functions, it requires no intervention in the machine's mechanics or power train. Nevertheless, the machine manufacturer must enable the individual functions, enter their parameters, and adapt them to the machine.



	TNC7	TNC 640	TNC 620	Page
Dynamic Precision	✓	✓	✓	
Cross Talk Compensation (CTC) Compensation of position errors arising from machine compliance	Option	Option	Option	36
Machine Vibration Control (MVC) Damping of machine oscillations	Option	Option	Option	40
Position Adaptive Control (PAC) Position-dependent adaptation of control parameters	Option	Option	Option	37
Load Adaptive Control (LAC) Load-dependent adaptation of controller parameters	Option	Option	Option	38
Motion Adaptive Control (MAC) Motion-dependent adaptation of control parameters	Option	Option	Option	39

Installation by the machine manufacturer

For more information, see the *Dynamic Precision* Technical Information document, and visit www.klartext-portal.com

At HEIDENHAIN, the term **Dynamic Efficiency** refers to innovative TNC functions that help make heavy machining and roughing processes more efficient and reliable. These software functions not only support the machine operator but also make the manufacturing process itself faster, more stable, and more predictable—in short, more efficient. Dynamic Efficiency helps boost the metal removal rate and reduce machining time.

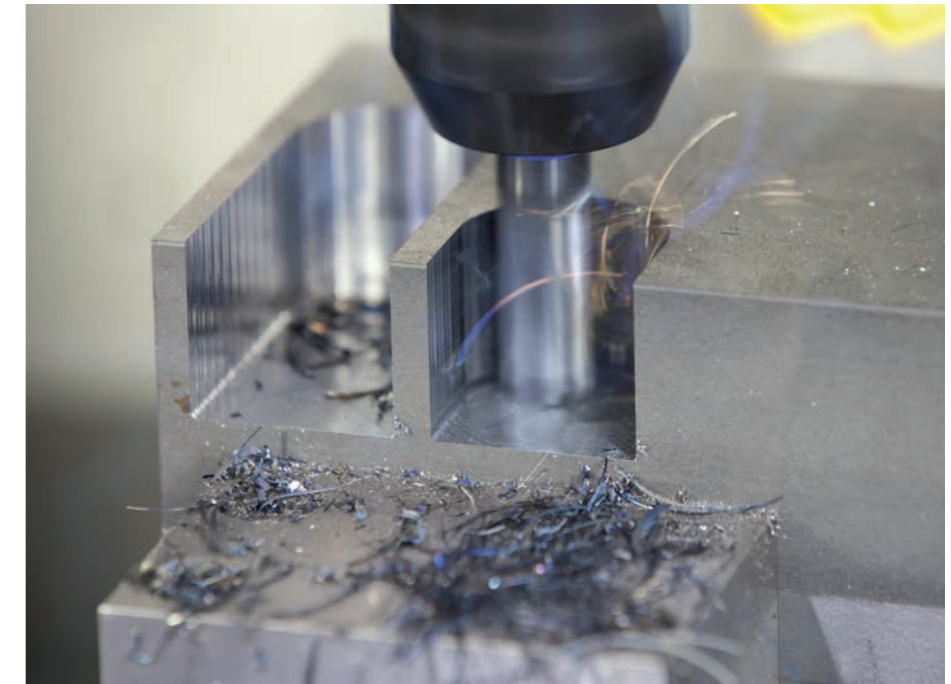
Dynamic Efficiency encompasses four software functions:

- ACC: reduces chatter susceptibility, thus permitting higher feed rates and greater infeeds
- AFC: controls the feed rate based on the machining situation
- Trochoidal milling: a function for tool-friendly roughing of slots and pockets
- OCM: optimizes roughing processes for fast and low-wear machining

The AFC, ACC, and OCM functions are described in greater detail later in this brochure and are identifiable by their Dynamic Efficiency logo.

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.

By permitting higher removal rates, Dynamic Efficiency boosts productivity without requiring operators to resort to special tools. The avoidance of tool overloading and premature tool wear, as well as the additional gain in process reliability, significantly contributes to more economical production.



	TNC7	TNC 640	TNC 620	Page
Dynamic Efficiency	✓	✓	–	
ACC – Active Chatter Control Active Chatter Control	Option	Option	Option	48
AFC – Adaptive Feed Control Adaptive feed control	Option	Option	–	47
Trochoidal milling	Standard	Standard	Standard	–
OCM	Option	Option	Option	49

Installation by the machine manufacturer/operator

For more information, see the *Dynamic Efficiency* Technical Information document, and visit www.klartext-portal.com

Connected Machining

Solutions for networked manufacturing



With the Connected Machining package of functions, the HEIDENHAIN control streamlines operations thanks to easy data usage, time-saving workflows, and transparent processes. These solutions put the operator at the center of digital job management through the control on his milling machine or lathe. The HEIDENHAIN control can be linked as desired to every production-related area of the company, featuring adaptability to already existing structures and extensibility for future innovation.

The benefits are time-saving workflows, workload reduction through easy data usage, and transparent processes in all areas of the company—not just on the shopfloor but also in design, production planning, management, logistics, servicing, and more. Purely digital job management with Connected Machining augments the strong manufacturing capabilities of modern equipment and machines.

Efficient and lossless data transmission
Use the TNC control to directly access data in your network or to transfer data from your PC to the TNC control. The program manager makes access to network drives quick and easy. The SFTP share directly in the Explorer of your PC offers secure access to local drives of the TNC controls.

Supported file formats
Being able to display, edit and evaluate common file types opens up many possibilities for using digital job data directly on the control. The Extended Workspace display provides a clear representation of your job data without losing track of important information about the control.

- PDF viewer for displaying task lists and manufacturing notes, for example (*.pdf)
- CAD viewer for displaying 3D models and design drawings, for example (*.step, *.stl, *.dxf, *.iges)
- Graphics files for displaying clamping situations, for example (*.gif, *.bmp., *.jpg, *.png)
- Spreadsheets for displaying working steps and tool data, for example (*.xls, *.xlsx, *.ods)
- HTML viewer for displaying documentation and help files, for example (*.htm, *.html, *.chm)

Remote operation
The following functions are available for remote operation of HEIDENHAIN controls or third-party CNC controls:

- HEIDENHAIN OPC UA NC Server (options 56 to 61) for cross-platform remote monitoring and controlling of a TNC. The task-oriented method of operation reduces the time required and allows interaction without in-depth knowledge about the control.
- HEIDENHAIN DNC (option 18) for monitoring and remote operation of older machines.
- Remote Desktop Manager (option 133) for access from the control to Windows PCs and the software installed there.
- Firefox web browser for accessing web-based applications, such as ERP and MES clients, e-mail clients, and HTML5 applications.

The following solutions and options are offered as well:

- **Remote Desktop Manager** (option 133) allows operators to access PCs and their installed software from the control.
- The **HEIDENHAIN DNC** interface (option 18) links the control to enterprise-resource planning and production-activity control systems.
- **Extended Workspace** displays processes and depicts job data at the machine in a large and clear format.
- The **OPC UA NC Server** option enables a convenient cross-platform connection to your IT system landscape.



	TNC7	TNC 640	TNC 620	Page
Connected Machining	✓	✓	✓	
Remote Desktop Manager	Option	Option	Option	55
HEIDENHAIN DNC	Option	Option	Option	54
OPC UA NC Server	Option	Option	Option	53
Extended Workspace	✓	✓	–	81

Installation by the machine manufacturer

For more information, see the TNC brochures.

Digital Shop Floor

Software solutions for your manufacturing operations



The software solutions of the Digital Shop Floor from HEIDENHAIN are ideal for manufacturing companies that use CNC machine tools. They provide support along the process chain, ensuring improved process quality and enabling sustainable productivity gains.

StateMonitor
StateMonitor provides insight into the production processes of machine tools in real time. It collects and visualizes machine statuses, tool data, NC program durations and other machine-related data. This software solution allows you to document setup and production times, plan maintenance work and speed up your response time to malfunctions.

- Interfaces:
- HEIDENHAIN DNC
 - OPC UA
 - umati
 - MTConnect
 - Modbus TCP
 - FOCAS

PlantMonitor
PlantMonitor gives you a real-time overview of machine data from multiple instances of StateMonitor and an analysis of individually configurable machine combinations across multiple sites and production centers.

Software options for use with StateMonitor



JobTerminal manages job data, job times and job analyses.



MaintenanceManager displays maintenance and malfunction messages. Maintenance work can be planned based on machining hours, machine messages and intervals of time.



5 Signals records additional machine data via PLC signals.



5 Machines allows you to connect an additional five machine tools.

Further information:

Find out more about StateMonitor under *Software* (page 66).



#Machine data collection with StateMonitor

Programming and operation

Machining with a rotary table

Many five-axis operations that initially seem very complex can actually be reduced to conventional 2D movements executed on a cylindrical surface. For creating and editing these programs quickly and easily without a CAM system, the TNC comes to your aid with helpful functions.

Cylinder surface machining

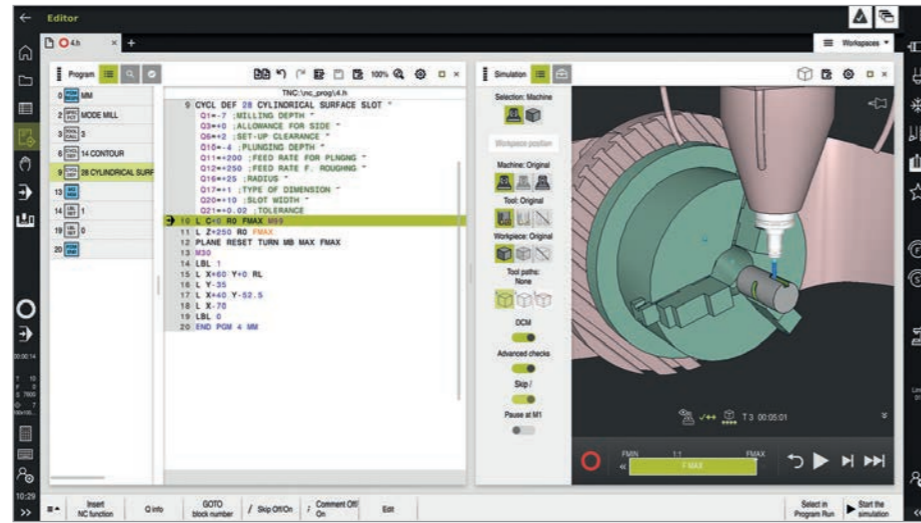
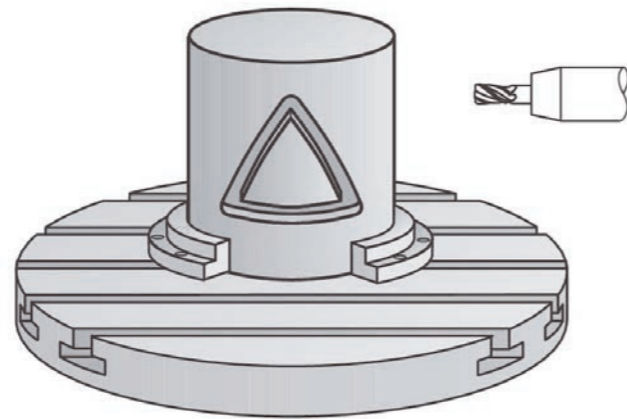
The TNC makes it easy for you to program straight-line and arc contours on cylindrical surfaces for rotary and tilting tables by conveniently allowing you to program the contour on the plane of an unrolled cylinder (axis-independently on the TNC7, TNC 640, TNC 620 and TNC 320). The TNC then executes the programmed machining operation on the cylinder surface.

The TNC provides four cycles for cylinder 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

Feed rate for rotary axes and tables in mm/min

By default, rotary axis feed rates are programmed in degrees/min, but the TNC can also interpret these feed rates in mm/min. This makes the feed rate at the contour independent of the distance between the tool center point and the center of the rotary axis.



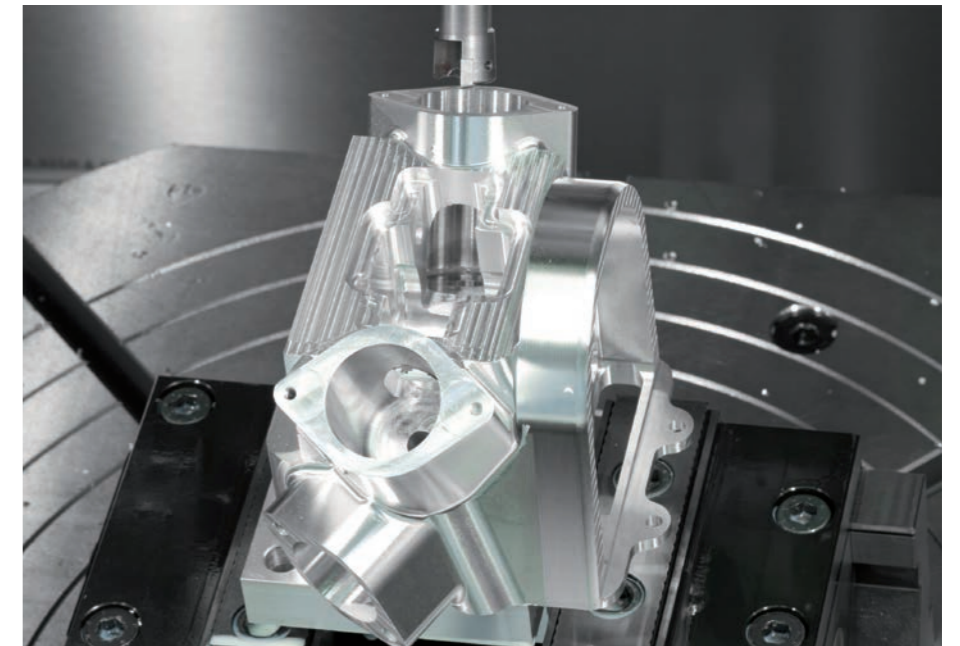
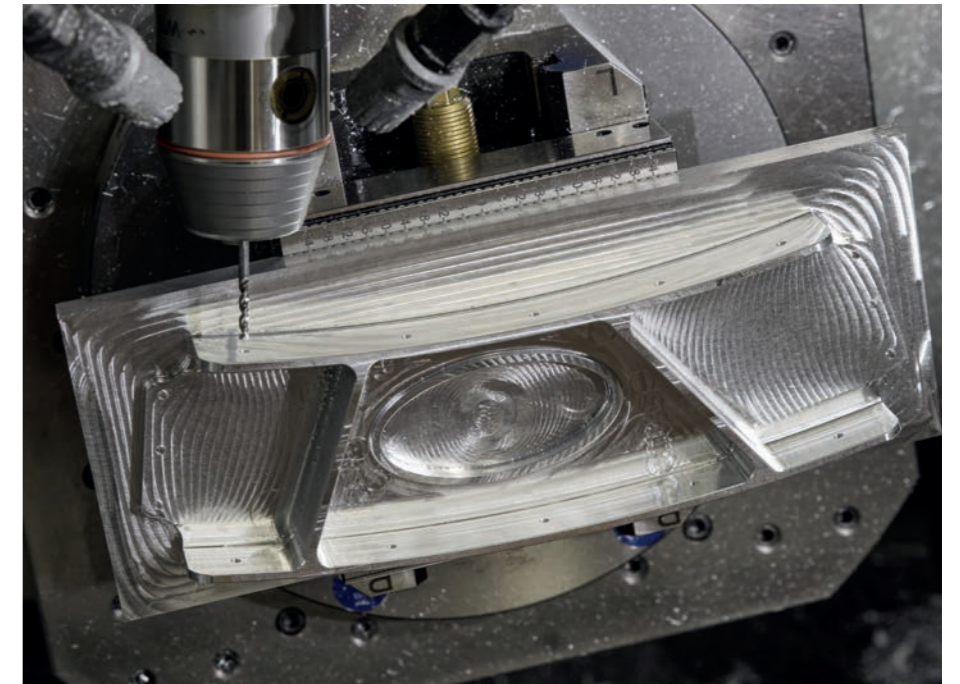
Programming and operation

Coordinate transformation: tilting the working plane, and the PLANE function

Programs for contours and holes on inclined surfaces are usually very complex, requiring significant computing resources and programming work. In cases like these, the TNC can save you a great deal of programming time. You simply program the machining operation as usual in the main plane (e.g., in XY). The machine then runs the program in a plane that has been tilted with respect to the main plane about one or more rotary axes.

The PLANE function makes it easy to define a tilted working plane and, based on the information in the workpiece drawing, offers seven different methods for doing so. In order to keep this complex function as easy to use as possible, each plane definition has its own animation that you can view prior to selecting the function. Intuitive help graphics assist you during programming.

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.



Advanced Function Set 1 TNC7/TNC 640/TNC 620 TNC 320	Option 8 ID 617920-01 ID 536146-01
TNC7 TNC 640 TNC 620 TNC 320	NC SW 81762x-16 or later NC SW 34059x-01 or later NC SW 34056x-01/73498x-01/81760x-01 or later NC SW 34055x-01/771851-01 or later
Installation by the machine manufacturer	
For more information , see the TNC brochures.	

Advanced Function Set 1 TNC7/TNC 640/TNC 620 TNC 320	Option 8 ID 617920-01 ID 536164-01
TNC7 TNC 640 TNC 620 TNC 320	NC SW 81762x-16 or later NC SW 34059x-01 or later NC SW 34056x-01/73498x-01/81760x-01 or later NC SW 34055x-01/771851-01 or later
Installation by the machine manufacturer	
For more information , see the TNC brochures, and visit www.klartext-portal.com	

Programming and operation

Touch probe cycles

Workpiece alignment

The HEIDENHAIN touch probes and probing functions of the TNC reduce the need for tedious manual alignment of the workpiece:

- Workpiece clamped in any position
- The touch probe determines the actual clamping situation by probing a surface
- The TNC compensates for this misalignment with a "basic rotation," either by rotating the NC program to the ascertained angle or by correcting the actual misalignment through a turn of the rotary table
- The TNC offers manual, automatic, and semiautomatic cycles for correcting misalignments in two or three dimensions
- Some automatic probing cycles feature semiautomatic mode, tolerance monitoring and actual-to-nominal value transfer
- Automatic probing cycle with which touch points in one direction can be repeated over a specified distance

Preset setting

Being able to define the preset quickly and reliably reduces nonproductive time and increases machining accuracy. The TNC features a myriad of probing cycles for automatic preset setting.

Workpiece inspection

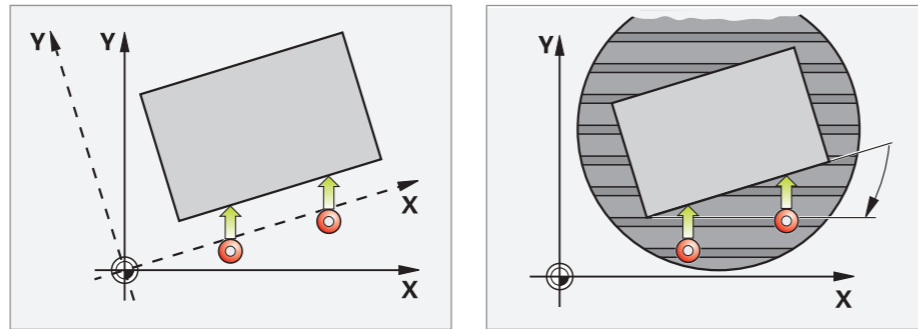
The TNC features numerous measuring cycles for inspecting the geometry of machined workpieces. This allows you to do the following:

- Identify a workpiece and call an appropriate part program
- Check for correct execution of all machining operations
- Detect and compensate for tool wear, etc.

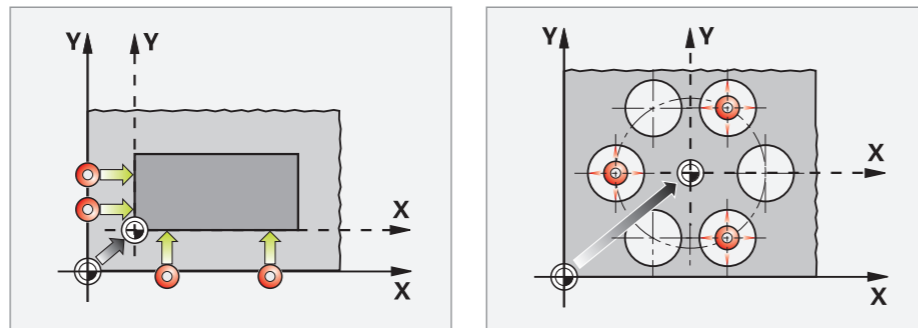
Tool measurement

In conjunction with the TT touch probes for tool measurement, the TNC makes it possible to automatically measure tools inside the machine. The measured tool length and radius are stored by the TNC in its central tool memory. In-process tool measurement allows you to immediately identify wear or breakage, thus avoiding scrap and rework.

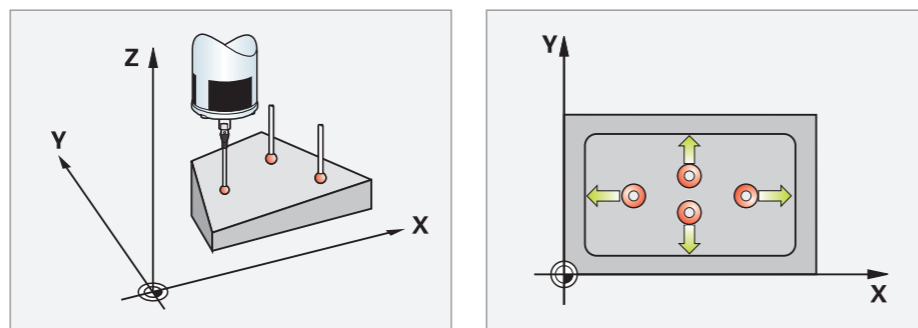
On the TNC 620, the touch probe cycles are available only with option 17. This option is automatically enabled with the use of HEIDENHAIN touch probes featuring the EnDat interface (NC SW 81760x-06 or later). The TNC7, TNC 640 and TNC 320 already include the touch probe cycles as part of their standard functions.



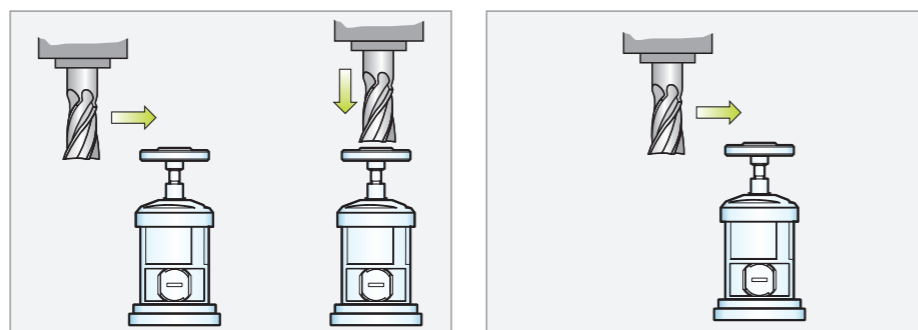
Compensating for workpiece misalignment by rotating the coordinate system or rotating the table



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



Workpiece measurement (e.g., angle of a plane or rectangular pocket)



Tool measurement (e.g., tool length and radius or tool wear)

Touch Probe Functions	Option 17	ID 634063-01
TNC7	Standard function	
TNC 640	Standard function	
TNC 620	NC SW 34056x-01/73498x-01/81760x-01 or later	
TNC 320	Standard function	
Installation by the machine manufacturer		
For more information , see the TNC brochures and the <i>Touch Probes</i> brochure.		

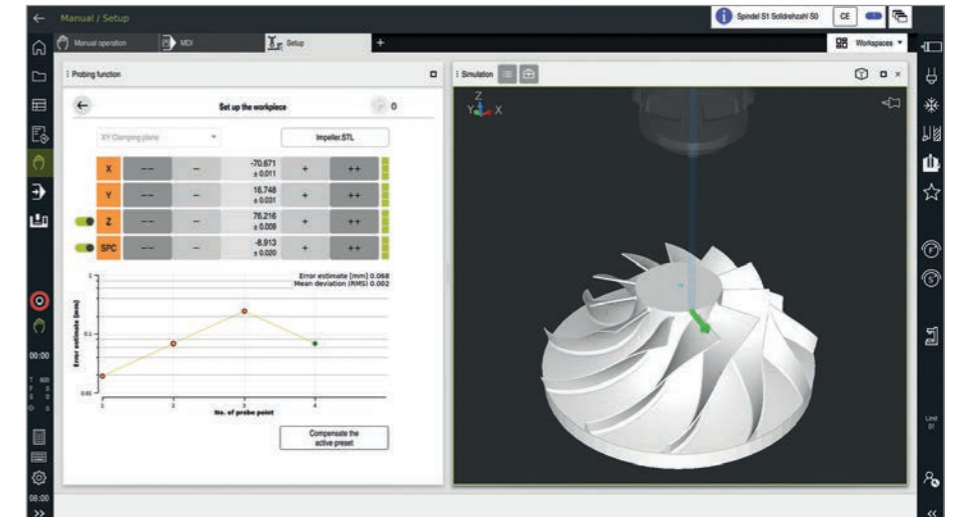
Programming and operation

Graphical 6D workpiece setup

For single parts and small lot sizes without a special holder with a defined location, the position of the workpiece blank almost always needs to be ascertained before machining. With this intuitive probing function, the TNC7 offers the possibility of setting up workpieces quickly, easily and safely with graphic guidance. The exact position of the workpiece blank is measured in the machine's working space and reported to the control.

The simulation view depicts a 3D model of the workpiece blank in the machine's working space. After the operator has roughly aligned the model manually, a green arrow indicates that the model is ready to be probed. The axis keys or the electronic handwheel are used to position the touch probe to the blank in order to record the touch points. The control automatically selects the probing direction. All of the six degrees of freedom can thus be measured with only one single function.

During the entire setup process, the control informs the operator about the quality of the probing points for determining the position and orientation of the workpiece. The operator can thus quickly see when the measurement of the actual position and orientation of the workpiece is complete. The operator can also move the rotary axes during the setup procedure in order to probe undercuts, inclined surfaces, or rounded surfaces, for example. This allows the operator to align even complex workpiece blanks relative to pre-machined features, such as is needed for mold repair or 3D-printed workpiece blanks.

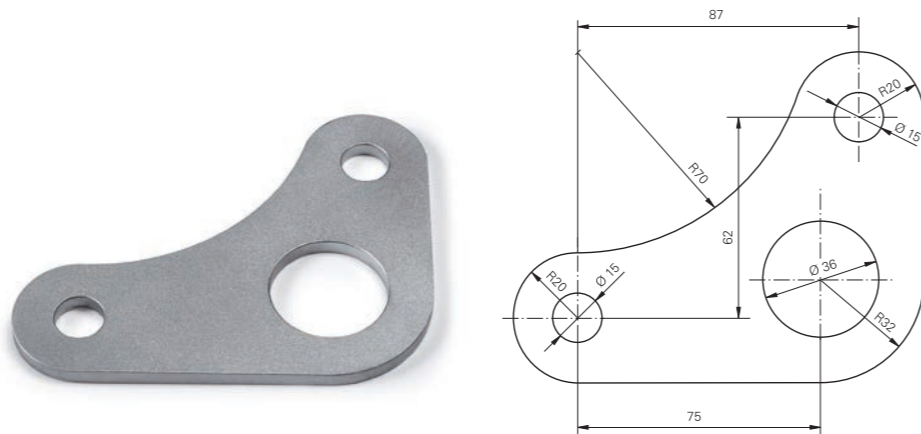


Programming and operation

Advanced programming functions: FK free contour programming, canned cycles

FK free contour programming

Not all workpieces are suitably dimensioned for conventional NC programming. In this case, the FK free contour programming function lets you simply enter data from the drawing without performing additional conversions or calculations! In the process, you can leave individual contour elements undefined as long as the overall contour is defined. If the data allow for more than one mathematical solution, then the TNC programming graphics helpfully render the possible variants to choose from. On the TNC7, graphical programming (a standard feature) replaces FK programming.



Standard cycles

In addition to the canned cycles for drilling and tapping (with or without a floating tap holder), option 19 also gives you cycles for thread milling, reaming, boring, and hole patterns, as well as milling cycles for plane-surface face milling and for roughing and finishing pockets, slots, and studs.

Cycles for complex contours

Special help for roughing pockets with any contour is provided by the Subcontour List (SL) cycles and the Optimized Contour Milling (OCM) software option. These groups of cycles each contain canned cycles for roughing and finishing. The contour or subcontours are defined in subprograms. As a result, a single contour description can be used for various operations with different tools.

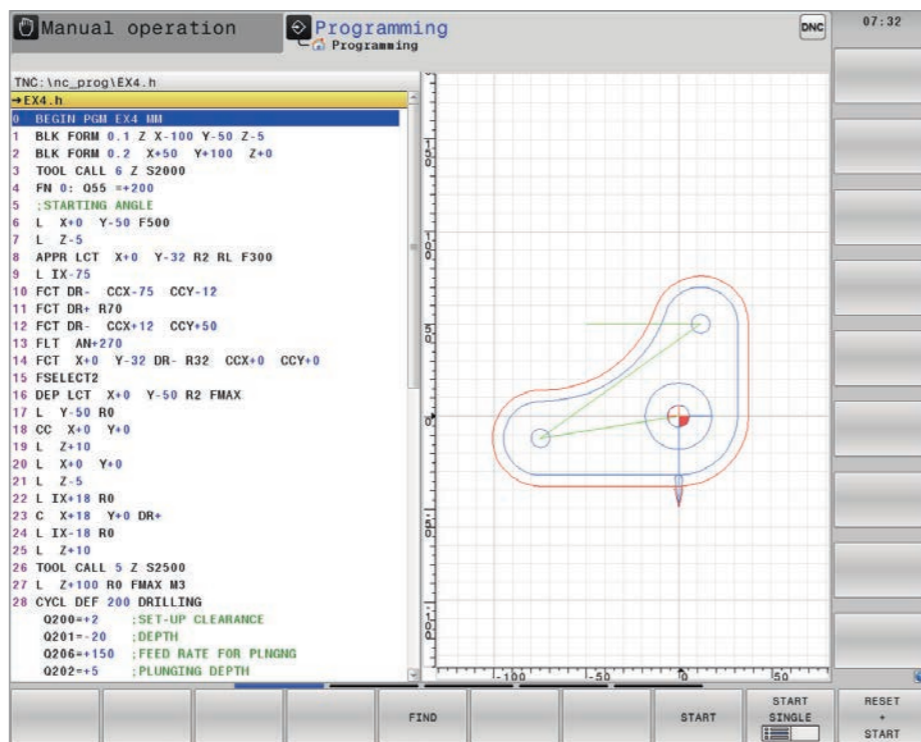
OEM cycles

As original equipment manufacturers, machine manufacturers can contribute their special manufacturing know-how by designing additional fixed cycles and saving them in the TNC. The end user is likewise given the option of programming his own cycles. 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.

Graphical contour programming*

Couple the familiar Klartext functions with the graphical programming capability to streamline your daily work at the machine. With the graphical programming feature, the operator draws the desired contours directly on the touchscreen. The TNC7 then converts the drawing into a Klartext program. This function makes it very easy to modify existing programs.

* Available only for the TNC7



Programming graphics on the TNC 640

Programming and operation

Verification graphics, program-run graphics

Programming graphics

HEIDENHAIN controls support you with detailed programming graphics. These graphics are a feature on the TNC 320, TNC 620 and TNC 640, and are described in their respective brochures.

Other graphical views are available as options:

Simulation

To be on the safe side before running a program, the TNC can graphically simulate the machining operation. The TNC can display this simulation in the following ways:

- In plan view with different shades of depth
- In different projections (as in the workpiece drawing)
- In 3D view

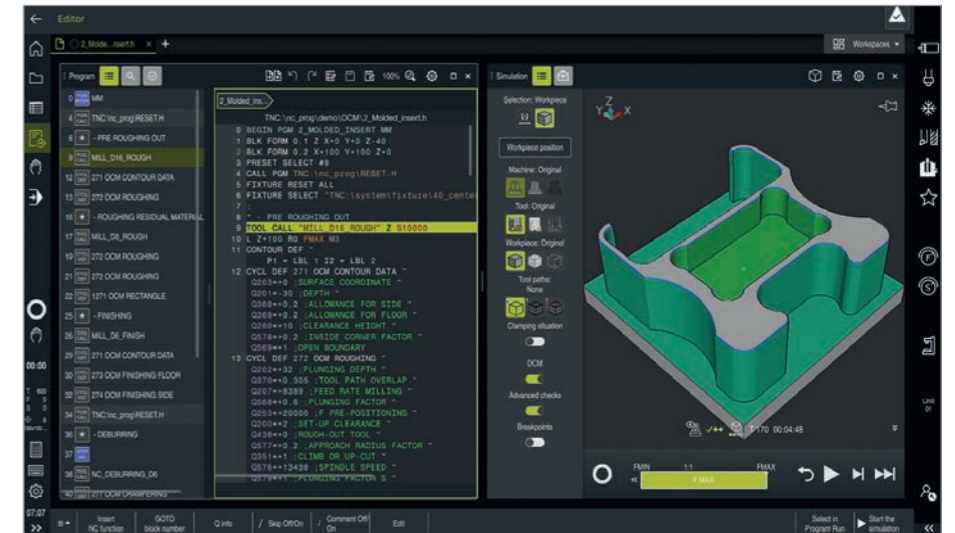
Details can be shown under magnification. In addition, the TNC displays the calculated machining time in hours, minutes, and seconds. During the simulation, the TNC7, TNC 640 and TNC 620 controls can also detect collisions between the workpiece and the tool holder, and can issue appropriate warning messages.

Model comparison

The TNC7 has a feature for comparing the workpiece blank and the finished part. Based on the different colors, you easily see where residual material still remains or where too much material was removed. There is also a measuring function that lets you ascertain the position and depth at any location on the workpiece, together with the current NC block and the associated tool.

Program-run graphics

Direct observation of the workpiece within the machine is usually impossible due to coolant and the safety enclosure. With the program-run graphics you can always see the current machining status of the workpiece as well as the positions of the collision objects in the machine on the control's screen. During machining, you can switch between various operating modes at any time, for example in order to create programs.



Simulation of machining operations on the TNC7



Program-run graphics on the TNC7

Advanced Programming Features	Option 19	ID 628252-01
TNC7	Standard function	
TNC 640	Standard function	
TNC 620	NC SW 34056x-01/73498x-01/81760x-01 or later	
TNC 320	Standard function	
Installation by the operator		
For more information , see the TNC brochures, and visit www.klartext-portal.com		

Advanced Graphic Features	Option 20	ID 628253-01
TNC7	Standard function	
TNC 640	Standard function	
TNC 620	NC SW 34056x-01/73498x-01/81760x-01 or later	
TNC 320	Standard function	
Installation by the operator		
For more information , see the TNC brochures, and visit www.klartext-portal.com		

Programming and operation

Detailed 3D view in the verification and program-run graphics

Finely detailed 3D view

The simulation and program-run graphics of the controls also include additional viewing functions with detailed 3D rendering. These functions help you evaluate the workpiece quality in the simulation prior to machining, as well as in real time during machining.

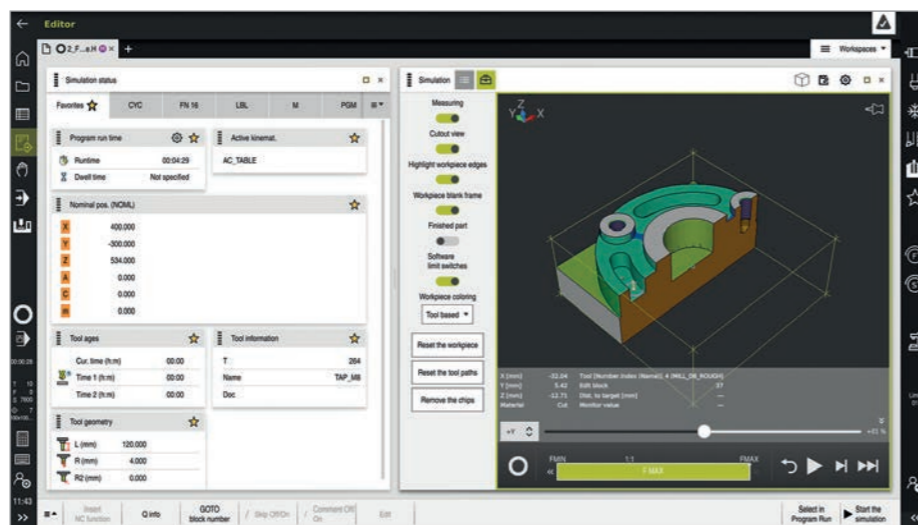
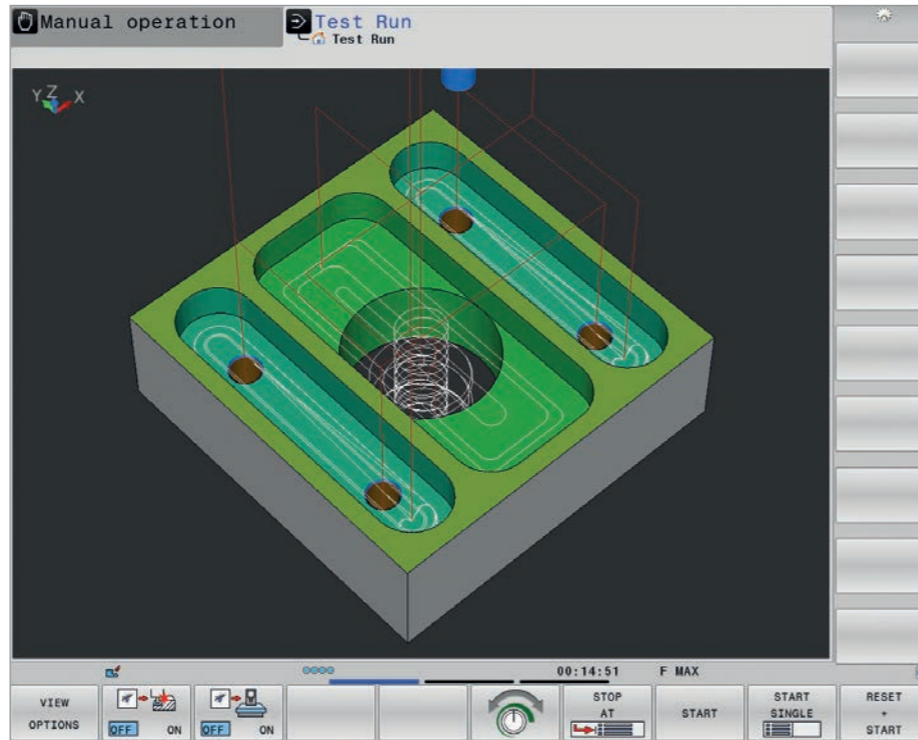
The high-resolution 3D view with strong zoom factors reveals even the tiniest program errors on the workpiece surface. For an even more precise view of the NC data analysis, you can also display the tool paths and their corresponding block numbers. Of particular help is the selectable workpiece transparency feature, which allows you to see hidden cavities and undercuts. The TNC7 offers a sectional view along a plane. This is useful, for example, when looking for residual material.

Enhanced graphics also make tool-specific machining operations easier to see: each machining operation performed with the same tool is shown in its own unique color. In 3D view, the TNC also includes a measuring function that displays the coordinates when you shift the focus within the graphics by touch.

If you just need a quick overview of the contour and the machining time, you can change the resolution and simulation mode to boost the calculation speed.

The TNC 640 (with NC SW 34059x-09 or later) and the TNC7 simulate not only material removal but also the complete 3D machine model, allowing the operator to precisely evaluate processes in the working space before machining. The machine's collision objects are configured and activated by the machine manufacturer.

The TNC7, TNC 640 (with NC SW 34059x-11 or later) and TNC 620 (with NC SW 81760x-08 or later) enable the convenient incorporation of complex workpiece blanks and finished parts as STL files (such as 3D models from CAM systems). In addition, the current machining status can be saved as an STL file during the simulation in order to integrate it into other programs as a workpiece blank.



Advanced Graphic Features	Option 20	ID 628253-01
TNC7 TNC 640 TNC 620 TNC 320	Standard function Standard feature with NC SW 34059x-04 or later NC SW 81760x-01 or later Standard feature in NC SW 771851-01 or later	
Installation by the operator		
For more information , see the TNC brochures, and visit www.klartext-portal.com		

Programming and operation

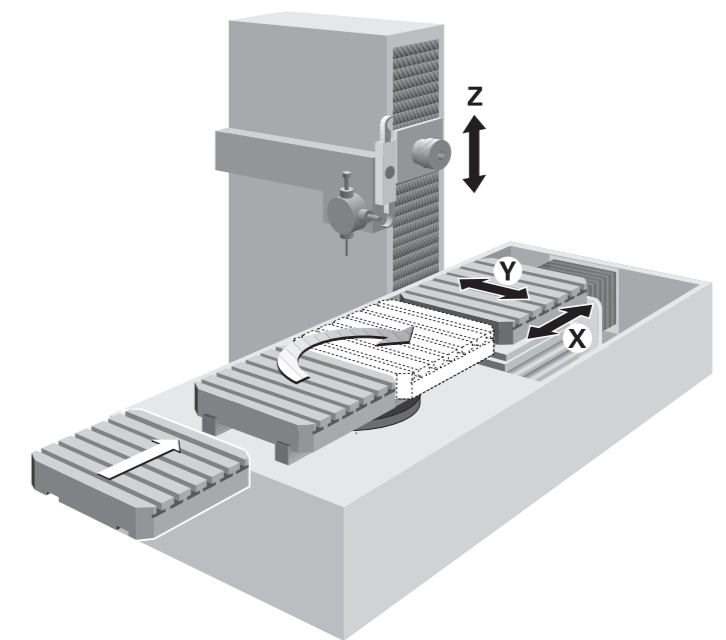
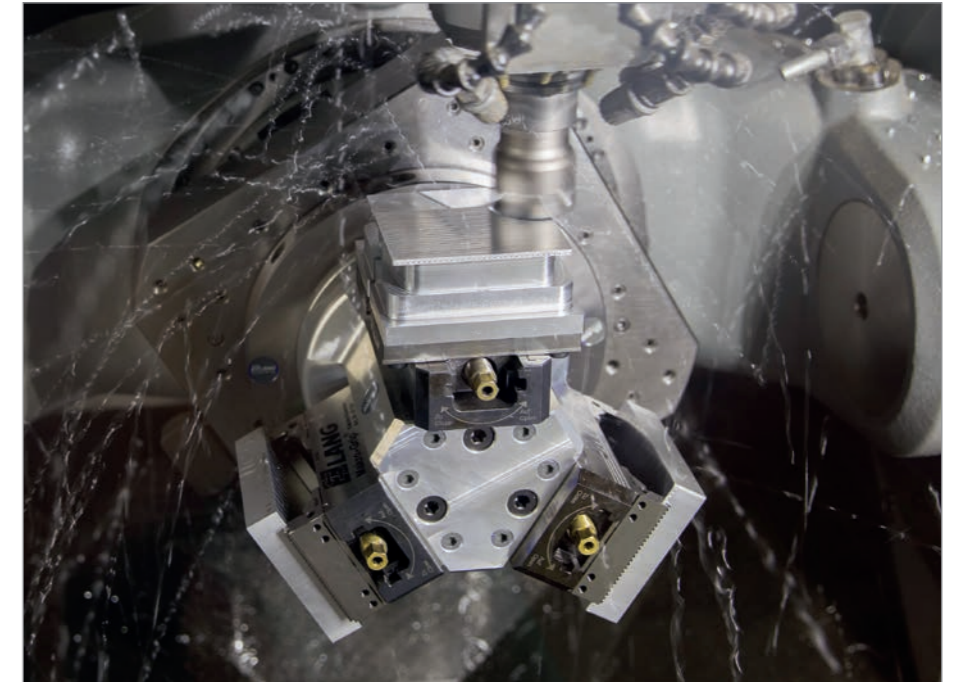
Pallet management

For any sequence of different workpieces loaded on pallets, the TNC is able to identify the proper machining program and datum shift.

When a pallet is loaded for machining, the TNC automatically calls the correct program. This allows different parts to be machined in any order desired.

Pallet insertions can be controlled via PLC axes. The desired sequence and presets for the pallet and workpieces are defined in the pallet table by the operator. The pallet table is freely configurable by the machine manufacturer, which means that any information can be stored in the tables and called later by the PLC.

Execution of the pallet tables with the TNC7 can be either workpiece- or tool-oriented (with NC SW 34059x-08 or later on the TNC 640; with NC SW 81760x-05 or later on the TNC 620).



Pallet Management	Option 22	ID 628255-01
TNC7 TNC 640 TNC 620 TNC 320	Standard function Standard function NC SW 34056x-01/73498x-01/81760x-01 or later -	
Installation by the machine manufacturer		
For more information , see the TNC brochures, and visit www.klartext-portal.com		

Programming and operation

Batch Process Manager: organization and fine tuning of job sequences on the machine

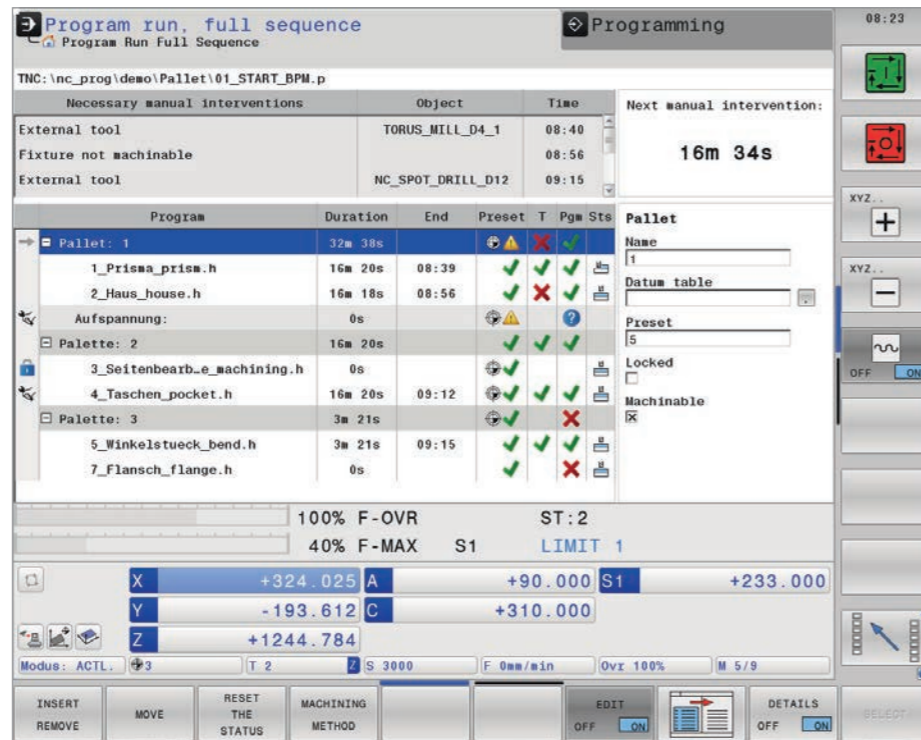
Batch Process Manager is a powerful function for pallet machining and series production. With its intuitive user interface, you can plan your production process and receive important information about upcoming machining operations.

Batch Process Manager makes it possible to easily change the sequence of pending jobs. Batch Process Manager also performs a duration calculation for all planned jobs or NC programs. It checks whether, for example, all NC programs will be able to run without error and whether all necessary tools are available with sufficient remaining service life. Batch Process Manager displays the results of these checks in the status overview, thereby ensuring the smooth execution of the planned jobs.

Batch Process Manager shows the following information in advance:

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

On the TNC 620, Batch Process Manager requires the additional enabling of Pallet Management (option 22). On controls with NC software versions up to 34059x-08 or 81760x-05, Extended Tool Management (option 93) must be enabled as well.



Programming and operation

CAD Import: import contours and machining positions from DXF files

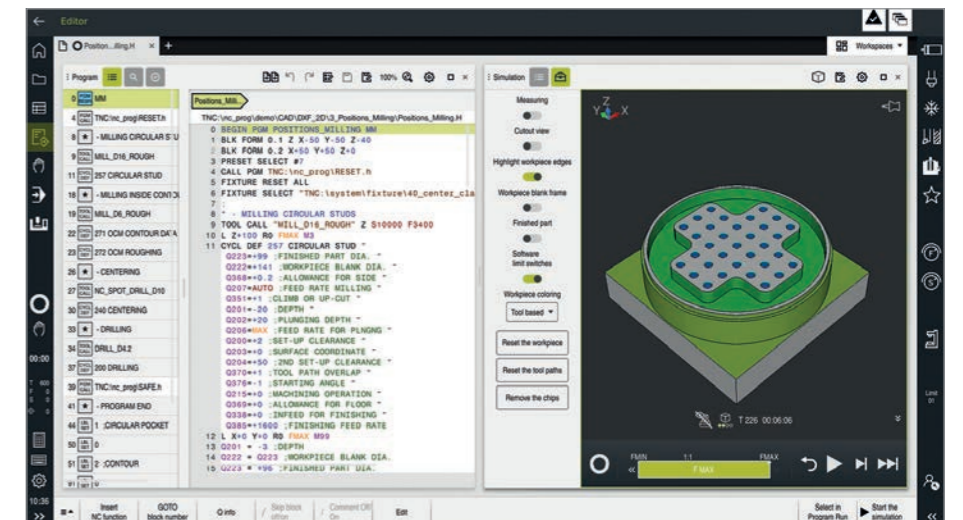
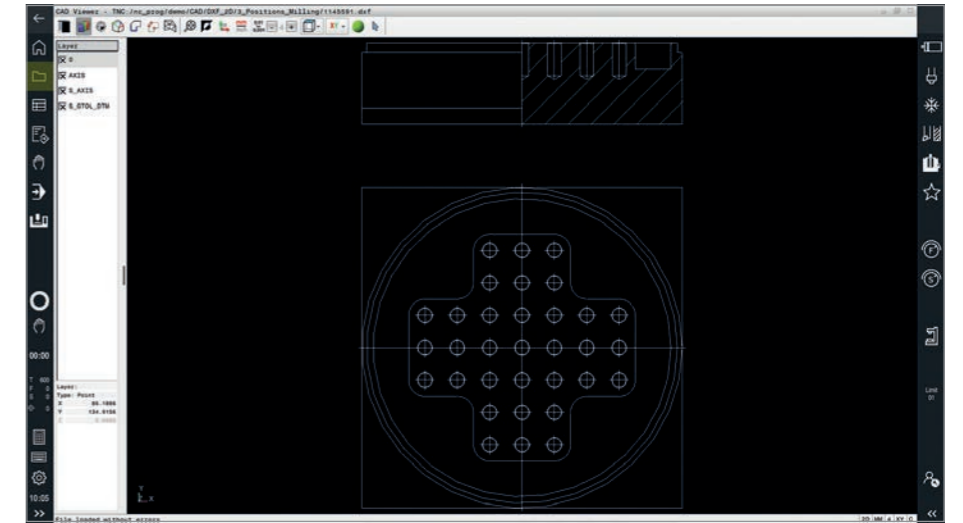
Why program contours when you already have the drawing as a DXF file? You can open DXF files directly on the TNC in order to extract contours or machining positions from them. This not only saves time otherwise spent on programming and testing but also provides assurance that the finished contour precisely complies with the design engineer's specifications.

DXF files usually contain multiple layers used by the design engineer to organize the drawing. In order to minimize unneeded information on your screen during contour selection, you can hide all of the **inessential layers** in the DXF file with just a gesture. This feature requires an operating panel with a touchpad or an external pointing device. The TNC can select a contour train even if it has been stored across **different layers**.

The TNC also supports you in **defining the workpiece preset**. For this purpose, the TNC provides a function that allows you to move the drawing's datum to a useful position by simply selecting an element. Selecting contours is also very convenient: you can select any element by touch. 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. At that point, you can select the next contour element. In this way you can quickly and easily define even extensive contours by touch. You can even shorten, lengthen, or interrupt the contour elements as needed.

In addition, you can easily select **machining positions** and save them as point files, especially for transferring drilling positions or starting points for pocket machining. Of course, the TNC saves the machining positions in such a way that they are approached via the shortest path.

With the CAD Import option, you can also import contours and machining positions from 3D models.



Batch Process Manager	Option 154 Option 22	ID 1219521-01 ID 628255-01
TNC7	NC SW 81762x-16 or later	
TNC 640	NC SW 34059x-08 or later	
TNC 620	NC SW 81760x-05 or later	
TNC 320	–	
Installation by the machine manufacturer		
For more information , see the TNC brochures, and visit www.klartext-portal.com		

CAD Import	Option 42	ID 526450-01
TNC7	NC SW 81762x-16 or later	
TNC 640	NC SW 34059x-02 or later	
TNC 620	NC SW 73498x-02/81760x-01 or later	
TNC 320	NC SW 771851-01 or later	
Installation by the operator		
For more information , see the TNC brochures, and visit www.klartext-portal.com		

Programming and operation

CAD Import: import contours from 3D models

CAD Import (option 42)

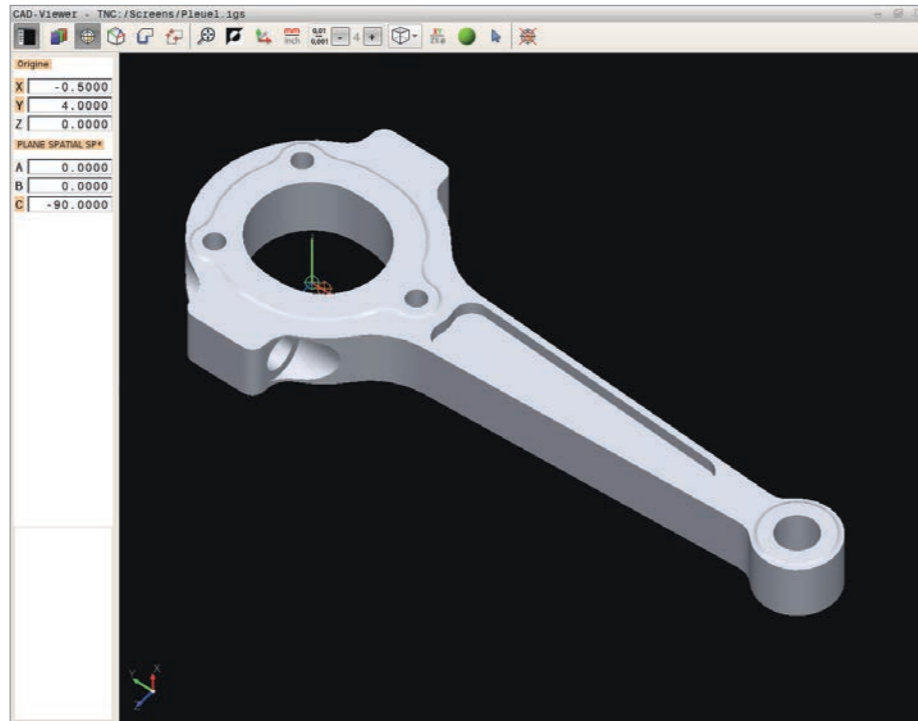
CAD Viewer enables the display of **2D and 3D models** right on the TNC (e.g., DXF, Step, STL or IGES files). Using the CAD Import option, you can easily incorporate contours and machining positions from these CAD files directly into your Klartext program. You can thereby reduce the amount of programming work and avoid input errors resulting from transposed digits, incorrectly placed decimal points, and the like.

In particular, extracting machining information directly from CAD data provides additional possibilities for creating NC programs with a tilted working plane. You can also define the preset on the 3D model with a **3D basic rotation**. In addition, you can place a datum with the appropriate 3D rotation on the desired working plane.

The working plane can be conveniently saved to the clipboard and transferred to the NC program with the proper transformation and relevant **PLANE command**. On the defined working plane, you can extract contours and machining positions, and transfer them to the NC program.

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

The CAD Import option is an expansion to the DXF Converter. All previous functions have been adopted and enhanced to include 3D import functions. Nearly all of the common DXF, Step and IGES file formats can be used.

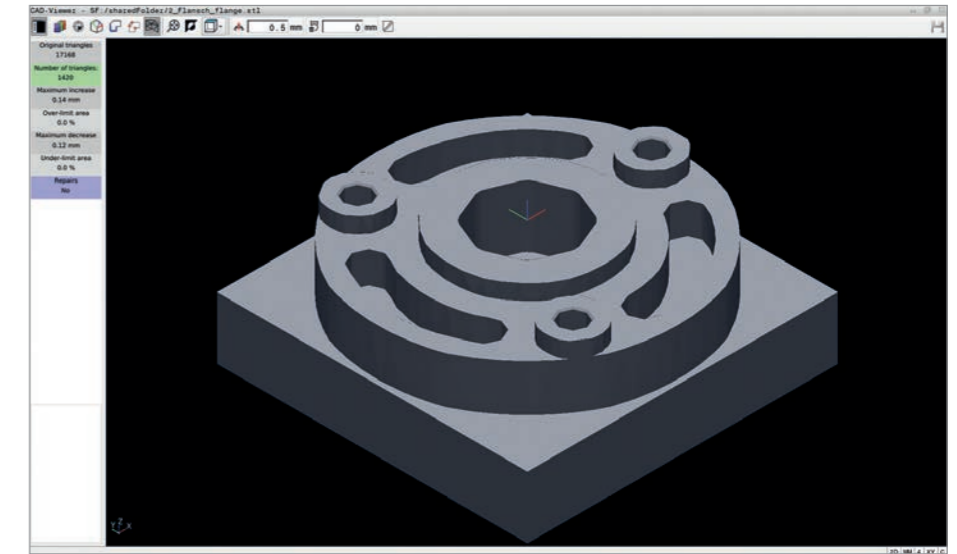


Programming and operation

Optimization of CAD models

The CAD Model Optimizer software option offers the operator the power to simplify 3D models and heal contours, thus creating valid STL files for collision monitoring or simulation.

The initial model can be imported through the CAD viewer. The 3D mesh function simplifies the model and autonomously corrects errors such as small holes in the solid model or self-intersecting lines on surfaces. The control covers the opened 3D model with a mesh of triangles. The control then generates an STL file that you can use for various functions of the control. For example, you can then quickly and easily repair faulty files of fixtures or tool holders.



CAD Import	Option 42	ID 526450-01
TNC7	NC SW 81762x-16 or later	
TNC 640	NC SW 34059x-08 or later	
TNC 620	NC SW 81760x-05 or later	
TNC 320	NC SW 771851-05 or later	
Installation by the operator		
For more information , see the TNC brochures, and visit www.klartext-portal.com		

CAD Model Optimizer	Option 152	ID 1353918-01
TNC7	NC SW 81762x-16 or later	
TNC 640	NC SW 34059x-16 or later	
TNC 620	NC SW 81760x-16 or later	
TNC 320	–	
Installation by the operator		
For more information , see the TNC brochures, and visit www.klartext-portal.com		

Programming and operation

Turning functions: milling and turning on the same machine

The TNC7 and TNC 640 provide powerful functions for NC-program-driven switching between turning mode and milling mode. You are completely free to decide how and when you wish to combine these two machining modes.

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 TNC7 and TNC 640 automatically take 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. You use exactly the same Klartext functions for the contour description as you would for defining a milling contour.

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 TNC7 and TNC 640 cut parallel to the axis or the contour. You define the machining operations (roughing, finishing) or the oversize with dialog guidance in the corresponding parameters.

Other machining operations:

- Recessing
- Thread machining
- Blank form update
- Orientation of the turning tool



Turning with a facing slide

A facing slide allows you to perform turning operations on a stationary workpiece. This, in turn, makes it possible to implement off-center or tilted-plane turning operations. With a facing slide, the cutting edge rotates in the spindle, while an axis integrated in the facing slide controls the turning tool (plan stroke). With the TNC7 and 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 TNC7 and TNC 640 handle all the conversions for you and automatically perform all motion sequences.



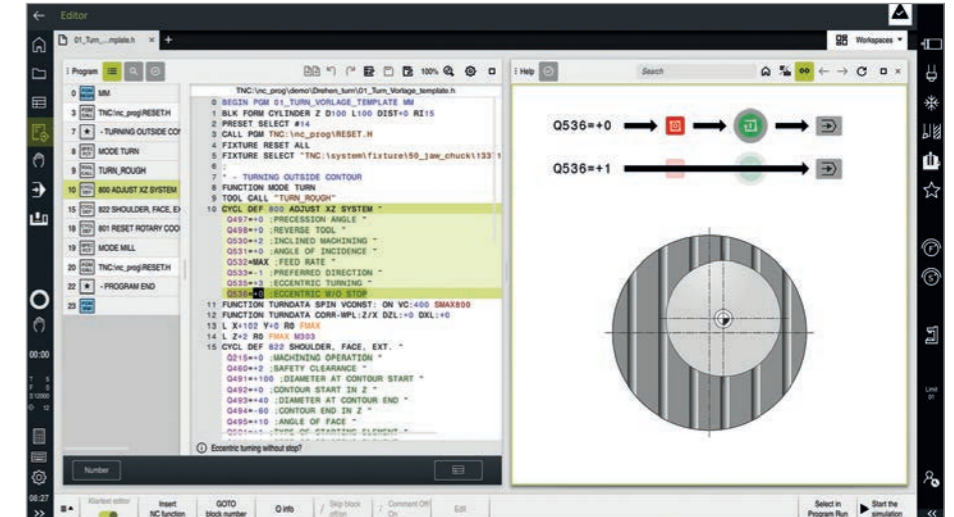
Turning	Option 50	ID 634608-01
TNC7	NC SW 81762x-16 or later	
TNC 640	NC SW 34059x-01 or later	
TNC 620	–	
TNC 320	–	
Installation by the machine manufacturer		
For more information , see the TNC brochures, and visit www.klartext-portal.com		

Programming and operation

Turning functions: eccentric turning

The eccentric turning function lets you perform turning operations even when the workpiece axis is misaligned with the axis of rotation due to the setup. During machining, the TNC7 and TNC 640 then compensate for any eccentricity through corrective movements of the rotary spindle coupled with the linear axis. This can significantly reduce your setup times.

* On controls with NC software version 34059x-04, RTC (option 135) must be enabled.



Turning	Option 50	ID 634608-01
TNC7	NC SW 81762x-16 or later	
TNC 640	NC SW 34059x-04 or later	
TNC 620	–	
TNC 320	–	
Installation by the machine manufacturer		
For more information , see the TNC brochures, and visit www.klartext-portal.com		

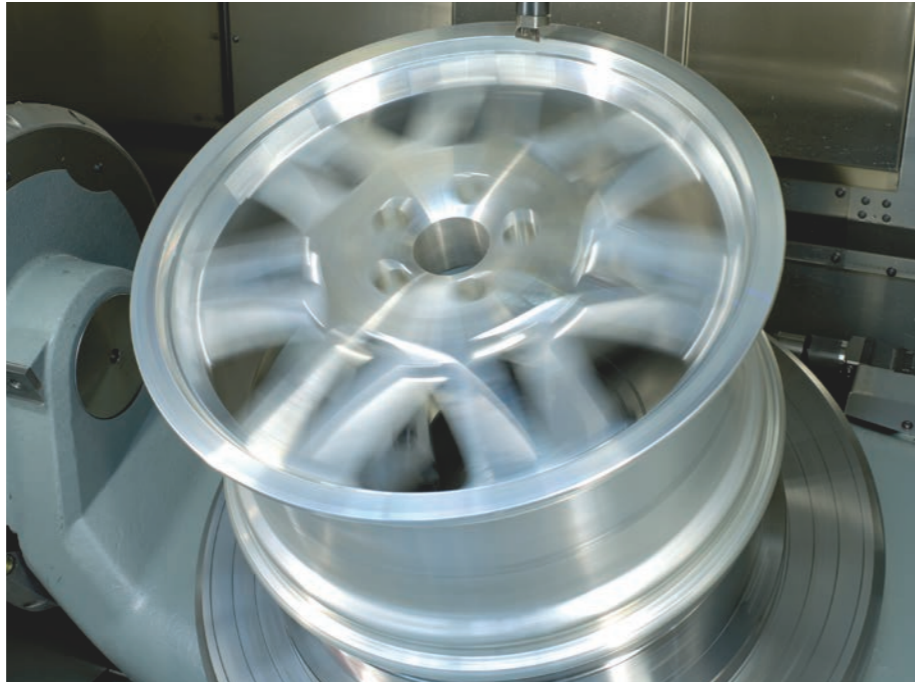
Programming and operation

Turning functions: simultaneous turning

This option allows you to produce complex contours that would otherwise require machining at varying tool angles. This way you can, for example, produce undercut contours with just one tool. For simple contours, this option can prolong tool life by using a larger area of the indexable insert. This involves at least three-axis motion (two linear axes and one rotary axis).

The cycles for simultaneous turning monitor the workpiece contour relative to the tool and the tool carrier. To attain optimal surface finishes, these cycles avoid unnecessary tilting movements. If you wish to activate specific tilting movements, you can define start and finish angles on the TNC7 and TNC 640. A cycle for simultaneous roughing is also available (with NC SW 34059x-11 or later).

The control now permits the use of FreeTurn tools for inclined and simultaneous turning operations. FreeTurn tools are lathe tools with a special tooth arrangement. Depending on the design, a single FreeTurn tool can perform both axis-parallel and contour-parallel roughing and finishing. The use of FreeTurn tools reduces machining times because fewer tool changes are needed (with NC software version 16 or later).



Advanced Function Set Turning*	Option 158	ID 1237237-01
TNC7	–	
TNC 640	From NC SW 34059x-09 through 34059x-16	
TNC 620 / TNC 320	–	

* Advanced Function Set Turning additionally requires option 50

Turning v2*	Option 158	ID 1359635-01
TNC7	With NC SW 81762x-16 or later	
TNC 640	With NC SW 34059x-17 or later	
TNC 620 / TNC 320	–	

Installation by the machine manufacturer

For more information, see the TNC brochures, and visit www.klartext-portal.com

* Turning v2 includes option 50

Programming and operation

Extended Tool Management

This software option extends tool management by the two tables **Tooling list** and **T usage order**. The **Tooling list** shows the tool requirements of the NC program or pallet to be run. The **Tooling list** contains information about all the tools called within an NC program. Before starting the program, you can check, for example, whether all tools are contained in the magazine. When you select an NC program in the **Program run** operating mode, the control automatically creates the **Tooling list** table. The control lists all the tools called within the active NC program in numerical order.

The **T usage order** table shows the tool sequence of the pending pallet or NC program. Before starting the program, you can see, for example, when a manual tool change will take place. When you select an NC program in the **Program run** operating mode, the control automatically creates the **T usage order** table. The control lists all the tools called within the active NC program in chronological order.

Extended tool management enables you to detect the tool requirements in time and thus prevent interruptions during program run.

NO.	T	NAME	TOOL INFO	T PROG	USAGE	TOOL TIME
0	5	DRILL_D5	OK	"DRILL_D5"	39	15:35:39.000 12.10.2022
1	203	NC_SPOT_DRILL_D10	Not found	"NC_SPOT_DRILL_D10"	17	15:36:18.000 12.10.2022
2	263	TAP_M6	Not found	"TAP_M6"	29	15:36:35.000 12.10.2022
3	5	DRILL_D5	OK	"DRILL_D5"	39	15:37:04.000 12.10.2022
4	203	NC_SPOT_DRILL_D10	Not found	"NC_SPOT_DRILL_D10"	17	15:37:43.000 12.10.2022
5	263	TAP_M6	Not found	"TAP_M6"	29	15:38:00.000 12.10.2022
6	5	DRILL_D5	OK	"DRILL_D5"	39	15:38:29.000 12.10.2022
7	203	NC_SPOT_DRILL_D10	Not found	"NC_SPOT_DRILL_D10"	17	15:39:08.000 12.10.2022
8	263	TAP_M6	Not found	"TAP_M6"	29	15:39:25.000 12.10.2022
9	5	DRILL_D5	OK	"DRILL_D5"	39	15:39:54.000 12.10.2022
10	203	NC_SPOT_DRILL_D10	Not found	"NC_SPOT_DRILL_D10"	17	15:40:33.000 12.10.2022
11	263	TAP_M6	Not found	"TAP_M6"	29	15:40:50.000 12.10.2022
12	5	DRILL_D5	OK	"DRILL_D5"	39	15:41:19.000 12.10.2022
13	203	NC_SPOT_DRILL_D10	Not found	"NC_SPOT_DRILL_D10"	17	15:41:58.000 12.10.2022
14	263	TAP_M6	Not found	"TAP_M6"	29	15:42:15.000 12.10.2022
15	5	DRILL_D5	OK	"DRILL_D5"	39	15:42:44.000 12.10.2022

Extended Tool Management	Option 93	ID 676938-01
TNC7	NC SW 81762x-16 or later	
TNC 640	NC SW 34059x-01 or later	
TNC 620	NC SW 81760x-03 or later	
TNC 320	NC SW 771851-02 or later	

Installation by the machine manufacturer

For more information, see the TNC brochures, and visit www.klartext-portal.com

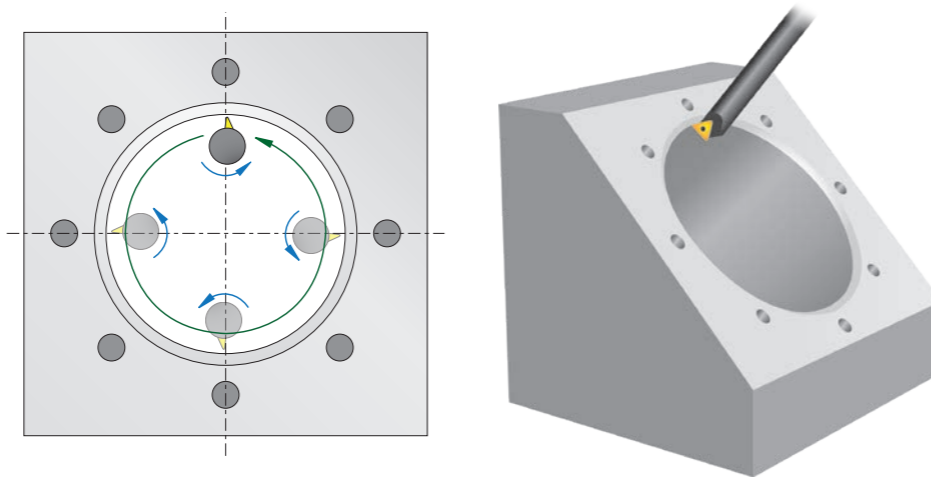
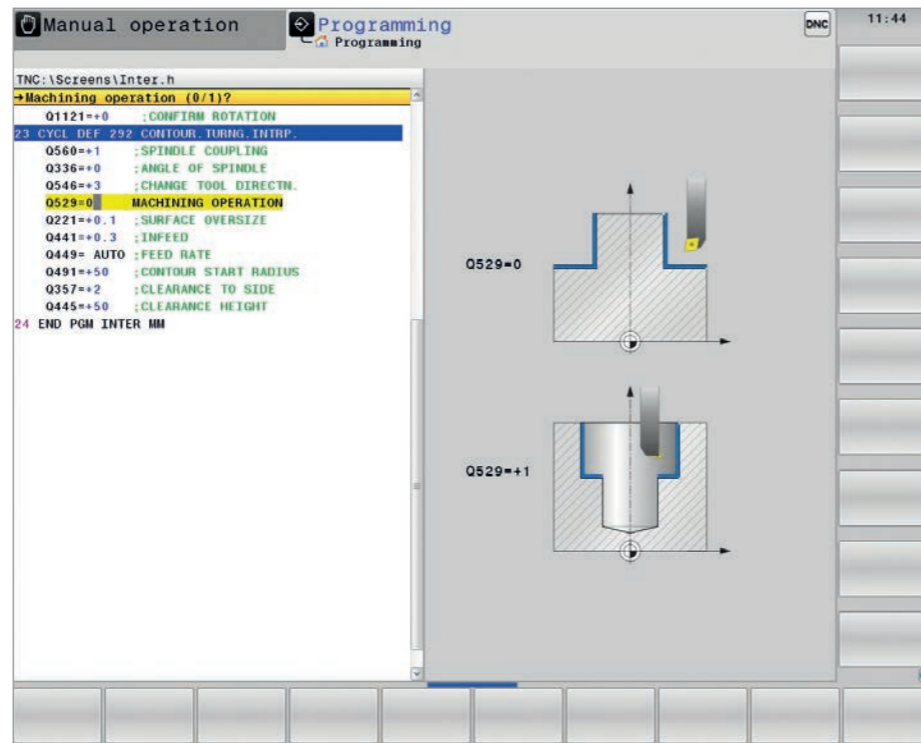
Programming and operation

Interpolating spindle: interpolation turning

In interpolation turning, the tool's cutting edge describes a circular motion, with the cutting edge always oriented either toward the center of the circle (outside machining) or away from the center (inside machining). By varying the circle radius and the axial position, any rotationally symmetrical object can be produced in any working plane.

With the interpolation turning cycle, the TNC can create a rotationally symmetrical shoulder defined by its start and end point in the active working plane. The center of rotation is the starting point in the working plane when the cycle is called. The rotational surfaces can be inclined or rounded away from each other.

This cycle can be used only for finishing work and does not permit roughing operations with multiple passes. The machining strategy can be selected as needed: both inside machining and outside machining are possible. The TNC7 and TNC 640 also let you machine any rotationally symmetrical contour (without undercuts).



Advanced Spindle Interpolation	Option 96	ID 751653-01
TNC7	NC SW 81762x-16 or later	
TNC 640	NC SW 34059x-05 or later	
TNC 620	–	
TNC 320	–	
Installation by the machine manufacturer		
For more information , see the TNC brochures, and visit www.klartext-portal.com		

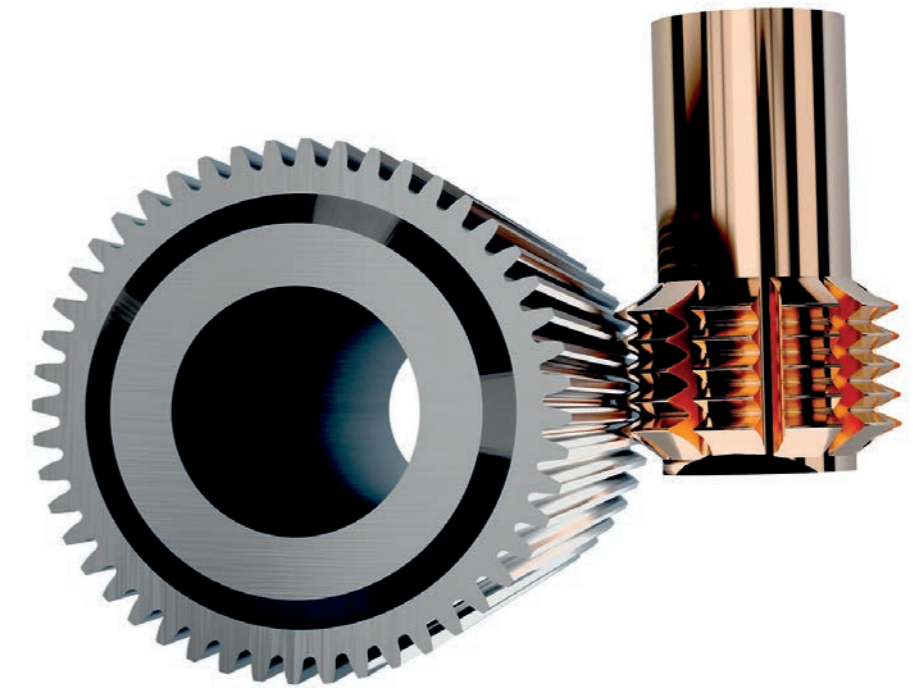
Programming and operation

Spindle synchronization

For certain machining operations, the rotations of the tool spindle must be synchronized with the movement of other axes. This is the case, for example, during the production of external gears via hobbing.

In connection with option 50 (Turning) or option 158 (Turning v2) and option 131 (Spindle Synchronism), the control offers Cycle 880 GEAR HOBGING, which allows you to machine external cylindrical gears or helical gears at any angle. During hobbing, the rotation of the tool spindle and rotary table are synchronized while the gear hob moves axially along the workpiece. Cycle 880 automatically controls these complex movements and lets you easily enter all of the relevant values. You can utilize the tooth parameters straight out of your drawing, which the cycle then uses to calculate the required five-axis motion.

The TNC 640 (with NC SW 34059x-11 or later) and TNC7 controls support tapping with one or more spindles. The spindles can be coupled for this purpose. Tapping is performed with the master spindle. The coupled slave spindles follow the machining operation.



Spindle Synchronism	Option 131	ID 806270-01
TNC7	NC SW 81762x-16 or later	
TNC 640	NC SW 34059x-05 or later	
TNC 620	–	
TNC 320	–	
Installation by the machine manufacturer		
For more information , see the TNC brochures		

Programming and operation

Grinding and dressing functions

With the TNC7 and TNC 640, you can even perform grinding operations on your machine. Cycles for jig grinding and dressing allow you to program the respective function with great convenience. The TNC7 and TNC 640 can also superimpose a tool-axis reciprocating stroke onto the programmed movements. Optimized tool management supports the operator during every process, including grinding and dressing. The TNC7 and TNC 640 are therefore the ideal basis for attaining exceptional surface quality and accuracy in your applications.

Jig grinding

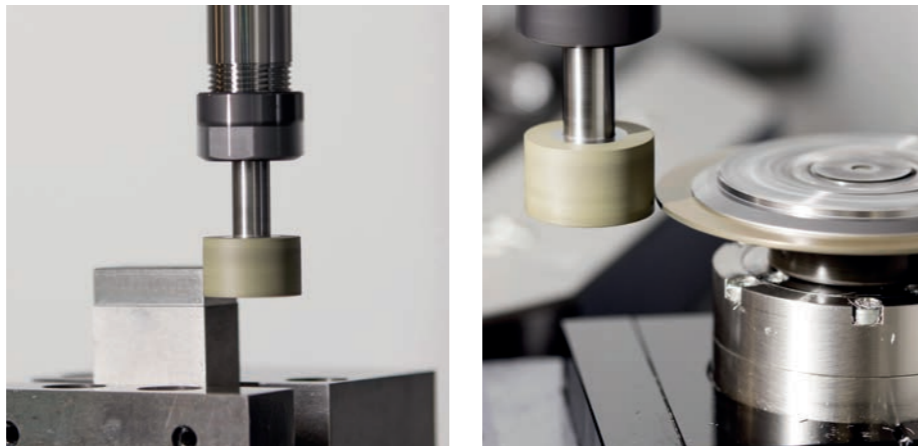
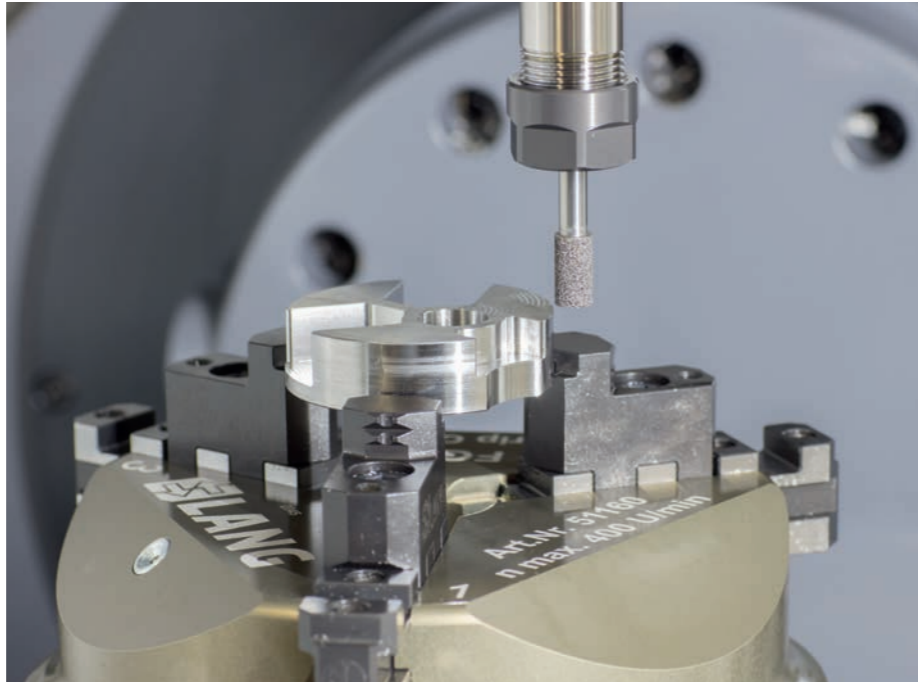
Jig grinding is the grinding of a 2D contour. On a milling machine, jig grinding is primarily used for finishing holes or pre-machined contours using a grinding tool. The grinding contour can be programmed in the defined grinding cycles and then machined.

Grinding open and closed contours

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.



Grinding	Option 156	ID 1237232-01
TNC7	NC SW 81762x-16 or later	
TNC 640	NC SW 34059x-10 or later	
TNC 620	–	
TNC 320	–	
Installation by the machine manufacturer		
For more information , see the TNC brochures, and visit www.klartext-portal.com		

Programming and operation

Gear machining in a single setup

Because gears are typically manufactured on specialized machine tools, time-consuming rechucking is often required.

With the Gear Cutting function, however, you can completely machine spur and helical gears in a single setup using skiving or hobbing operations. If you wish to use the Gear Cutting option in turning mode, then option 50 or option 158 (Turning v2) will need to be enabled. The Gear Cutting function helps you program complex sequences by letting you define just the gear geometry and tool data. The control will take care of the remaining calculations, particularly the complex synchronization of motions. As a result, the production of internal gears turns into an easy-to-master standard application.

Skiving

Skiving owes its current success to its significantly higher efficiency and throughput compared with traditional gear shaping.

With skiving, external and internal gears can be produced on machines with synchronized spindles.

Hobbing

Hobbing is particularly well suited to external gear machining. Its benefits include high throughput and a wide variety of tooth shapes that can be produced with relatively easy-to-make tools.

Lift-off

This option not only makes programming easier but also provides better protection for your equipment. To prevent damage during unforeseen program interruptions (e.g., a power outage), its cycles support optimized lift-off capability. These cycles automatically define the tool direction and path for retraction from the workpiece.



Gear Cutting	Option 157	ID 1237235-01
TNC7	NC SW 81762x-16 or later	
TNC 640	NC SW 34059x-09 or later	
TNC 620	–	
TNC 320	–	
Installation by the machine manufacturer		
For more information , see the TNC brochures, and visit www.klartext-portal.com		

Machine accuracy

KinematicsOpt: easy calibration of rotary axes

Accuracy requirements are becoming increasingly stringent, particularly when it comes to five-axis machining. Complex parts are expected to be machined to exact specifications with reproducible accuracy over extended periods.

The **KinematicsOpt** function for TNC controls is a key ingredient in helping you meet these high demands. First, a probing cycle automatically measures your machine's rotary axes with a HEIDENHAIN touch probe.

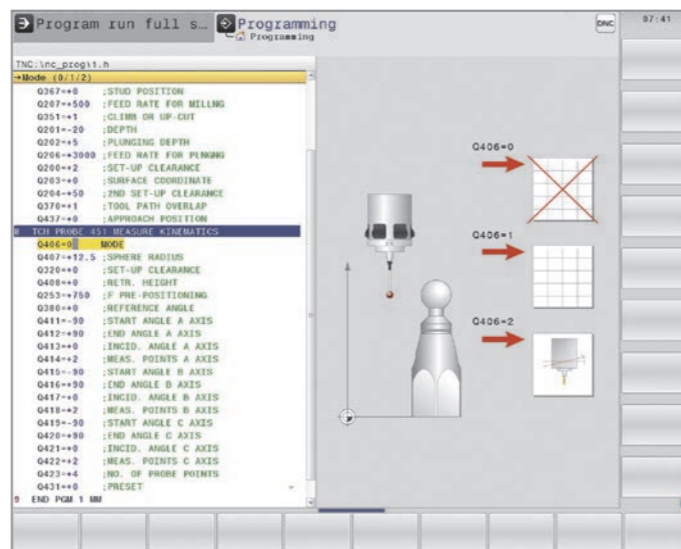
To measure the rotary axes, a calibration sphere is fixed on the machine table and probed with the HEIDENHAIN touch probe. Prior to probing, you specify the touch point grid and define a different measuring range for each rotary axis.

Based on the measured values, the TNC determines the spatial errors that arise from axis tilting. The cycle then calculates an optimized kinematic machine description that minimizes these errors and saves it as the machine's kinematic model.

Of course, a detailed log file is also available containing the actual measured values, the measured and optimized dispersion (for the static tilting accuracy), and the actual compensation values. Together with option 52 (KinematicsComp), the KinematicsOpt option permits compensation for spatial misalignment of the rotary axes and makes 3D compensation of rotary swivel heads on a rotary axis grid possible.

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

Calibration spheres are available as accessories:
 KKH 80 Height: 80 mm ID 655475-03
 KKH 250 Height: 250 mm ID 655475-01



KinematicsOpt	Option 48	ID 630916-01
TNC7	NC SW 81762x-16 or later	
TNC 640	NC SW 34059x-01 or later	
TNC 620	NC SW 34056x-03/73498x-01/81760x-01 or later	
TNC 320	—	
Installation by the machine manufacturer		
For more information , see the TNC brochures, and visit www.klartext-portal.com		

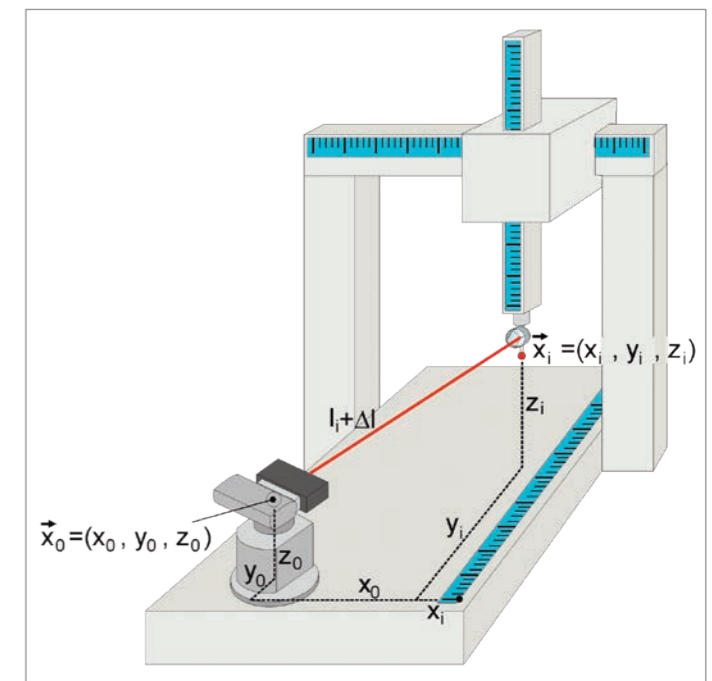
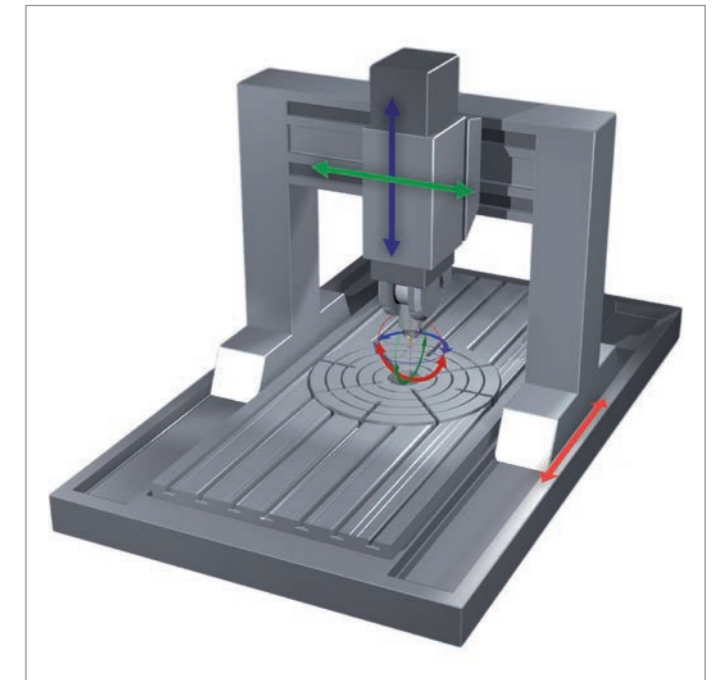
Machine accuracy

KinematicsComp: 3D spatial compensation

Tight workpiece tolerances require high machine accuracy. However, machine tools inevitably exhibit installation- or production-related errors.

The more axes a machine has, the more sources of error there are. The ISO 230-1 standard, for example, lists eight relevant types of error for a linear axis (six component errors and two relevant position errors) and eleven for a rotary axis (six component errors and five relevant position errors). Counteracting such errors at the mechanical level requires extraordinary design effort. Moreover, these errors are particularly noticeable on five-axis machines and very large machines. What should also not be neglected is thermal expansion, which can cause highly complex changes in the geometry of machine components.

The **KinematicsComp** function gives the machine manufacturer a way to significantly improve his machine's accuracy. The TNC's standard kinematic model describes the machine's degrees of freedom and the rotational centers of its rotary axes. The enhanced kinematics description of KinematicsComp additionally includes the option of considering positioning, straightness and angular errors of all axes through compensation value tables. In addition to featuring multidimensional tables, KinematicsComp also uses formulas with which even errors beyond the scope of the rigid body error model can be compensated for. This is done so well that the tool center point (TCP) can exactly follow the ideal nominal contour. Even thermally induced errors can be compensated for after being detected by sensors and the PLC. The spatial errors of the tool center point, for example, can be measured with a laser tracer or laser interferometer and converted into compensation tables. Option 48 (KinematicsOpt) uses KinematicsComp to compensate for the misalignment of rotary axes and for the grid compensation of rotary swivel heads.



Ascertaining geometric error with a laser coordinate measuring device (source: PTB Notification 117)

KinematicsComp	Option 52	ID 661879-01
TNC7	NC SW 81762x-16 or later	
TNC 640	NC SW 34059x-05 or later	
TNC 620	—	
TNC 320	—	
Installation by the machine manufacturer		
For more information , see the TNC brochures, and visit www.klartext-portal.com		

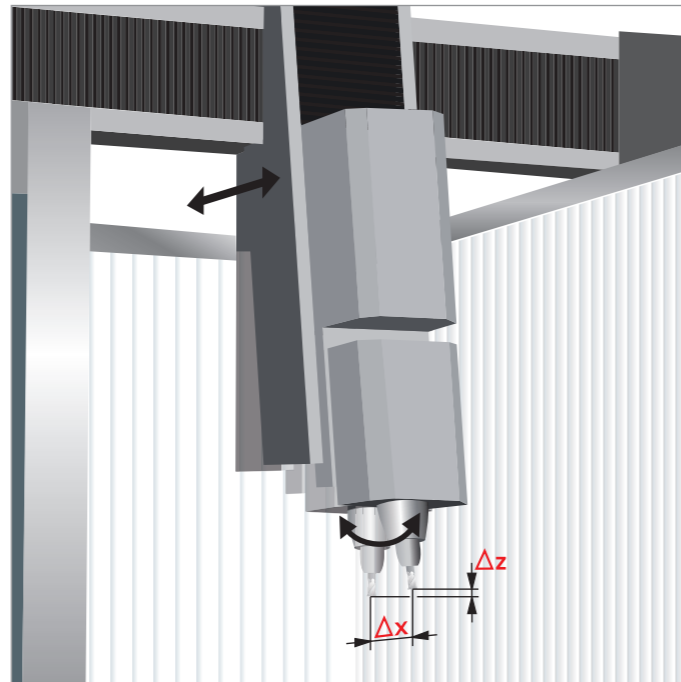
Machine accuracy

CTC: compensation of position errors due to axis couplings

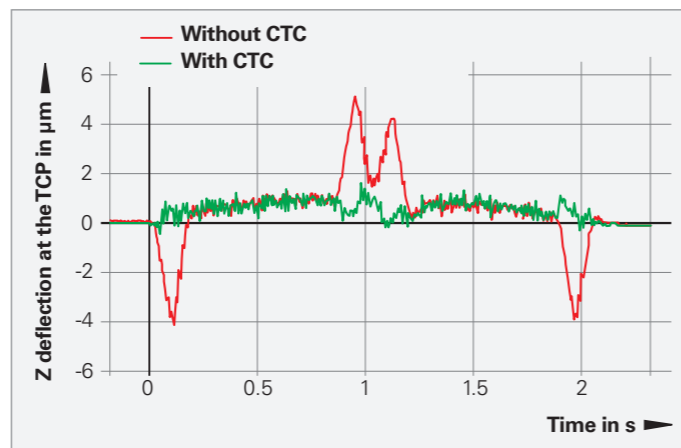
During dynamic acceleration processes, a machine tool's structure undergoes forces capable of deforming machine components, resulting in errors at the tool center point (TCP). The dynamic acceleration of an axis causes not only axial deformations but also deformations that are lateral to the direction of acceleration (due to mechanical axis couplings). This is especially the case if the point of force applied to an axis does not coincide with its center of gravity, thus potentially resulting in pitching motion during braking and acceleration phases. The resulting position error at the TCP in the direction of the accelerated axis and lateral axes is proportional to the amount of acceleration.

If the dynamic position error as a function of the axis acceleration is known from measurements at the TCP, then this acceleration-dependent error can be compensated for with the **CTC** (Cross Talk Compensation) servo-control option in order to prevent negative effects on the surface quality and accuracy of the workpiece.

A grid encoder (KGM) in the plane defined by two mechanically coupled axes can be used to measure the acceleration-dependent position error of these axes. The resulting error at the TCP often depends not only on the acceleration but also on the position of the axes in the working space. The CTC servo-control option can take this into account as well.



Deflection at the TCP in the Z axis during movement in the X direction



Servo control optimized for Z = 0; following error within the tolerance band



Cross Talk Compensation	Option 141	ID 800542-01
TNC7	NC SW 81762x-16 or later	
TNC 640	NC SW 34059x-02 or later	
TNC 620	NC SW 34056x-04/73498x-02/81760x-01 or later	
TNC 320	-	
Installation by the machine manufacturer		
For more information, see the TNC brochures, see the <i>Dynamic Precision</i> Technical Information document, and visit www.klartext-portal.com		

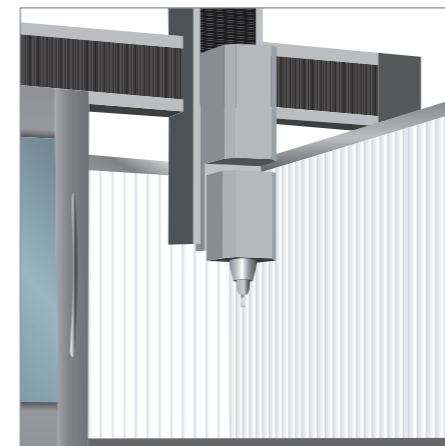
Machine accuracy

PAC: position-dependent adaptation of control parameters

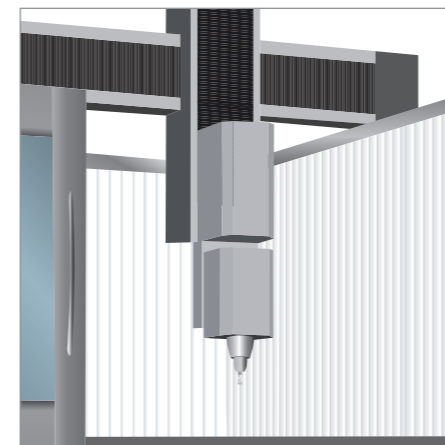
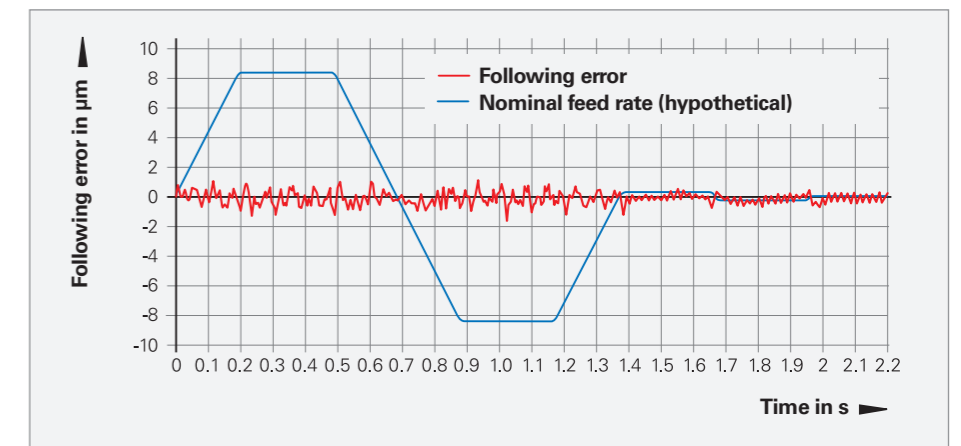
Depending on the positions of the axes in the working space, a machine's kinematics give rise to variable dynamic behavior that can adversely affect servo-control stability.

To fully exploit the machine's dynamic performance, you can use the **PAC** option (Position Adaptive Control) to change machine parameters based on position.

This makes it possible to assign the respective optimal loop gain to defined interpolation points. Additional position-dependent filter parameters can be defined in order to further increase control loop stability.

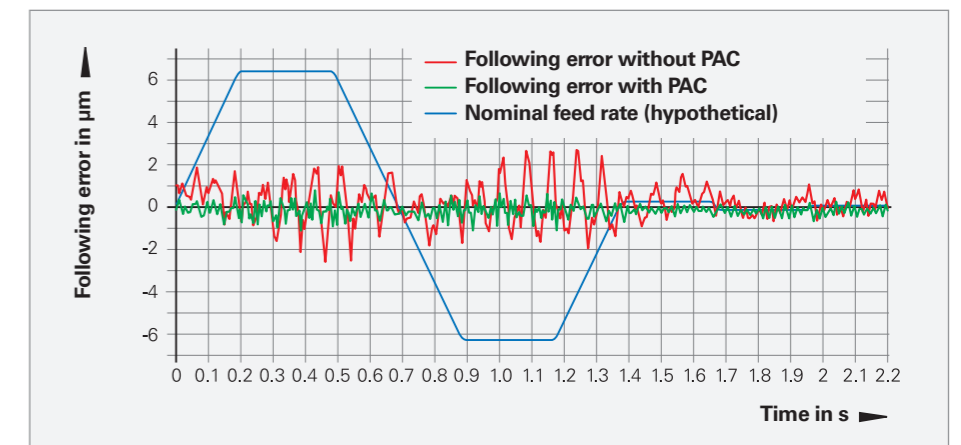


Servo control optimized for Z = 0; following error within the tolerance band ($\pm 1 \mu\text{m}$)



Servo control at Z = -500

- Without PAC: clearly visible oscillations and following error ($\pm 3 \mu\text{m}$) outside of the tolerance band
- With active PAC: following error within the tolerance band ($\pm 1 \mu\text{m}$)



Position Adaptive Control	Option 142	ID 800544-01
TNC7	NC SW 81762x-16 or later	
TNC 640	NC SW 34059x-02 or later	
TNC 620	NC SW 34056x-04/73498x-02/81760x-01 or later	
TNC 320	-	
Installation by the machine manufacturer		
For more information, see the TNC brochures, see the <i>Dynamic Precision</i> Technical Information document, and visit www.klartext-portal.com		

Machine accuracy

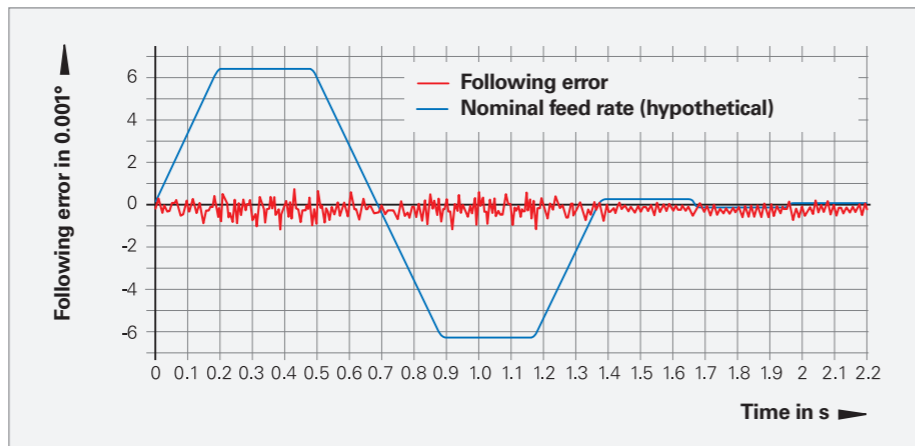
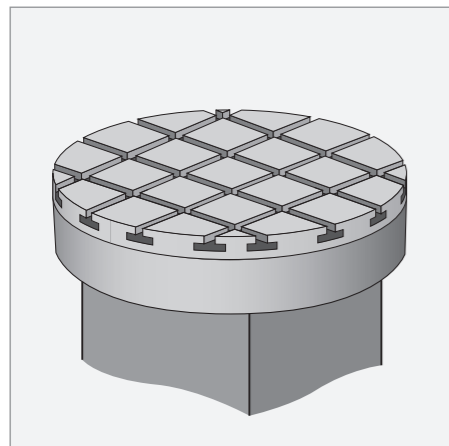
LAC: load-dependent adaptation of control parameters

The dynamic behavior of machines with moving tables can vary depending on the mass or mass moment of inertia of the clamped workpiece.

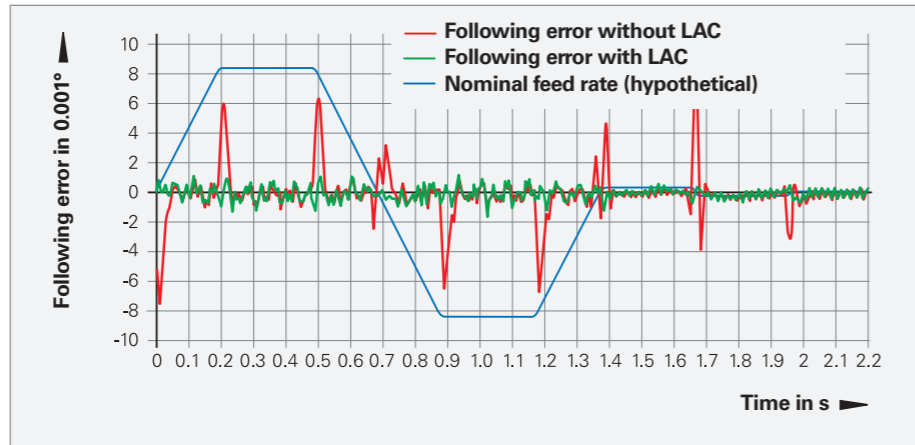
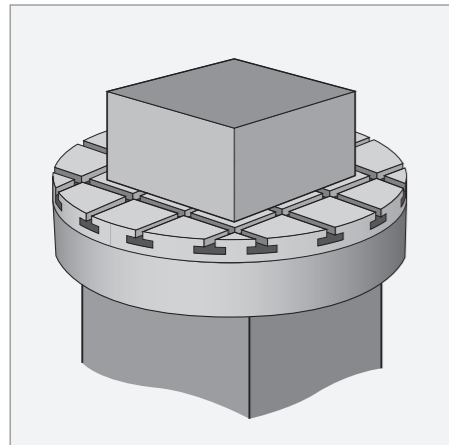
The **LAC** option (Load Adaptive Control) enables the control to automatically ascertain the current workpiece mass or mass moment of inertia and the acting frictional forces. In order to ensure an optimum response to changes in machine

behavior at different loads, adaptive feedforward controls can be activated, and the control loop gain can be changed based on the load.

In order to allow rapid adjustment in response to sudden changes in the load (e.g., from loading and unloading the workpiece), Cycle 239 is provided for ascertaining the current load status.



Optimal feedforward control for rotary tables without additional load and with following error within the tolerance band ($\pm 0.001^\circ$)



- With change in load
- Without LAC: with unchanged feedforward control, the following error ($\pm 0.008^\circ$) is outside of the tolerance band
 - With LAC: with feed-forward control and active LAC, the following error is within the tolerance band ($\pm 0.001^\circ$)

Load Adaptive Control	Option 143	ID 800545-01
TNC7	NC SW 81762x-16 or later	
TNC 640	NC SW 34059x-02 or later	
TNC 620	NC SW 34056x-04/73498x-02/81760x-01 or later	
TNC 320	-	
Installation by the machine manufacturer		
For more information , see the TNC brochures, see the <i>Dynamic Precision</i> Technical Information document, and visit www.klartext-portal.com		



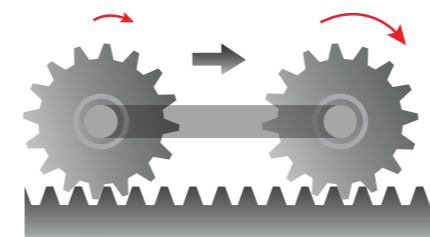
Machine accuracy

MAC: motion-dependent adaptation of control parameters

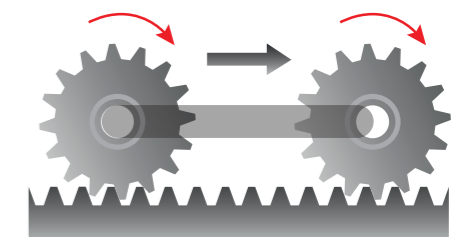
In addition to the position-dependent adaptation of control parameters through the PAC option, the **MAC** option (Motion Adaptive Control) provides a way of changing machine parameters based on other input quantities such as speed or motor acceleration. This motion-dependent adaptation of the control parameters makes it possible, for example, to implement speed-dependent adaptation of the k_v factor for drive systems exhibiting stability changes at different traversing speeds.

A further application is the acceleration-dependent change of the tensioning torque between the master and slave axes for master-slave torque control.

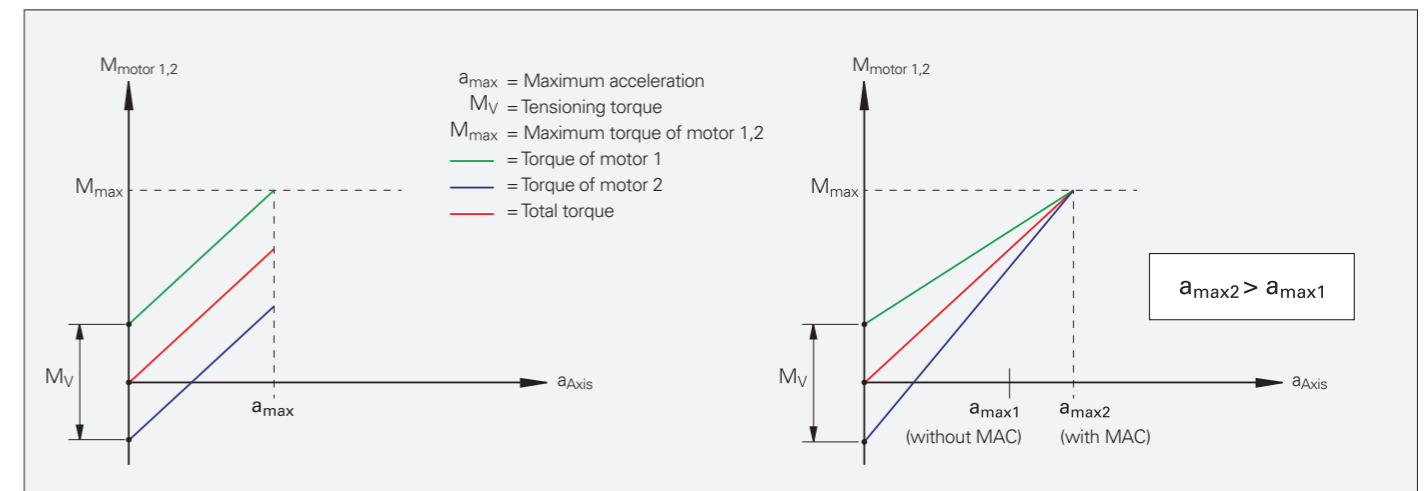
With the MAC option, this configuration makes it possible to attain a significantly higher maximum acceleration at rapid traverse; for example, through parameter-based reduction of the tensioning torque at increasing acceleration.



Without MAC



With MAC



Motion Adaptive Control	Option 144	ID 800546-01
TNC7	NC SW 81762x-16 or later	
TNC 640	NC SW 34059x-02 or later	
TNC 620	NC SW 34056x-04/73498x-02/81760x-01 or later	
TNC 320	-	
Installation by the machine manufacturer		
For more information , see the TNC brochures, see the <i>Dynamic Precision</i> Technical Information document, and visit www.klartext-portal.com		



Machine accuracy

MVC: active vibration damping

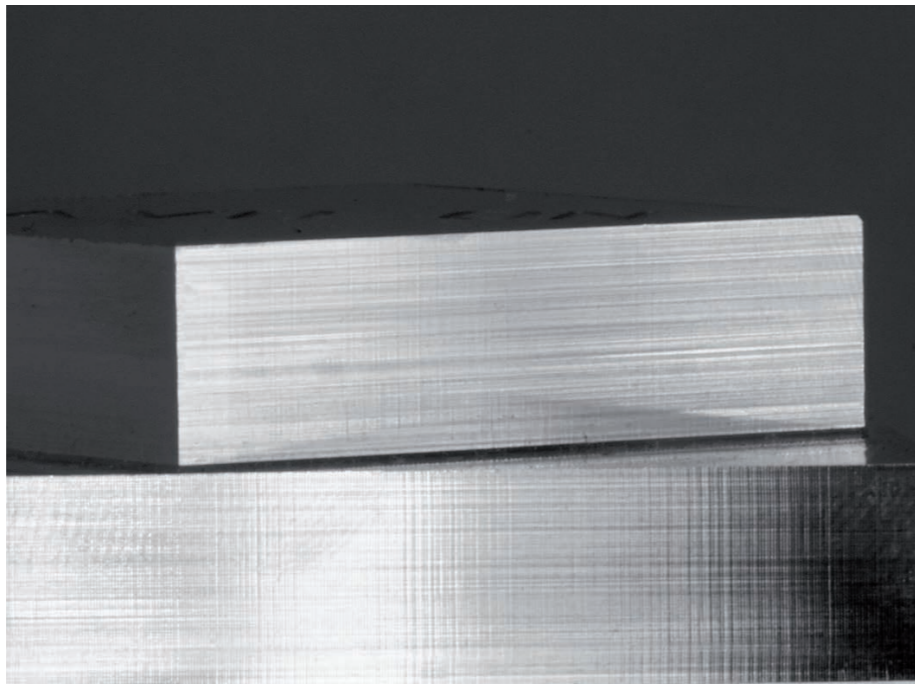
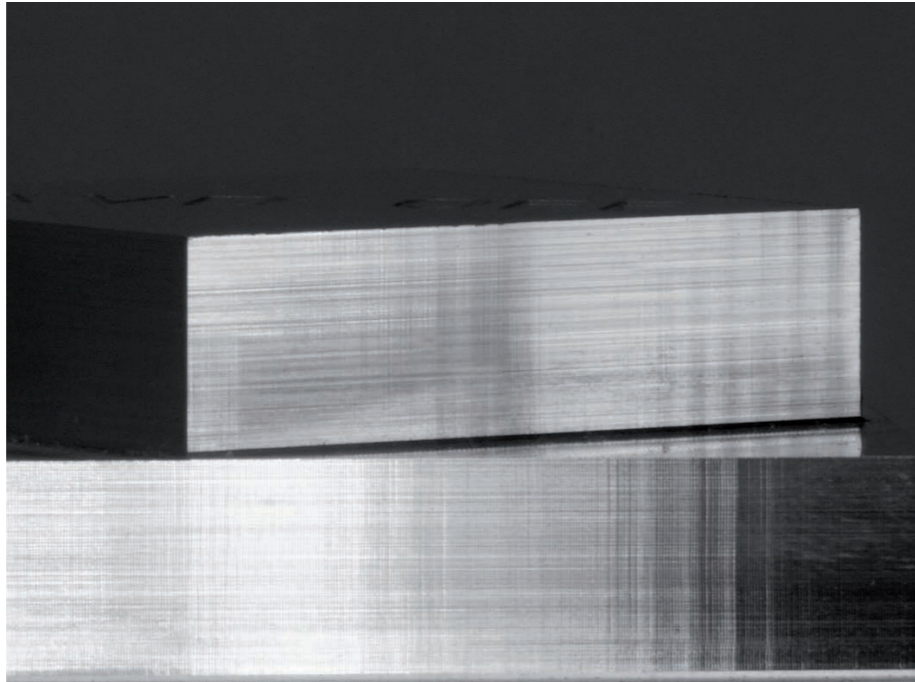
In the presence of low-frequency vibrations on a machine tool, inclined or curved surfaces can often exhibit surface-finish problems in the form of visible shadows or inconsistent contrast. Peaks as small as 1 µm or smaller may be visually noticeable on a workpiece. These disturbances often necessitate surface rework, which comes with additional cost.

Low-frequency disturbances are often caused by elasticity in the drive train, such as vibrations between the motor and the slides, or machine setup vibrations, in which high accelerations of the machine axes cause disturbances in the machine tool through its fastening elements or connected base.

Excitations due to high accelerations can be lowered by reducing the jerk, but this results in longer machining times.

The Machine Vibration Control (MVC) option suppresses low-frequency vibrations via the control loop of the control. To this end, the Active Vibration Damping (AVD) and Frequency Shaping Control (FSC) functions are available with this option. MVC works in two ways: it causes a clean workpiece surface because any vibrations that would be visible on the workpiece surface are suppressed, and it also enables fast and low-vibration milling.

MVC thus increases the productivity of a machine tool and/or improves the surface quality of workpieces.



Machine Vibration Control	Option 146	ID 800548-01
TNC7	NC SW 81762x-16 or later	
TNC 640	NC SW 34059x-04 or later	
TNC 620	NC SW 34056x-04/73498x-02/81760x-01 or later	
TNC 320	–	
Installation by the machine manufacturer		
For more information , see the TNC brochures, see the <i>Dynamic Precision</i> Technical Information document, and visit www.klartext-portal.com		

dynamic + **precision**

Machining functions

5-axis simultaneous machining

The TNC provides many powerful functions specifically developed for five-axis simultaneous machining.

The NC programs for five-axis simultaneous machining are created with CAM systems in conjunction with postprocessors. In principle, such programs contain either all of the coordinates of the machine's existing NC axes or NC blocks with surface normal vectors. During five-axis machining with three linear axes and two additional tilting axes, the tool is always perpendicular to the workpiece surface or is tilted at a specific angle relative to it (inclined tool machining).

Regardless of the type of five-axis programs you wish to run, the TNC performs all of the necessary compensating movements in the linear axes arising from movements in the tilting axes. The TNC's tool center point management feature (TCPM)—an improvement on the proven TNC function M128—provides optimal tool guidance and prevents contour gouging.

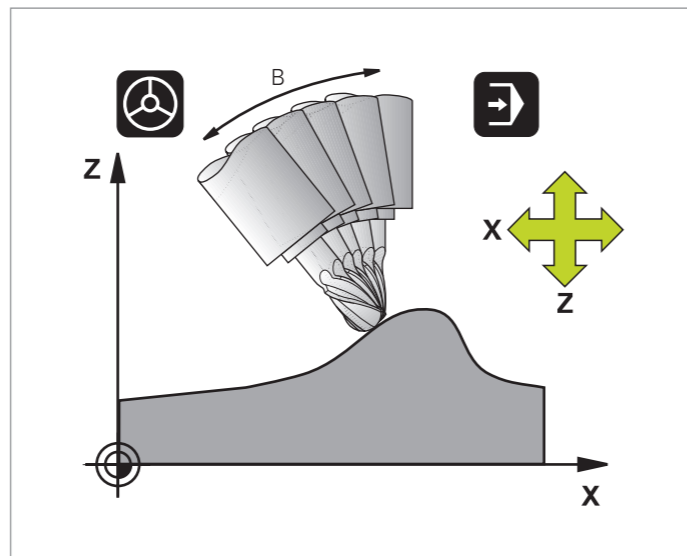
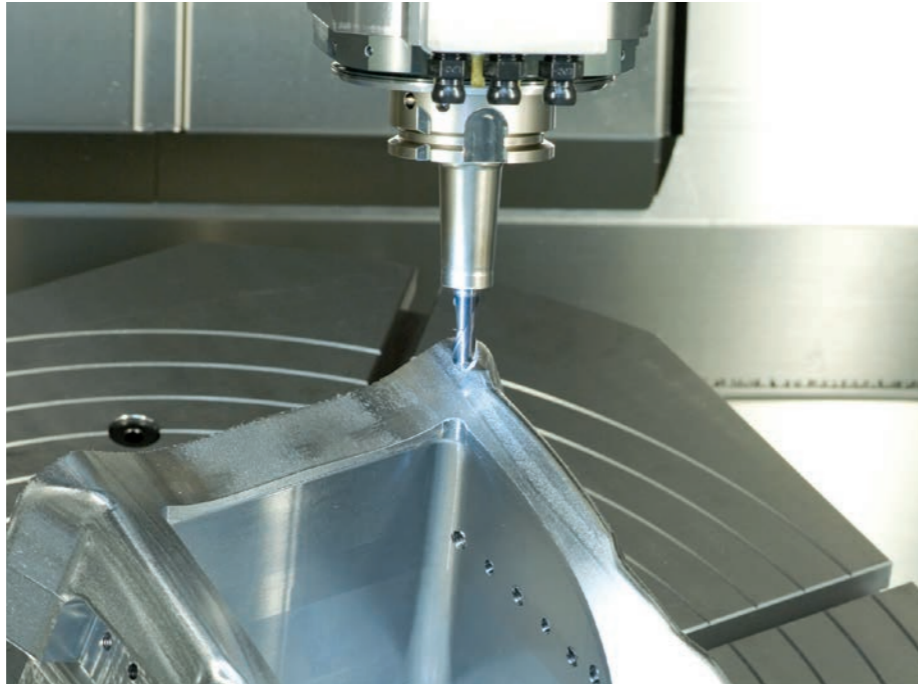


Advanced Function Set 2	Option 9	ID 617921-01
TNC7	NC SW 81762x-16 or later	
TNC 640	NC SW 34059x-01 or later	
TNC 620	NC SW 34056x-01/73498x-01/81760x-01 or later	
TNC 320	–	
Installation by the machine manufacturer		
For more information , see the TNC brochures		

Machining functions

Handwheel superimposition: superimpose handwheel positioning during program run

The **Handwheel Superimposition** function (M118) enables you to make manual corrections with the handwheel during program run. This is particularly helpful when you wish to change the inclination angles of rotary axes: in externally written NC programs such changes often cause collisions between the tilting head and the workpiece. You can also use handwheel superimposition to adjust the offset compensation in linear axes without having to change the NC program.

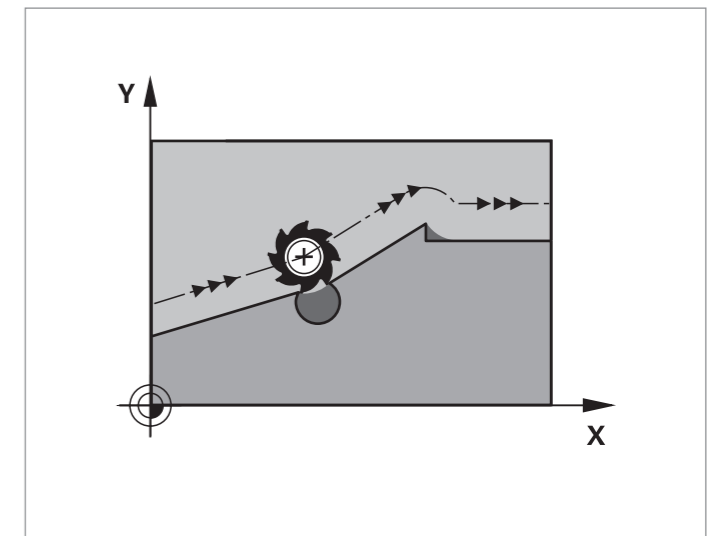
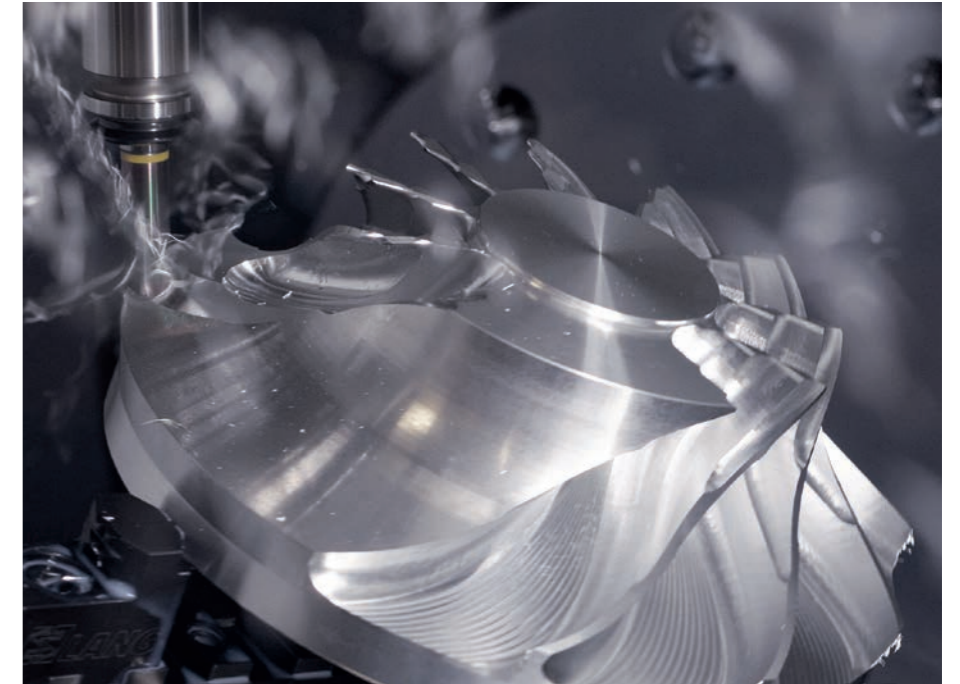


Advanced Function Set 3	Option 21	ID 628254-01
TNC7	Standard function	
TNC 640	Standard function	
TNC 620	NC SW 34056x-01/73498x-01/81760x-01 or later	
TNC 320	Standard function	
Installation by the machine manufacturer		
For more information, see the TNC brochures		

Machining functions

Tool compensation: radius-compensated contour precalculation (LOOK AHEAD)

The LOOK AHEAD function in the TNC's geometry processing system monitors radius-compensated contours for undercuts and intersections, and calculates the tool path in advance starting from the current block. Sections of the contour that would be damaged by the tool are simply not machined (dark areas in the figure) and can be reworked with a smaller tool at a later time. You can also use this function to add tool radius compensation to NC programs created with an external programming system and output as an uncompensated contour. Consequently, inaccuracies in the NC programs arising from calculations in the CAM system can be compensated for.



Advanced Function Set 3	Option 21	ID 628254-01
TNC7	Standard function	
TNC 640	Standard function	
TNC 620	NC SW 34056x-01/73498x-01/81760x-01 or later	
TNC 320	Standard function	
Installation by the machine manufacturer		
For more information, see the TNC brochures		

Machining functions

Dynamic Collision Monitoring (DCM)

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

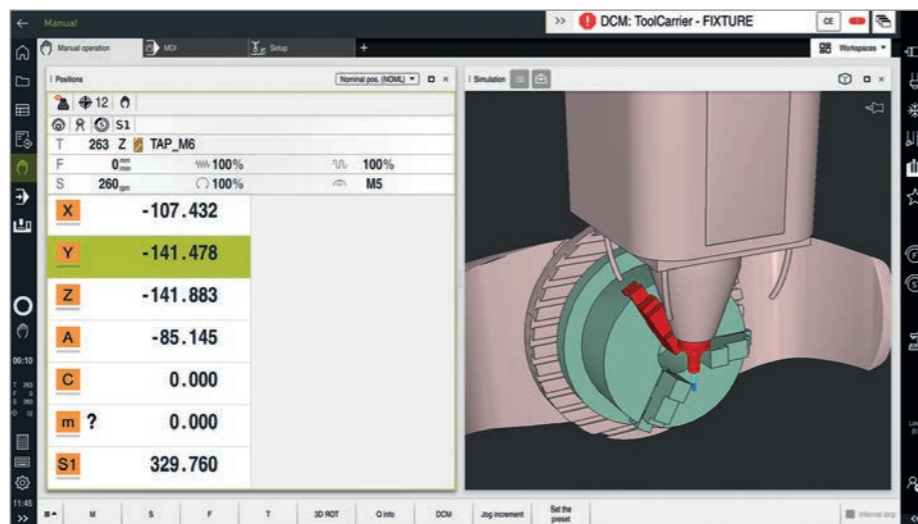
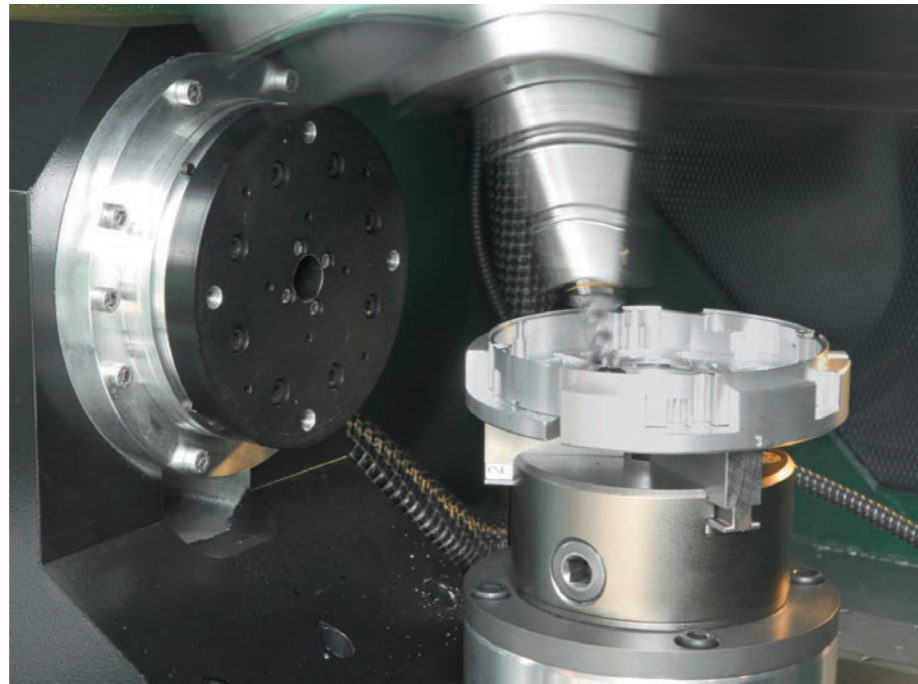
In these cases, the machine operator is supported by the TNC's **Dynamic Collision Monitoring (DCM)** option. The control interrupts the machining process whenever a collision is imminent, thereby improving safety for both the machine and its operator. This, in turn, helps prevent machine damage and any resulting costly downtime. Unattended shifts also become more reliable.

The DCM option works not only in **automatic mode** but is also active in **manual mode**. If, for example, the machine operator is on a collision course during setup, the TNC detects this, stops axis movement, and issues an error message. Before actually machining a part, you can also check for collisions in the simulation, with a real preset and real tools.

Of course, the TNC also shows the machine operator which machine components are at risk, both by means of an error message and graphically. If a collision warning occurs, the TNC permits tool retraction only in directions that will increase the clearance between the colliding objects.

The TNC 640 (with NC SW 34059x-05 or later) and TNC7 controls also offer a convenient way to import collision objects from standard CAD models (e.g., STL) into the control as M3D data. This permits highly detailed modeling of the machine components and better utilization of the machine's working space.

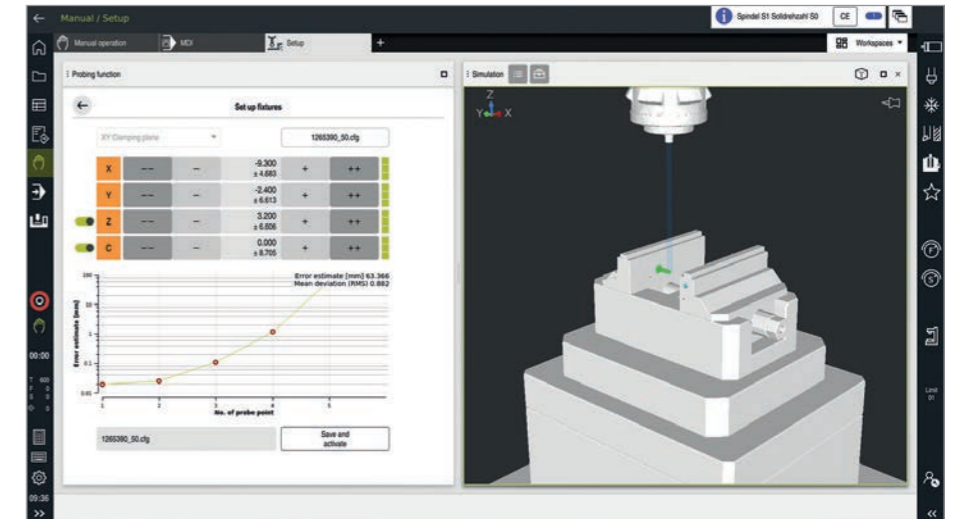
On the TNC 640 (with NC SW 34059x-11 or later) and the TNC7, clamping setups can be incorporated in *.cfg, *.m3d, or *.stl formats, including via Klartext commands.



Machining functions

Dynamic Collision Monitoring version 2 (DCM v2)

DCMv2 includes all features of option 40. The expanded collision monitoring of the TNC7 not only prevents tools from colliding with machine components, but also with fixtures. The TNC7 features a probing function that graphically and interactively assists you in determining the exact position of the fixture. That way you no longer have to worry about either the sequence or the actual probing functions. Colored arrows in a virtualized work envelope show you whether the position is suitable and the probing operation can be performed. This graphically supported measurement can be used for any workholding equipment. An accurate 3D model is all that is needed.



Dynamic Collision Monitoring	Option 40	ID 526452-01
TNC7	NC SW 81762x-16 or later	
TNC 640	NC SW 34059x-02 or later	
TNC 620	—	
TNC 320	—	
Installation by the machine manufacturer		
For more information , see the TNC brochures, and visit www.klartext-portal.com		

DCMv2	Option 140	ID 1353266-01
TNC7	NC SW 81762x-16 or later	
TNC 640	—	
TNC 620	—	
TNC 320	—	
Installation by the machine manufacturer		
For more information , see the TNC7 brochure, and visit www.klartext-portal.com		

Machining functions

Global Program Settings

The global program settings, which come into play particularly in large-scale mold making, are available in the Program Run and MDI modes. This option lets you define global and overriding coordinate transformations and settings for the NC program without actually changing the NC program.

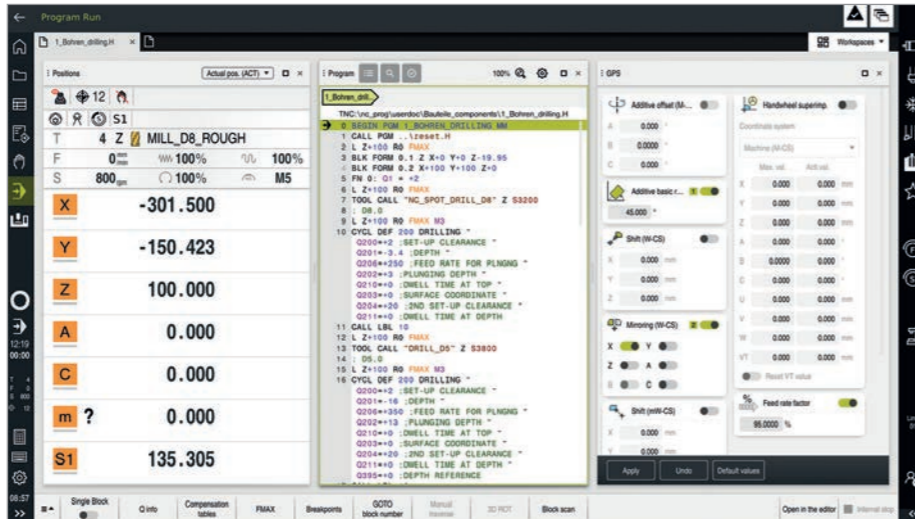
You can change the Global Program Settings during a program stop, even mid-program. A clearly structured form is provided for this. Upon starting, the TNC then moves as needed to a new position in accordance with a positioning logic that you can define. The following functions are available:

- Additional rotation based on basic rotation or 3D basic rotation
- Shifting the workpiece preset in a single axis
- Mirroring individual axes
- Additional shift of an already shifted workpiece datum
- Rotation about the active tool axis
- Handwheel superimposition
- Feed rate factor

Handwheel superimposition is possible in various coordinate systems:

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

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



Global PGM Settings	Option 44	ID 576057-01
TNC7	NC SW 81762x-16 or later	
TNC 640	NC SW 34059x-08 or later	
TNC 620	-	
TNC 320	-	
Installation by the machine manufacturer		
For more information , see the TNC brochures		

Machining functions

Adaptive Feed Control (AFC)

Adaptive Feed Control (AFC) automatically regulates the feed rate of the TNC based on the present spindle power and other process data. During a teach-in cut, the TNC records the maximum occurring spindle power. Prior to machining, you define the limits (in a table) within which the TNC is allowed to influence the feed rate in "control" mode. Naturally, various overload reactions can be specified as well, including by your machine manufacturer.

The TNC's Adaptive Feed Control option offers a series of benefits:

Optimization of the machining time

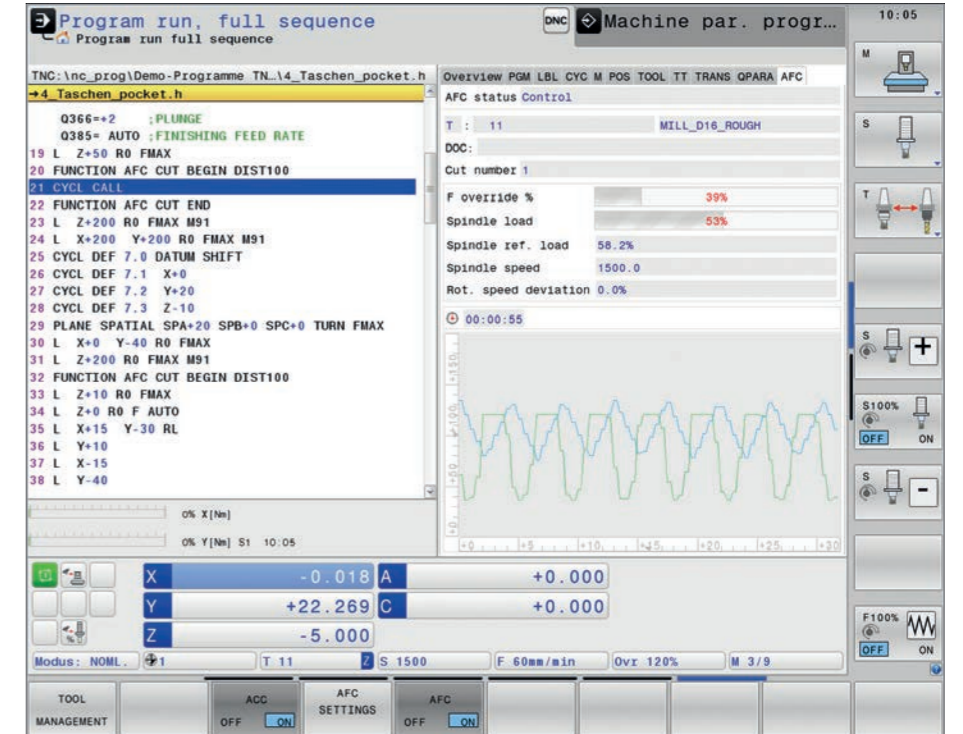
Particularly in the case of cast parts, oversizes and material irregularities (cavities) can occur to varying degrees. By controlling the feed rate, the control strives to stay within the bounds of the previously "learned" maximum spindle power throughout the entire machining time. Increasing the feed rate in areas requiring less material removal shortens the overall machining time.

Tool monitoring

The Adaptive Feed Control option continuously compares the spindle power with the feed rate. As a tool becomes blunt, the spindle power increases, causing the TNC to reduce the feed rate. As soon as the feed rate falls below a specified minimum, the TNC reacts with an NC stop or warning, or automatically replaces the tool. This prevents secondary damage after tool breakage or wear.

Protecting machine components

When the feed rate is reduced in response to the exceeded learned maximum spindle power, machine components undergo less strain. This protects the main spindle from overloading.



Adaptive Feed Control (AFC)	Option 45	ID 579648-01
TNC7	NC SW 81762x-16 or later	
TNC 640	NC SW 34059x-02 or later	
TNC 620 / TNC 320	-	
Installation by the machine manufacturer		
For more information , see the TNC brochures, see the <i>Dynamic Efficiency</i> Technical Information document, and visit www.klartext-portal.com		

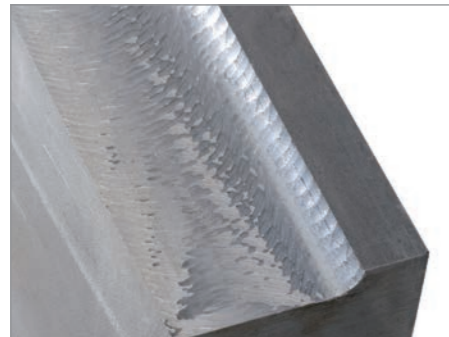


Machining functions

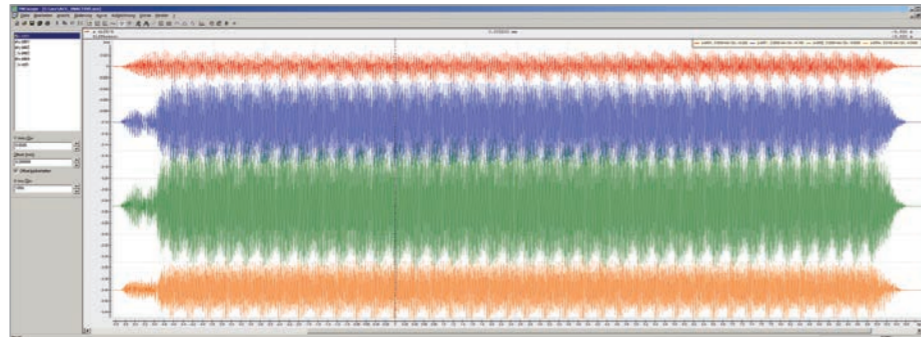
Active Chatter Control (ACC)

Strong forces come into play during roughing (power milling). Depending on the rotational tool speed, the resonances in the machine tool, and the chip volume (metal-removal rate during milling), a phenomenon known as "chatter" may occur. Chatter induces heavy strain on the machine and causes ugly marks on the workpiece surface. It also increases the progress and irregularity of tool wear, even leading to tool breakage under extreme circumstances.

With **Active Chatter Control (ACC)**, HEIDENHAIN now provides an effective control function for reducing a machine's tendency to chatter. This control function is particularly advantageous when used during heavy machining. Thanks to ACC, substantially higher removal rates are possible. Depending on the type of machine, the removal rate can be increased by up to 25 percent or more. At the same time, you are reducing strain on the machine and increasing tool life.



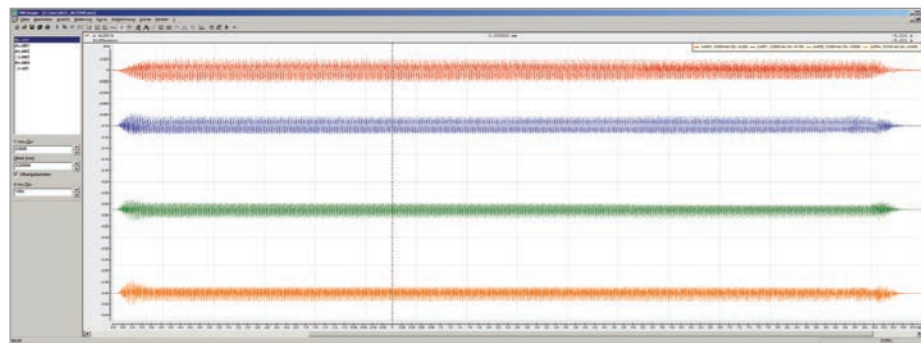
Heavy machining without ACC



Following error without ACC



Heavy machining with ACC



Reduced following error with ACC

Active Chatter Control	Option 145	ID 800547-01
TNC7	NC SW 81762x-16 or later	
TNC 640	NC SW 34059x-02 or later	
TNC 620	NC SW 34056x-04/73498x-02/81760x-01 or later	
TNC 320	-	
Installation by the machine manufacturer		
For more information , see the <i>Dynamic Efficiency</i> Technical Information document, and visit www.klartext-portal.com		

dynamic + **efficiency**

Machining functions

Optimized Contour Milling (OCM)

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

Process reliability and maximum removal rates require optimal adaptation of the cutting data to the characteristics of the tool and workpiece material. For this purpose, **Optimized Contour Milling (OCM)** provides a cutting data calculator that draws on an extensive integrated material database. The operator 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 rates)

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

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

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

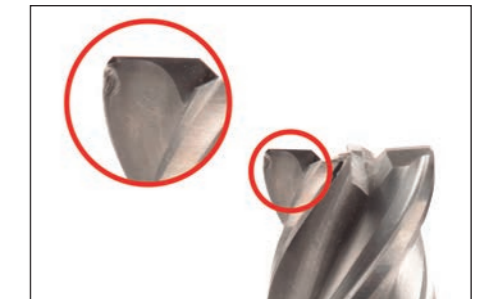
dynamic + **efficiency**

The TNC 640 (with NC SW 34059x-11 or later), the TNC 620 (with NC SW 81760x-07 or later) and the TNC7 provide OCM cycles for chamfering or deburring and for standard shapes.

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

Conventional machining
<i>S5000, F1200, a_p: 5.5 mm</i>
<i>Overlap factor: 5 mm</i>
<i>Machining time: 21 min 35 s</i>

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



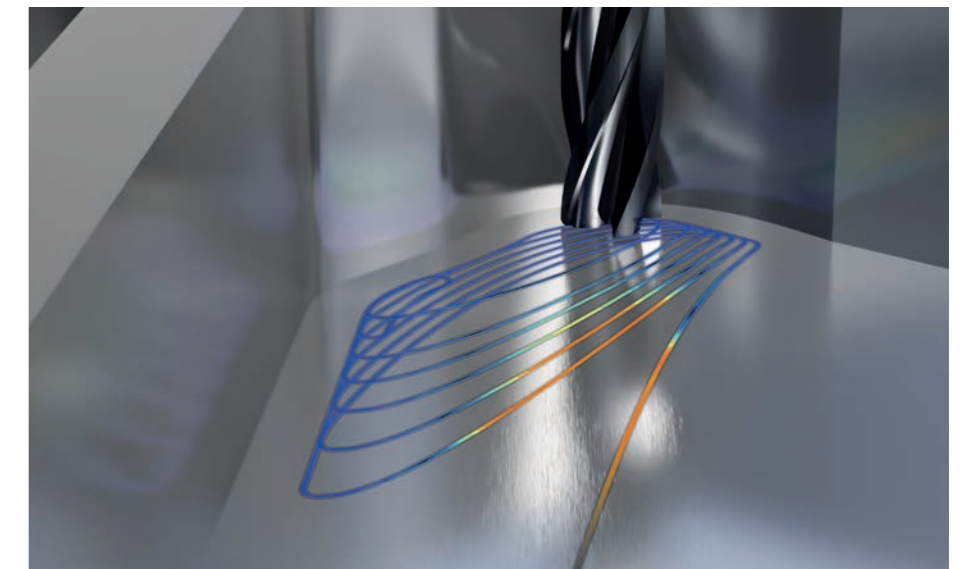
Tool after two parts

Machining with OCM
<i>S8000, F4800, a_p: 22 mm</i>
<i>Overlap factor: 1.4 mm</i>
<i>Machining time: 6 min 59 s</i>

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



Tool after six parts



Optimized Contour Milling	Option 167	ID 1289547-01
TNC7	NC SW 81762x-16 or later	
TNC 640	NC SW 34059x-10 or later	
TNC 620	NC SW 81760x-07 or later	
TNC 320	-	
Installation by the machine manufacturer		
For more information , see the TNC brochures, and visit www.klartext-portal.com		

Machining functions

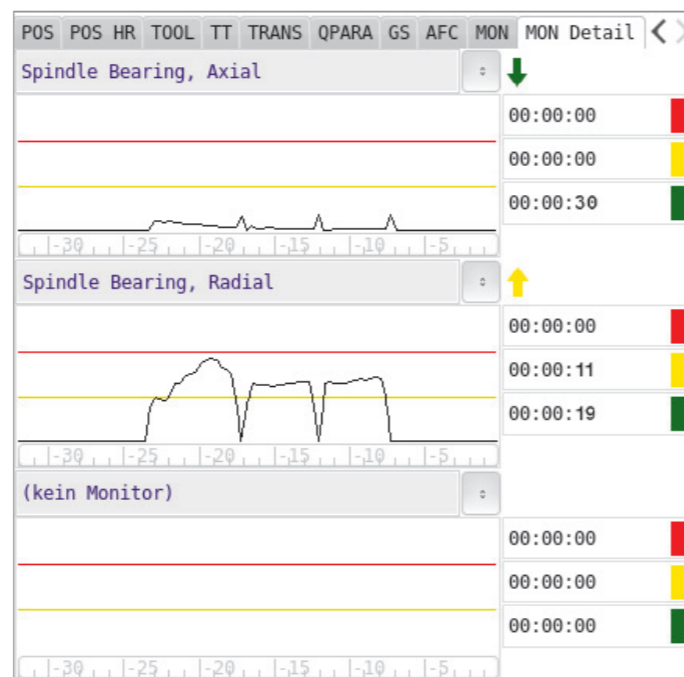
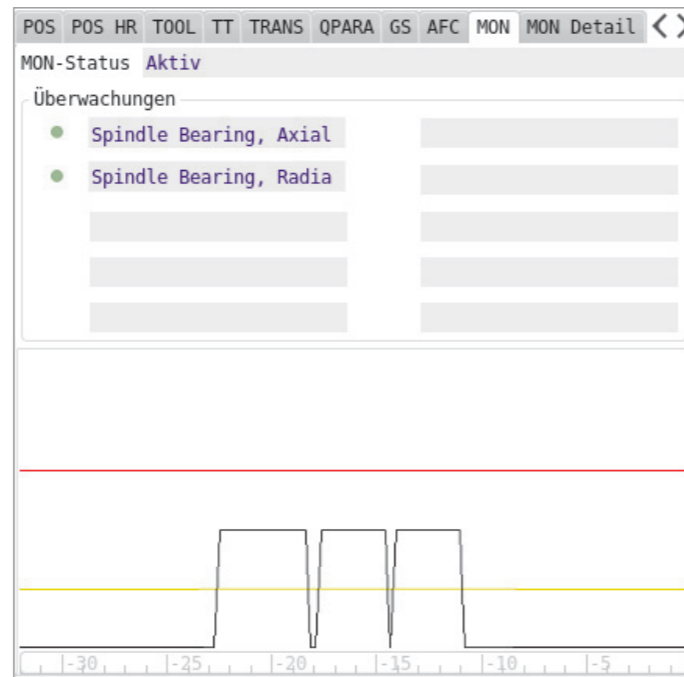
Component Monitoring

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 can detect these dangers, and issue warnings or 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 recirculating ball screws, are subject to wear and thus have an influence on the finished product. The TNC7 uses the Component Monitoring option to ascertain 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 allows you to individually color, from within the NC program, the workpiece view of the concurrent simulation. That way you quickly see in the workpiece view where components were subject to a strong load.

Cycle 238, MEASURE MACHINE STATUS*, is part of software option 155 (Component Monitoring). This cycle allows you to ascertain and record the current machine status. Through data comparison, deviations in machine parameters can be detected and documented over extended periods of time, allowing you to track the machine aging processes.

* Available on the following controls: TNC7, TNC 640 (with NC SW 34059x-10 or later), TNC 620 (with NC SW 81760x-07 or later)



Component Monitoring	Option 155	ID 1226833-01
TNC7	NC SW 81762x-16 or later	
TNC 640	NC SW 34059x-09 or later	
TNC 620	NC SW 81760x-06 or later	
TNC 320	—	
Installation by the machine manufacturer		
For more information , see the <i>Information for the Machine Tool Builder</i> brochures		

Machining functions

Process Monitoring

The fully-integrated process monitoring capability of the TNC7 is a unique feature for making production processes safer and more reliable. Reliable process monitoring is essential for avoiding damage resulting from unavoidable disturbances during automated production. For example, if breakage inspection is performed for tools during the machining operations, then valuable productive time is lost. Here the operator can save time and costs with the fully-integrated process monitoring feature from HEIDENHAIN. Process monitoring entails the recording of one or more reference machining operations. During subsequent operations, the control monitors the process and can react to deviations in a timely manner. For example, if there is a disturbance in the process, the machine can be stopped and the tool disabled. In conjunction with pallet machining, the next workpiece can automatically be started with a replacement tool. That way the machine tool remains productive despite the disturbance, and no manual intervention is necessary. Process Monitoring uses signals from within the control and does not require any additional sensors.

Process Monitoring increases safety and efficiency:

- Detect deviations from a reference machining run
- Enjoy reliable monitoring thanks to robust program synchronization down to the block level
- Ensure productivity through an extensive range of possible reactions, such as inserting a replacement tool
- Easy checking of the process result via 3D visualization of the workpiece and a 2D graph
- Easy programming
- Benefit from zero installation effort



Process Monitoring	Option 168	ID 1302488-01
TNC7	NC SW 81762x-16 or later	
TNC 640	—	
TNC 620	—	
TNC 320	—	
Installation by the machine manufacturer		
For more information , see the <i>TNC7</i> brochure, and visit www.klartext-portal.com		

Machining functions

3D-ToolComp: 3D radius compensation based on tool inclination

3D-ToolComp is a powerful option for three-dimensional tool radius compensation. In a compensation table, angle-dependent delta values are defined describing the tool's deviation from an ideal circular form (see figure).

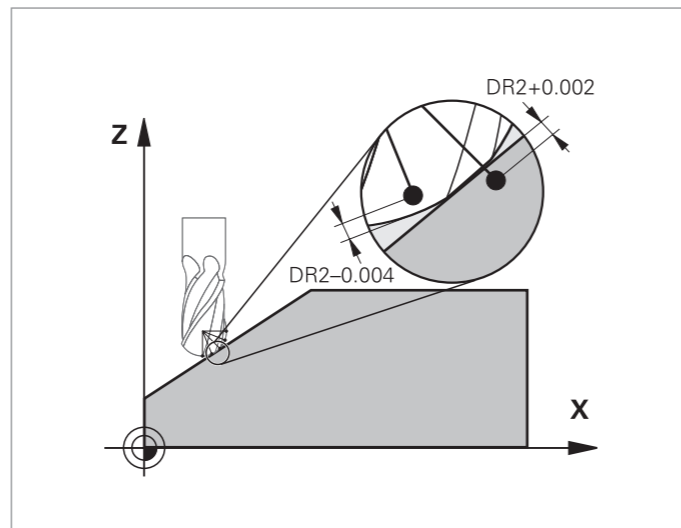
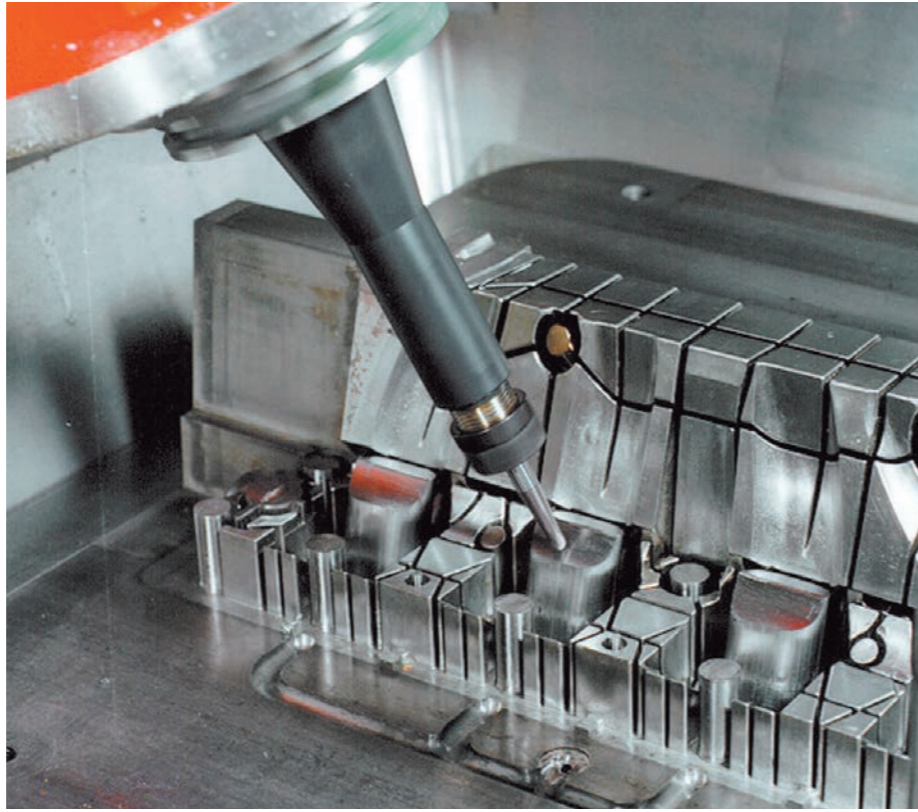
The TNC then corrects the radius value defined for the tool's current point of contact with the workpiece. In order for the exact point of contact to be determined, the NC program must be created by the CAM system with surface-normal blocks (LN blocks). The surface-normal blocks specify the theoretical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

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

Measuring 3D geometries

The control features a cycle that allows you to measure points on 3D geometries. To do so, you simply enter the point to be measured, along with its coordinates and associated normal vectors, into Cycle 444, 3D PROBING. After probing, the TNC automatically calculates whether the measured point is within a specified 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.

Software option 92 (3D-ToolComp) is required for 3D calibration.



3D-ToolComp	Option 92	ID 679678-01
TNC7	NC SW 81762x-16 or later	
TNC 640	NC SW 34059x-07 or later	
TNC 620	–	
TNC 320	–	
Installation by the operator		
For more information , see the TNC brochures, and visit www.klartext-portal.com		

Communication

HEIDENHAIN OPC UA NC Server: standardized communication

OPC UA NC Server (options 56 to 61)

Anyone wishing to digitally network their manufacturing environment needs effective technology with an assured future. Discover the HEIDENHAIN OPC UA NC Server, which provides HEIDENHAIN controls with an interface based on the OPC UA standard. This internationally standardized and widespread communication protocol makes it fast and easy to connect machines to your production IT setup. OPC UA is a cross-operating-system interface that can connect the HEIDENHAIN control not only to widespread Windows systems but also to Linux-based systems and Apple computers with macOS.

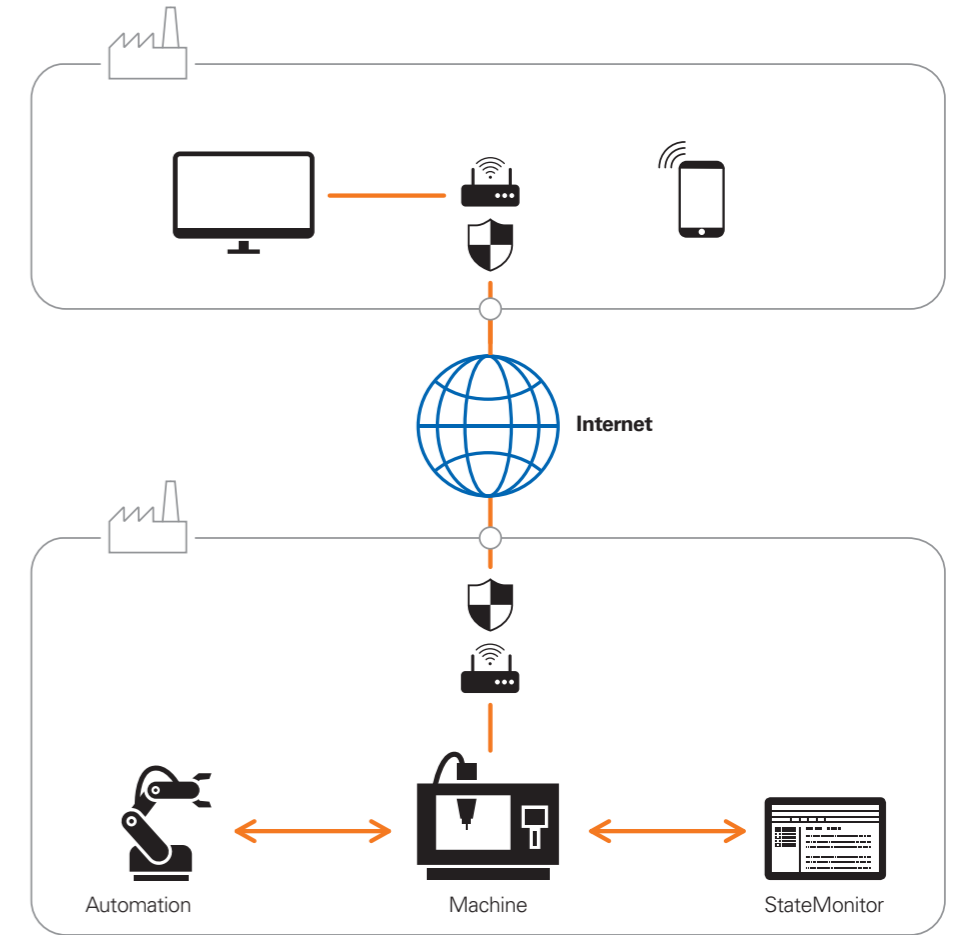
Numerous developer toolkits are available for OPC UA. RemoTools SDK is not needed. This option's standardized protocol, freely selectable toolkit, and application-oriented HEIDENHAIN information model allow standard software solutions and highly individualized applications to be developed with significantly less time to market.

The HEIDENHAIN OPC UA NC Server supports the following functions:

- Reading and writing variables
- Subscribing to value changes
- Executing methods
- Subscribing to events
- Transferring of files

Six SIK options are available for the HEIDENHAIN OPC UA NC Server. Each of the six SIK options enables an incoming OPC UA connection. Multiple connections can be configured and activated as needed on the control.

Explore the full HEIDENHAIN OPC UA NC Server yourself and test new OPC UA applications on a virtual machine such as the HEIDENHAIN programming station. Current versions of the programming station feature a free, full-version demo of the OPC UA NC Server.



Classification	Control	Software
<ul style="list-style-type: none"> • Machine identification • Production data acquisition • Machine messages • Program sequence check 	TNC7 TNC 640 TNC 620	81762x-16 or later 34059x-10 or later 81760x-08 or later
<ul style="list-style-type: none"> • File transfer • Remote management of file systems 	TNC7 TNC 640 TNC 620	81762x-16 or later 34059x-11 or later 81760x-08 or later
<ul style="list-style-type: none"> • OEM enhancements (e.g., PLC) 	TNC7 TNC 640 TNC 620	81762x-16 or later 34059x-16 or later 81760x-16 or later
<ul style="list-style-type: none"> • Tool data management 	TNC7 TNC 640 TNC 620	81762x-17 or later 34059x-17 or later 81760x-17 or later

OPC UA NC Server	Options 56 to 61	ID 1291434-01 to ID 1291434-06
TNC7	NC SW 81762x-16 or later	
TNC 640	NC SW 34059x-10 or later	
TNC 620	NC SW 81760x-08 or later	
Installation by IT specialists		
For more information , visit www.heidenhain.de/opcu-nc-server		

connected  machining

Communication

HEIDENHAIN DNC: communication over a COM component

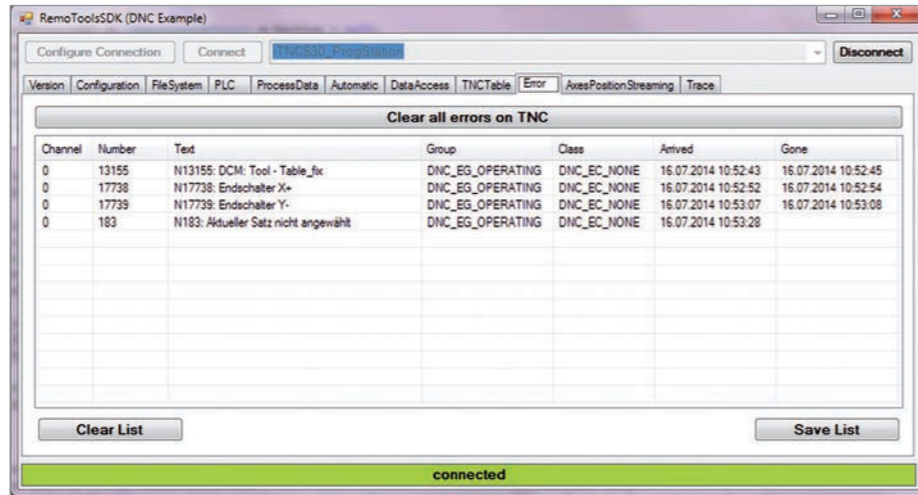
Paperless job management requires the seamless exchange of data about all process steps in the production process. The **HEIDENHAIN DNC** option allows a Windows application to access data on the control and change them as needed. Possible fields of application include the following:

- Software solutions for controlling the manufacturing process
 - Machine data acquisition systems and production data acquisition systems (MDA/PDA)
 - Connection to higher-level ERP/MES systems
 - Preventive maintenance planning based on the machine's actual condition
- Standardized or customized PC software
 - Greater process reliability and equipment availability
 - Error reporting systems (e.g., notifying the customer's smartphone when the current machining process encounters problems)
 - Current status overviews of all production machines
 - Database creation for extensive data mining

Since HEIDENHAIN DNC has been around for many years already, the interface is also particularly suitable for connecting older machine tools.

RemoTools SDK development package

In order for you to make use of the HEIDENHAIN DNC software interface, HEIDENHAIN offers the RemoTools SDK software development package. RemoTools SDK provides a Microsoft COM component for development environments on Windows operating systems, thus enabling communication with the HEIDENHAIN control. The COM component is registered in the Windows operating system during installation of RemoTools SDK.



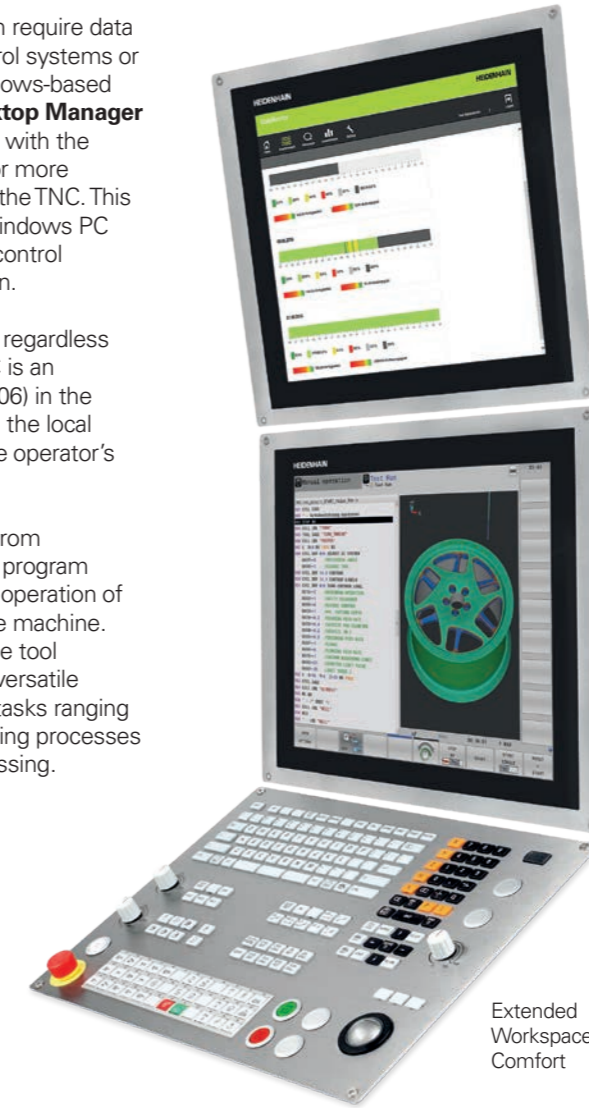
Communication

Remote Desktop Manager: view and remotely operate external computers

A typical work day may often require data entry into planning and control systems or diagnostic testing with Windows-based software. The **Remote Desktop Manager** option provides the operator with the opportunity to operate one or more Windows PCs directly from the TNC. This option fully integrates the Windows PC operating interface into the control interface on the TNC's screen.

This functionality is available regardless of whether the Windows PC is an industrial PC (e.g., the IPC 306) in the machine's control cabinet, in the local network as a server, or at the operator's office workstation.

Possible applications range from centralized job, tool, and NC program management to the remote operation of CAD/CAM systems from the machine. With this option, the machine tool operating panel becomes a versatile and efficient workspace for tasks ranging from specialized manufacturing processes to decentralized order processing.



Extended Workspace Comfort

Extended Workspace Comfort

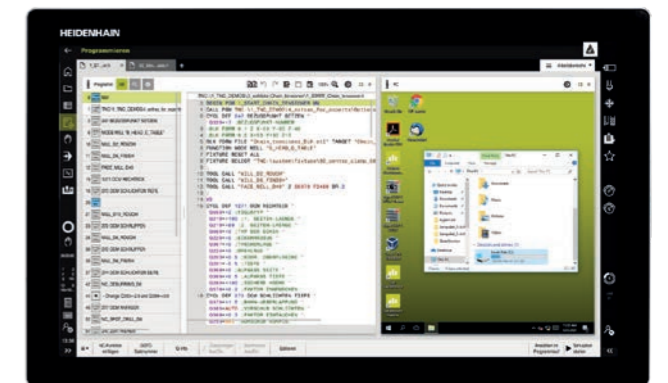
With Extended Workspace Comfort, you can add a second 19-inch HEROS screen next to or above the control screen of the TNC 640. While the current NC program is displayed on the control screen, Extended Workspace Comfort offers, for example, a view of an office PC that is connected via Remote Desktop Manager.

Extended Workspace Compact

Extended Workspace Compact lets any additional applications run in an extra window on a 24-inch touchscreen of the TNC 640; for example, the screen of a Windows PC can be shown. Its position alongside the familiar control screen gives you an optimum overview.

Embedded Workspace

The TNC7 offers the possibility of seamlessly embedding remote desktops or applications as a workspace or separate operating mode directly in the user interface of the TNC7. The workspaces are therefore able to display the content in an optimized way in the display areas selected by the operator.



Remote Desktop Manager

Option 133

ID 894423-01

TNC7
TNC 640
TNC 620
TNC 320

NC SW 81762x-16 or later
NC SW 34059x-02 or later
NC SW 81760x-02 or later
NC SW 77185x-04 or later

Installation by IT specialists

For more information, see the TNC brochures, see the *Connected Machining* brochure



HEIDENHAIN DNC	Option 18	ID 526451-01
RemoTools SDK	Accessory	ID 340442-xx
TNC7	NC SW 81762x-16 or later	
TNC 640	NC SW 34059x-01 or later	
TNC 620	NC SW 34056x-01/73498x-01/81760x-01 or later	
TNC 320	NC SW 34055x-01/771851-01 or later	
Installation by the operator		
For more information, see the TNC brochures, see the <i>Connected Machining</i> brochure		

Adaptation to the machine

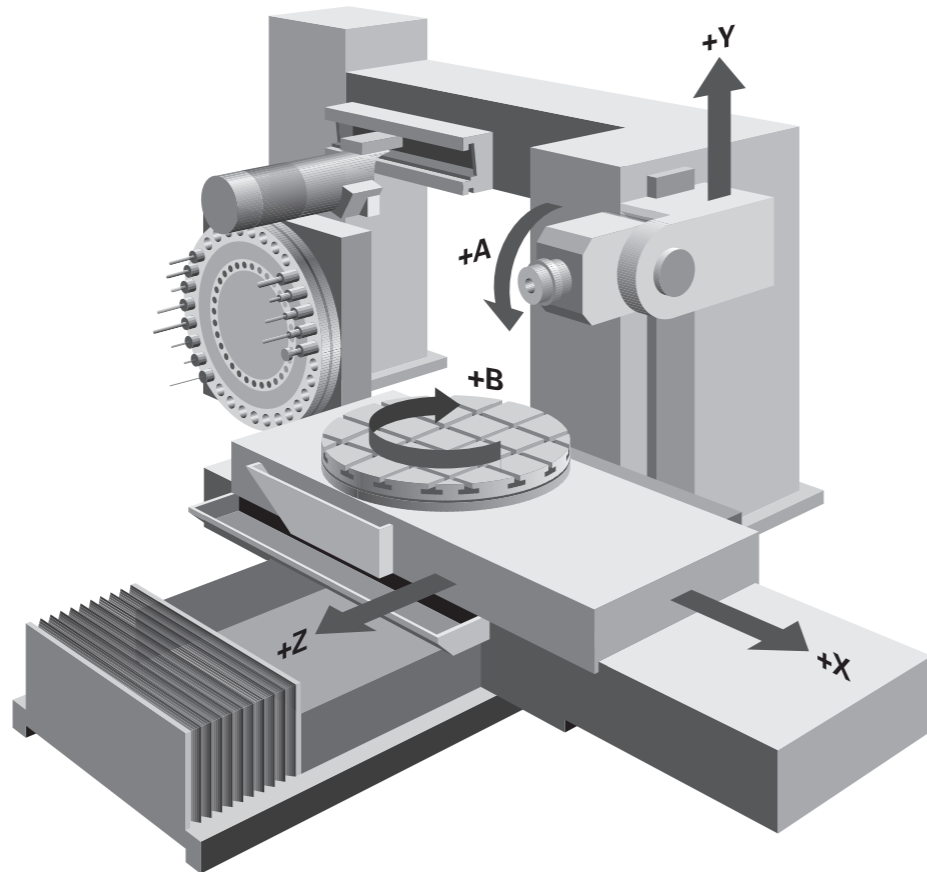
Additional control loops

The number of enabled control loops depends on the SIK being used and on any additionally enabled control loops (which can be ordered anytime as needed).

Additional control loops can be enabled either singly or in groups. Through combinations of control-loop groups and single control loops, any number of control loops can be enabled.

The maximum possible number of control loops depends on the control:

- TNC7: 24 control loops
- TNC 640: 24 control loops
- TNC 620: 8 control loops
- TNC 320: 6 control loops



Individual control loops		ID
Additional Axis 1	Option 0	354540-01
Additional Axis 2	Option 1	353904-01
Additional Axis 3	Option 2	353905-01
Additional Axis 4	Option 3	367867-01
Additional Axis 5	Option 4	367868-01
Additional Axis 6	Option 5	370291-01
Additional Axis 7	Option 6	370292-01
Additional Axis 8	Option 7	370293-01
Control-loop groups		
4 Additional Axes	Option 77	634613-01
8 Additional Axes	Option 78	634614-01
TNC7	NC SW 81762x-16 or later	
TNC 640	NC SW 34059x-01 or later	
TNC 620	NC SW 34056x-01/73498x-01/81760x-01 or later	
TNC 320	NC SW 34055x-01/771851-01 or later	
Installation by the machine manufacturer		
For more information , see the <i>Information for the Machine Tool Builder</i> brochures		

Adaptation to the machine

Integrated Functional Safety (FS)

Controls featuring integrated functional safety (FS) from HEIDENHAIN can attain Safety Integrity Level 2 (SIL 2) as per EN 61508, and Performance Level "d," Category 3, as per EN ISO 13849-1. In these standards, safety-related systems are assessed based on criteria such as the failure probabilities of integrated components and subsystems. This modular approach aids the manufacturers of safety-related machines in implementing their systems, since they can then build upon prequalified subsystems.

The controls with functional safety (FS) are based on two redundant, mutually independent safety channels. All safety-relevant signals are collected, processed, and output over two channels. Errors are detected through a reciprocal data comparison of the two channels' states. Consequently, the occurrence of a single error in the control does not cause a loss in safety functionality.

Gen3 exclusive

For Gen 3 drives, integrated functional safety (FS) is enabled through software options and is scalable. When purchasing these options, you can enable the exact number of safe control loops you actually need. For this purpose, new software options (160 to 166) are being introduced exclusively for the Gen 3 drives. Up to now, integrated functional safety (FS) has been enabled in conjunction with a PLB 62xx FS.

Option number	Option	ID	Comment
160	Integrated FS: Basic	1249928-01	Only for Gen 3: enabling of functional safety (FS) and enabling of four safe control loops
161	Integrated FS: Full	1249929-01	Only for Gen 3: enabling of functional safety (FS) and enabling of the maximum number of safe control loops (≥ 10)
162	Add. FS Ctrl. Loop 1	1249930-01	Only for Gen 3: additional safe control loop 1
163	Add. FS Ctrl. Loop 2	1249931-01	Only for Gen 3: additional safe control loop 2
164	Add. FS Ctrl. Loop 3	1249932-01	Only for Gen 3: additional safe control loop 3
165	Add. FS Ctrl. Loop 4	1249933-01	Only for Gen 3: additional safe control loop 4
166	Add. FS Ctrl. Loop 5	1249934-01	Only for Gen 3: additional safe control loop 5
169	Add. FS Full	1319091-01	Enabling all remaining FS control loops

The manufacturer needs option 169 if safe control loops are to be enabled via single options and ten or more safe control loops are needed.

Software option 169 enables all remaining FS axes if the following requirements are fulfilled:

- Software option 160 is enabled
- Software options 162 to 166 are enabled



Integrated FS	Options 160 to 166 ID 1249928-01 to 1249934-01	Option 169 ID 1319091-01
TNC7	81762x-16	81762x-16
TNC 640	34059x-10	34059x-11
TNC 620	81760x-07	81760x-08
Installation by the machine manufacturer		
For more information , see the Technical Information documents		

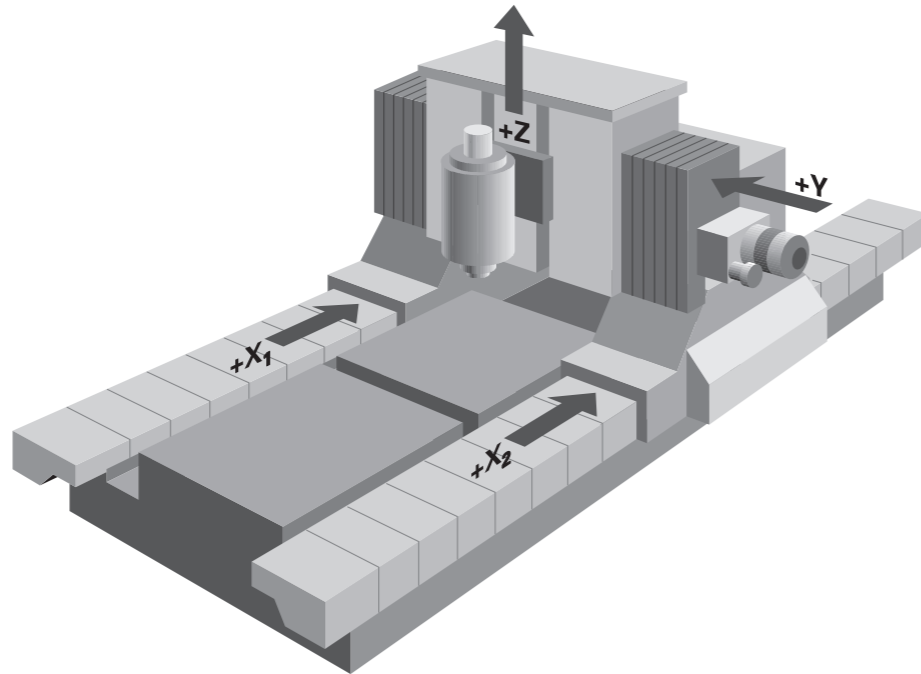
Adaptation to the machine

Synchronized axes: gantry axes, tandem tables

Synchronized axes are synchronously moving axes that are programmed using the same axis designation.

With HEIDENHAIN controls, parallel axis systems (gantry axes) such as those on portal-type machines or tilting tables can be moved synchronously thanks to high-accuracy, dynamic position control. The fast and particularly precise positioning movements are perfectly harmonized, thus enabling five-axis simultaneous motion for highly demanding tasks. Multiple slave axes can be assigned to a single master gantry axis.

Master-slave torque systems are typically used when large parts need to be moved or when rack and pinion drive systems require pre-stressing for backlash-free motion. Up to six drive systems can be combined into a single master-slave network and mutually pre-stressed as desired. As a result, fast and precise axis positioning becomes attainable even on large machine tools.



Gantry Axes	Option 24	ID 634621-01
TNC7	Standard function	
TNC 640	Standard function	
TNC 620	NC SW 34056x-01/73498x-01/81760x-01 or later	
TNC 320	NC SW 34055x-01/771851-01 or later	
Installation by the machine manufacturer		
For more information , see the <i>Information for the Machine Tool Builder</i> brochures		

Adaptation to the machine

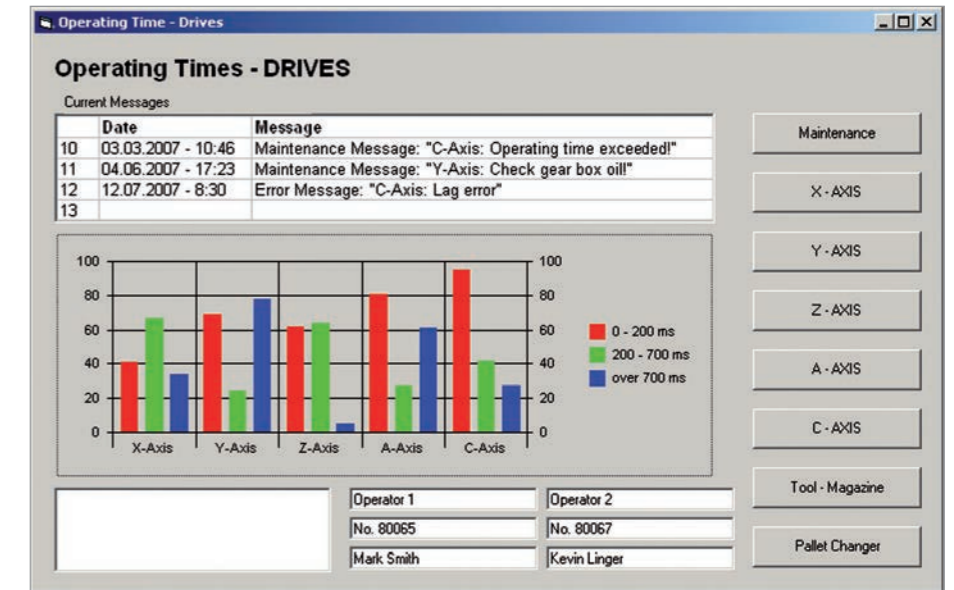
Python OEM Process: realize special functions

The **Python OEM Process** option gives machine manufacturers a powerful tool for using an object-oriented programming language within the control. Python is an easy-to-learn script language supporting all necessary high-level language elements.

Python OEM Process can be employed universally for machine functions, complex calculations, and the display of special user interfaces. User-specific or machine-specific solutions can be efficiently implemented. Regardless of whether you intend to create special algorithms for specialized functions or separate solutions (e.g., interfaces for machine maintenance software), a myriad of existing Python- and GTK-based functions are at your disposal.

You can integrate your newly created applications through the PLC in familiar PLC windows, or you can display them in your own windows incorporated into the TNC interface, even making them as large as the control's full screen.

Simple Python scripts (e.g., for display screens) can also be executed without enabling Python OEM Process (software option 46). 10 MB of dedicated memory are reserved for this function. For more information, refer to the "Python in HEIDENHAIN Controls" Technical Manual.



Python OEM Process	Option 46	ID 579650-01
TNC7	NC SW 81762x-16 or later	
TNC 640	NC SW 34059x-01 or later	
TNC 620	NC SW 34056x-01/73498x-01/81760x-01 or later	
TNC 320	NC SW 34055x-04/771851-01 or later	
Installation by the machine manufacturer		
For more information , see the <i>Information for the Machine Tool Builder</i> brochures		

Adaptation to the machine

Double Speed: short control-loop cycle times for direct drive motors

Single-speed control loops are usually sufficient for linear motors, torque motors, and conventional axes. **Double-speed control loops**, however, are the go-to choice for HSC spindles and difficult-to-control axes. In the control's default settings, all axes are set to single speed. Each axis that is switched from single speed to double speed can reduce the number of available control loops by one. PWM frequencies greater than 5 kHz require double-speed control loops. Option 49 must be enabled for this.

Double-speed control loops permit higher PWM frequencies and shorter speed-controller cycle times, thereby enabling improved current control for spindles and increased controller performance for linear and torque motors.

Control loop cycle times

Fine interpolation

Single speed: 0.2 ms
Double speed: 0.1 ms (with option 49)

Position controller

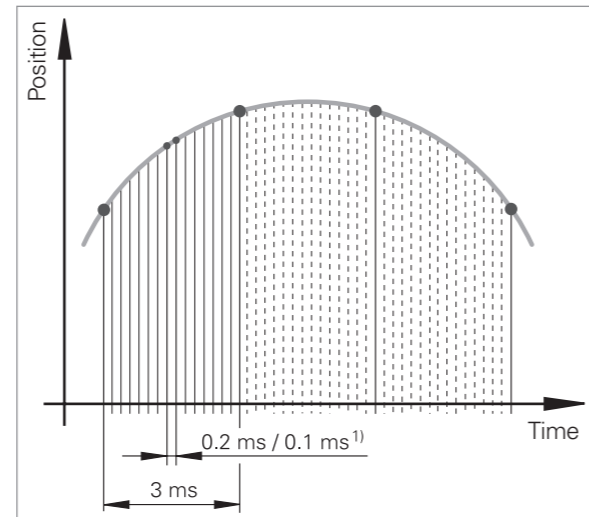
Single speed: 0.2 ms
Double speed: 0.1 ms (with option 49)

Speed controller

Single speed: 0.2 ms
Double speed: 0.1 ms (with option 49)

Current controller

f_{PWM}	T_{INT}
3333 Hz	150 μs
4000 Hz	120 μs
5000 Hz	100 μs
6666 Hz	75 μs with option 49
8000 Hz	60 μs with option 49
10 000 Hz	50 μs with option 49



¹⁾ Single speed/double speed (with option 49)

Adaptation to the machine

OEM Option

Machine manufacturers often equip their machine tools with added useful functions stored in the control's configuration parameters (e.g., PLC). These functions are then offered to the operator as options. To give the operator the greatest possible flexibility in enabling these options, HEIDENHAIN can reserve a range within the options menu (SIK menu) for use by the machine manufacturer as desired.

Options 101 to 130 provide thirty options that can be enabled by the machine manufacturer and enabled through his own PLC program. A particular benefit is the operator's ability to enable options via the SIK menu without the need for on-site support by the machine manufacturer.

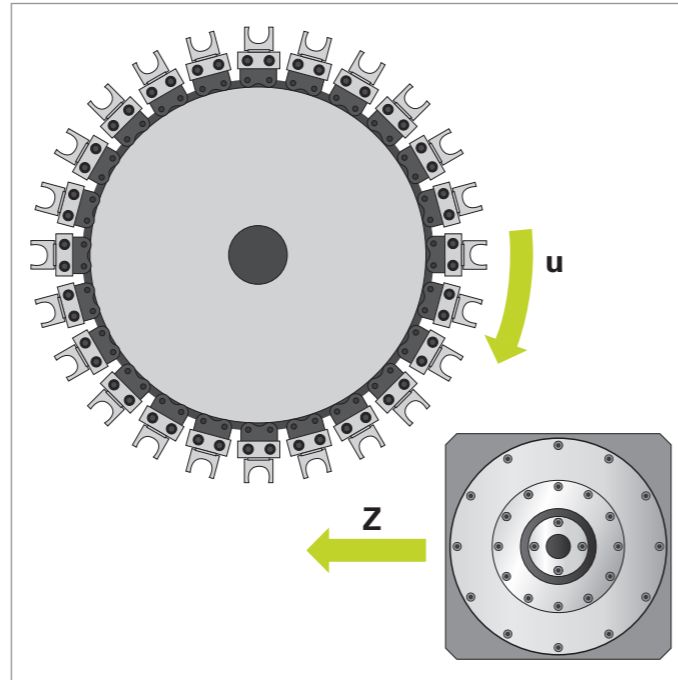
Double-Speed Axes	Option 49	ID 632223-01
TNC7	NC SW 81762x-16 or later	
TNC 640	NC SW 34059x-01 or later	
TNC 620	NC SW 34056x-01/73498x-01/81760x-01 or later	
TNC 320	–	
Installation by the machine manufacturer		
For more information , see the <i>Information for the Machine Tool Builder</i> brochures		

OEM Option	Options 101 to 130	ID 579651-01 to ID 579651-30
TNC7	–	
TNC 640	NC SW 34059x-02 or later	
TNC 620	–	
TNC 320	–	
Installation by the machine manufacturer		
Further information	–	

Adaptation to the machine

RTC: real-time coupling function for synchronizing axes and spindles

The **Real-Time Coupling** function (RTC) enables the cyclical calculation of a position offset for an axis based on the actual and nominal values of any other axes within the system. This function allows complex, simultaneous movements of multiple NC or PLC axes to be implemented. The interdependencies of the axes are defined in mathematical formulas. Possible applications, for example, are in PLC axes that need to be synchronized with an NC axis during a tool change in order to avoid collisions with the tool holders. The machine manufacturer defines these movements with RTC. The real-time coupling function makes complex traversing movements possible through the coupling of principal and secondary axes. Many new solutions are thus made possible, ranging from process-specific movements to tool changes with special requirements.



Software

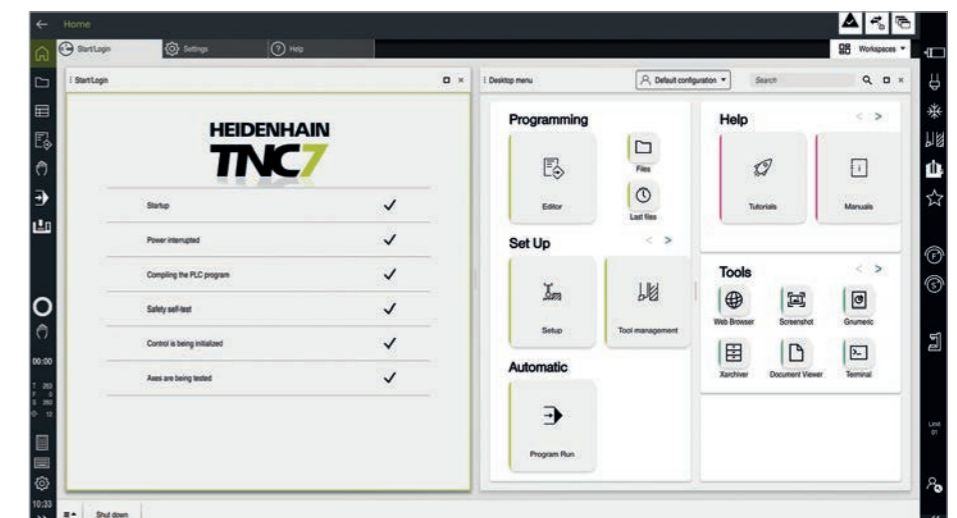
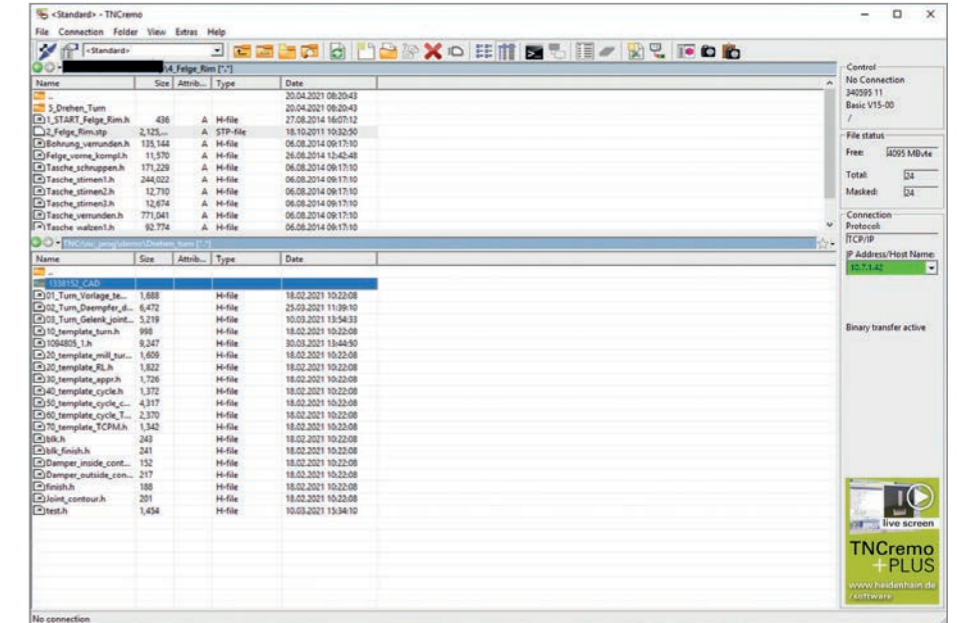
TNCremo: programs for data transfer

The free PC software package **TNCremo** supports the operator in transferring data from a PC to HEIDENHAIN controls or programming stations. TNCremo lets you bidirectionally transfer externally stored machining programs, tool tables, pallet tables, and backups of the hard drive, as well as query the operating status of the machine.

Functions:

- Data transfer and file management
- Creation of screenshots of the control
- Reading out of the control log
- Data backup for the control
- Creation of service files

In addition to the features already familiar to you from TNCremo, **TNCremoPlus** can transfer the current content of the control's screen to the PC (live screen). This allows you to implement convenient machine monitoring.



Synchronizing Functions	Option 135	ID 1085731-01
TNC7	NC SW 81762x-16 or later	
TNC 640	NC SW 34059x-04 or later	
TNC 620	–	
TNC 320	–	
Installation by the machine manufacturer		
For more information, see the <i>Information for the Machine Tool Builder</i> brochures		

TNCremo	Free download	
TNCremoPlus	Enabling module requires a fee	ID 340447-xx
TNC7	NC SW 81762x-16 or later	
TNC 640	NC SW 34059x-01 or later	
TNC 620	NC SW 34056x-01/73498x-01/81760x-01 or later	
TNC 320	NC SW 34055x-01/771851-01 or later	
Installation by the operator		
For more information, visit www.klartext-portal.com		

Software

RemoteAccess: remote diagnostics for HEIDENHAIN controls

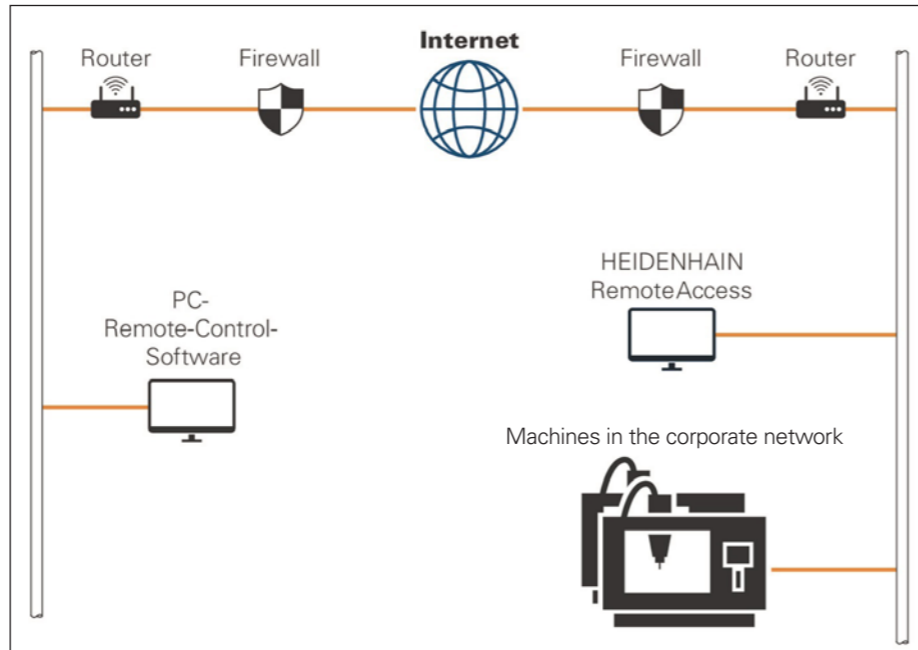
PC software for remote diagnostics, monitoring and operation

RemoteAccess grants quick and easy access to HEIDENHAIN controls that are installed within the same local network (intranet).

RemoteAccess offers the following functions:

- Display of the control's user interface on the PC
- Operating the control directly through the live view as well as with the integrated keyboard
- HEIDENHAIN PC tools are integrated automatically
- Can be enhanced with OEM-specific applications

Single station license ID 1339577-01
 Network license (14 stations) 1339577-02
 Network license (20 stations) 1339577-03



Remote connection over PC remote control software

Extension:

Secure Remote Access (SRA)

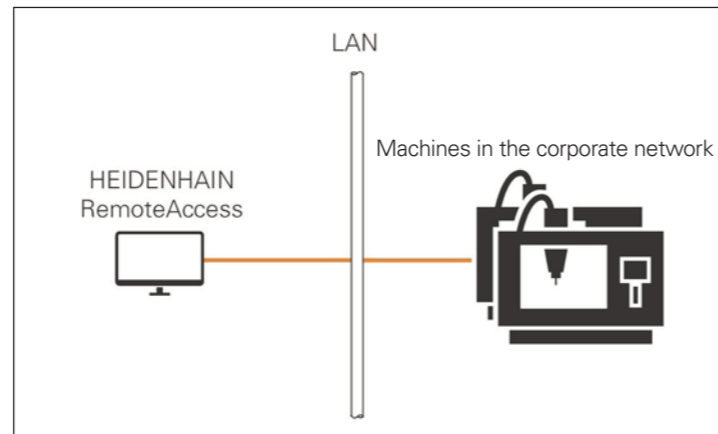
The Secure Remote Access enhancement permits setting up an encrypted remote connection to a HEIDENHAIN control via the Internet.

Once the SRA connection has been set up, RemoteAccess behaves like a local network connection.

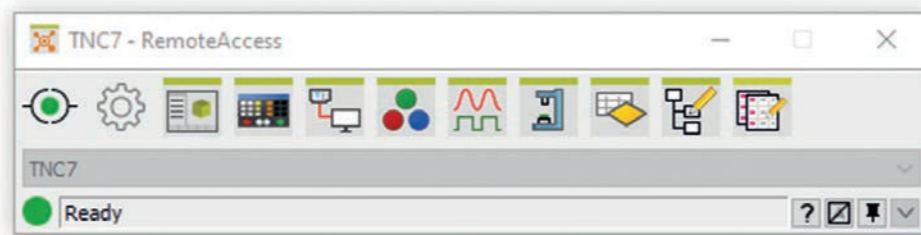
Possible applications when using SRA:

- User support
- Online seminars
- Diagnostics and service
- Other OEM services

Secure Remote Access enhancement ID 1356741-01



RemoteAccess in the corporate network



RemoteAccess	CD-ROM with dongle
Single station license	ID 340449-xx
Network lic. for max. 14 participants	ID 340454-xx
Network lic. for max. 20 participants	ID 340455-xx

TNC7	NC SW 81762x-16 or later
TNC 640	NC SW 34059x-01 or later
TNC 620	NC SW 34056x-01/73498x-01/81760x-01 or later
TNC 320	NC SW 34055x-01/771851-01 or later

Installation by the machine manufacturer

Further information -

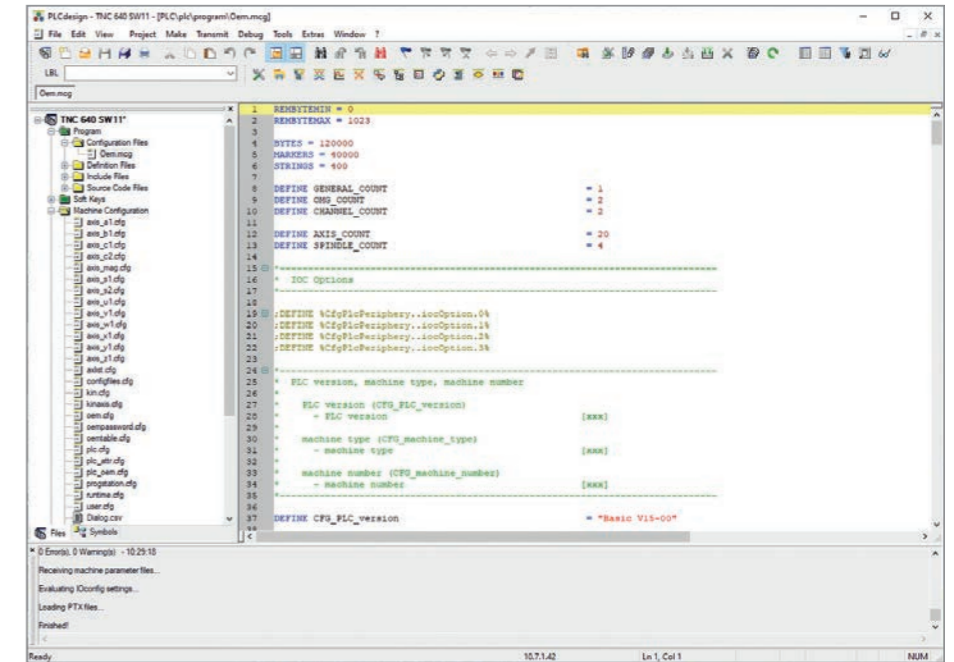
Software

PLCdesign: PLC program development software

The **PLCdesign** software is a convenient way to create PLC programs. It also comes with an extensive selection of sample PLC programs.

Functions

- Management of all project files with integrated project management in an intuitive tree structure
- Creation of PLC programs in IL
- Integrated editor with syntax highlighting for PLC programs
- Context-sensitive output windows
- Extensive help functions
- PLC beautifier
- Soft-key creation and generation
- Postprocessor for resolving symbolic operands in generic files
- Support for machine-specific configuration
- Uploading of all files to the control
- Debug functions:
 - Source text monitoring
 - Storage table
 - Monitoring list for symbolic operands
- Source-code browser for input wizard and navigation
- Supports source-code control systems



PLCdesign

TNC7	NC SW 81762x-16 or later
TNC 640	NC SW 34059x-01 or later
TNC 620	NC SW 34056x-01/73498x-01/81760x-01 or later
TNC 320	NC SW 34055x-01/771851-01 or later

Installation by machine manufacturers or servicing providers

For more information, see the *Information for the Machine Tool Builder* brochures

Software

StateMonitor: collect and evaluate machine data

The StateMonitor software records and visualizes the statuses of machines in a manufacturing environment. 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 utilization level. Based on these collected data, it is possible to uncover optimization potential.

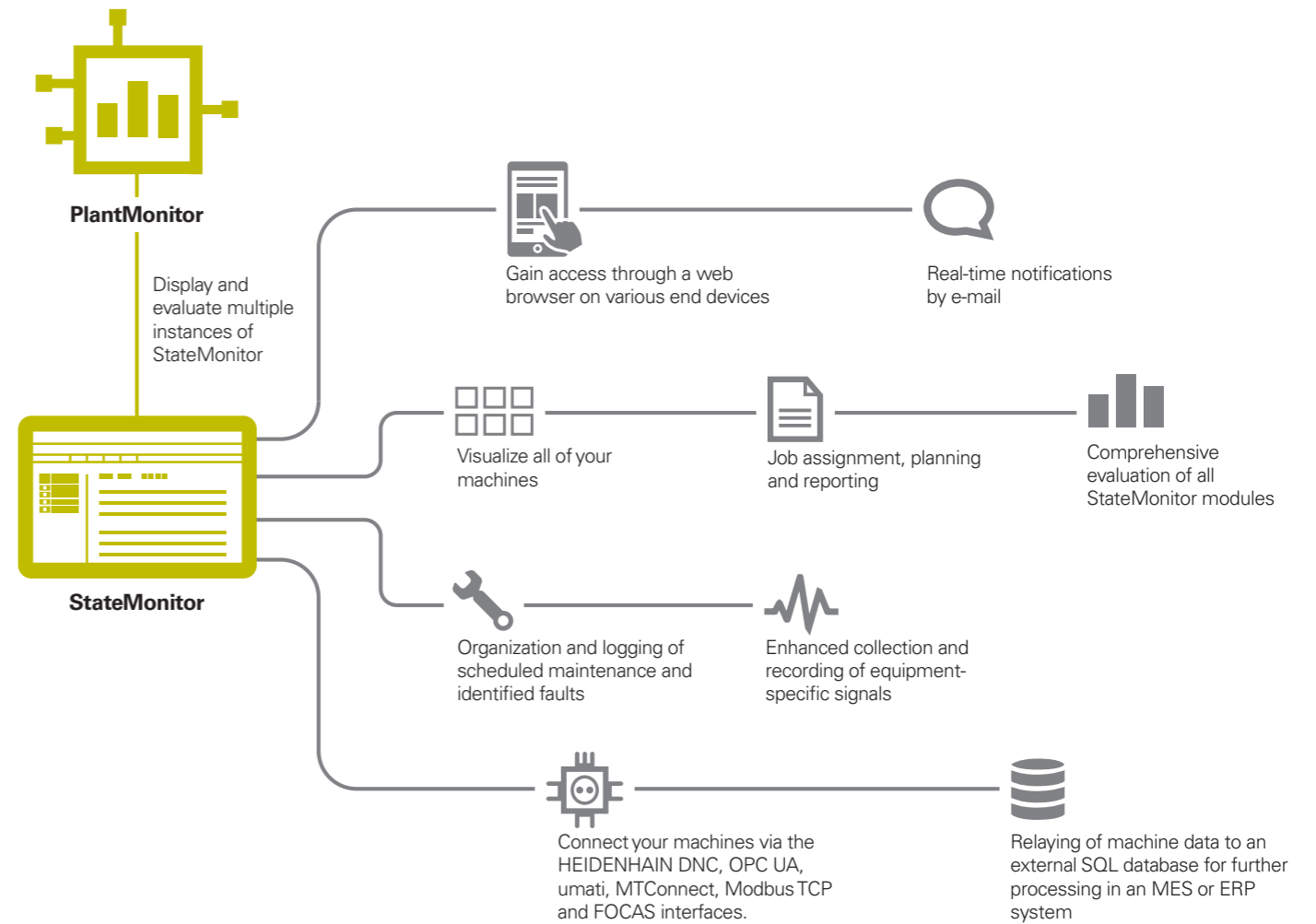


StateMonitor can record and visualize the following information:

- 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
- Maintenance entries and the maintenance history
- Job times
- Other machine-specific signal values

Machines with different controls can be connected to StateMonitor. The software supports the protocol types HEIDENHAIN DNC, OPC UA, umati, MTConnect, Modbus TCP and FOCAS.

For more information, please contact the team of the Digital Shop Floor from HEIDENHAIN.



StateMonitor	ID 1218930-xx	TNC 7	NC SW 81762x-16 or later
StateMonitor for an additional five machines	ID 1220884-xx	TNC 640	NC SW 34059x-01 or later
MaintenanceManager	ID 1308520-xx	TNC 620	NC SW 34056x-01/73498x-01/ 81760x-01 or later
JobTerminal	ID 1268674-xx	TNC 320	NC SW 34055x-01/771851-01 or later
5 Signals	ID 1308521-xx	Prerequisite: option 18 (DNC)	
OPC UA Interface	ID 1268673-xx		
MTConnect Interface	ID 1268675-xx		
Modbus Interface	ID 1268670-xx		

Installation by the machine manufacturer or end user

For more information, visit www.digital-shop-floor.heidenhain.com and www.klartext-portal.com, and see the *Digital Shop Floor* flyer

Software

KinematicsDesign: create machine kinematic models

KinematicsDesign is a PC software application for the graphically supported creation of kinematic descriptions for HEIDENHAIN controls. It also offers a convenient way to configure and start using the DCM collision monitoring function.

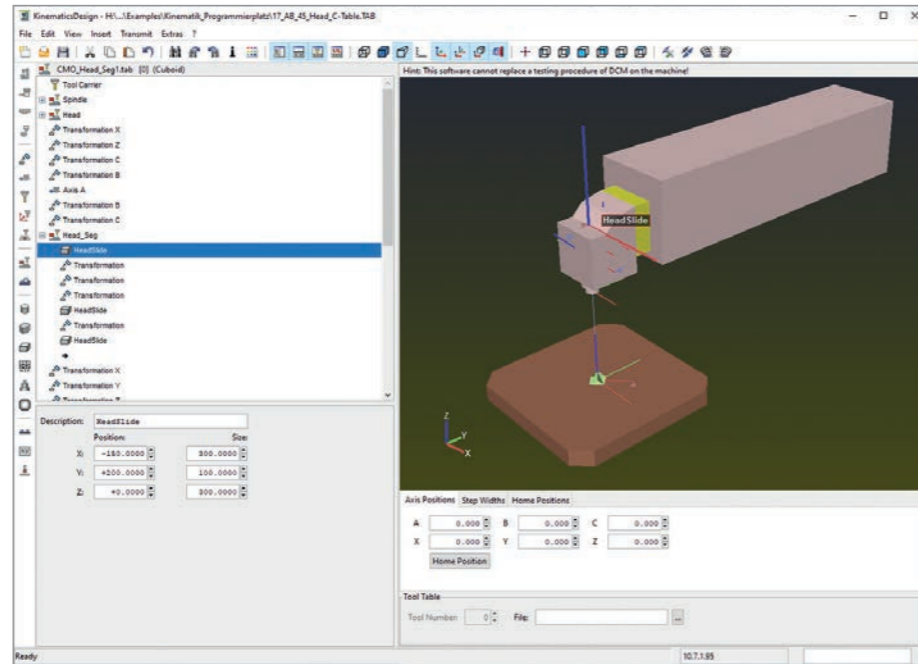
KinematicsDesign can create all objects of the kinematic chain, transfer them to and from the control, and edit them. Extensive visualization possibilities allow the operator to display the information that he currently needs in a clear format. Transformations, collision objects, and designations, for example, can be shown or hidden independently of each other. Collision objects can be shown as wire models, transparently, in full, etc. Colored highlighting makes it easy in KinematicsDesign to discern which collision objects have been checked in relation to each other or are excluded from the collision inspection. These functions are also available in the KinematicsDesign tool directly in the control.

KinematicsDesign visualizes the kinematic model with the current axis positions of a connected control (or programming station) or with manually entered axis positions and displays collisions. In addition, the current tool or a specified tool from a tool table can be shown along with a kinematic model of its tool carrier. KinematicsDesign can import kinematic models from the iTNC 530 and convert them into the format of the NCK controls, as well as generate clamping equipment and tool carrier kinematic models.

The TNC7 and TNC 640 let you import collision objects from a CAD file and incorporate them as M3D data into the machine kinematics. For the creation of the M3D data, the installation package of KinematicsDesign contains the free-standing PC program M3D Converter.

M3D Converter

This tool can upload, check, repair, simplify, and optimize Step and STL files, thereby preparing them for use as collision objects. In the process, the operator can always keep the difference from the original object in view. M3D Converter requires a software release module (ID 1124969-xx).



M3D Converter	Enabling module requires a fee	ID 1124969-xx
TNC7 TNC 640 TNC 620 TNC 320	NC SW 81762x-16 or later NC SW 34059x-05 or later NC SW 81760x-02 or later NC SW 77185x-02 or later	
Installation by machine manufacturers or servicing providers		
For more information , see the <i>Information for the Machine Tool Builder</i> brochures		
KinematicsDesign	Registration required	
TNC7 TNC 640 TNC 620 TNC 320	NC SW 81762x-16 or later NC SW 34059x-01 or later NC SW 34056x-01/73498x-01/81760x-01 or later NC SW 34055x-01/771851-01 or later	
Installation by machine manufacturers or servicing providers		
For more information , see the <i>Information for the Machine Tool Builder</i> brochures		

Software

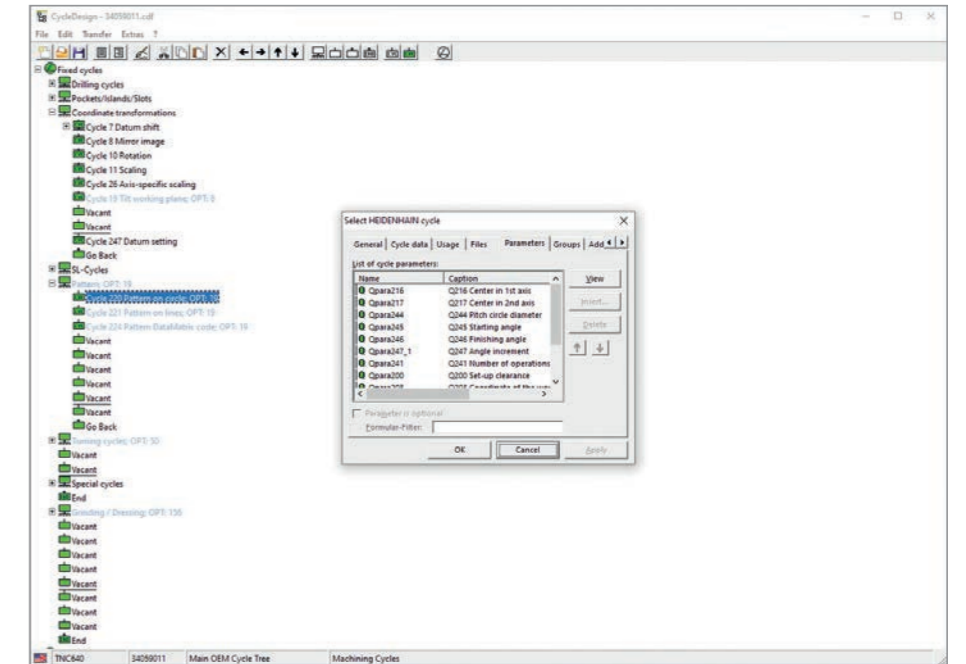
CycleDesign: save NC subprograms as cycles

For frequently recurring operations, HEIDENHAIN controls provide you with NC subprograms containing configurable parameters. These subprograms are referred to as cycles. The TNC guides you through the process of entering these parameters with prompts, questions, and help graphics.

You access the cycles on the control by pressing the CYCL DEF key.

Using **CycleDesign**, you can even incorporate your own NC subprograms as cycles into the soft-key structure of the control (or the menu structure of the TNC7). You can choose whether to add your cycles to the HEIDENHAIN cycle bar on the side or completely replace the HEIDENHAIN cycle bar.

CycleDesign lets you save this cycle data in the control's memory.



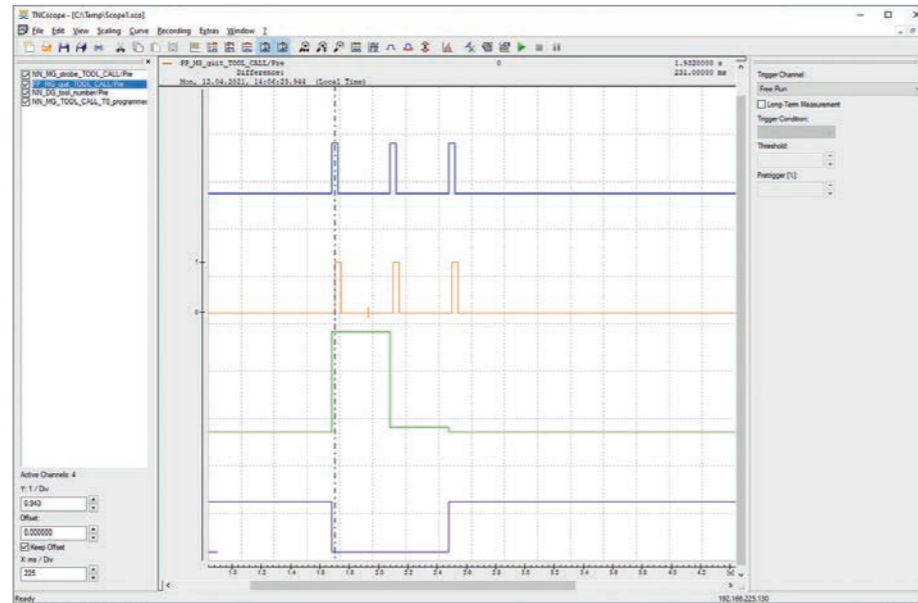
CycleDesign	Registration required
TNC7 TNC 640 TNC 620 TNC 320	NC SW 81762x-16 or later NC SW 34059x-01 or later NC SW 34056x-01/73498x-01/81760x-01 or later NC SW 34055x-01/771851-01 or later
Installation by operators or machine manufacturers	
For more information , see the <i>Information for the Machine Tool Builder</i> brochures	

Software

TNCscope: export oscilloscope files to a PC

TNCscope is a PC oscilloscope software program for the machine manufacturer for recording and evaluating control signals, drive signals, and PLC operands from HEIDENHAIN controls. TNCscope enables convenient diagnosis from a Windows PC. With a simple Ethernet connection, you can also use TNCscope to perform remote maintenance on the control. The TNCscope software also lets you display oscilloscope files that have been recorded on the control (offline mode). An extensive selection of mathematical functions is available for analysis.

- Multi-channel recording
- Various triggering options
- Convenient analysis of measured values
- Display of circular interpolation test and XY files of the control
- Display of logic-trace files
- Cursor and auxiliary cursors
- Setting of marks
- Measurement and comparison of curves
- Gating of curves with a formula
- Calculation of integrals and derivatives
- XY view
- Frequency analysis with Fast Fourier Transformation (FFT)
- Copying of curves to another file
- Printing of a graph with additional information
- Import function for any ASCII tables
- Long-term measurements
- Recording in batch mode
- Support during the initial setup of various control functions (PAC, LAC, CTC, MAC, ACC, Component Monitoring)



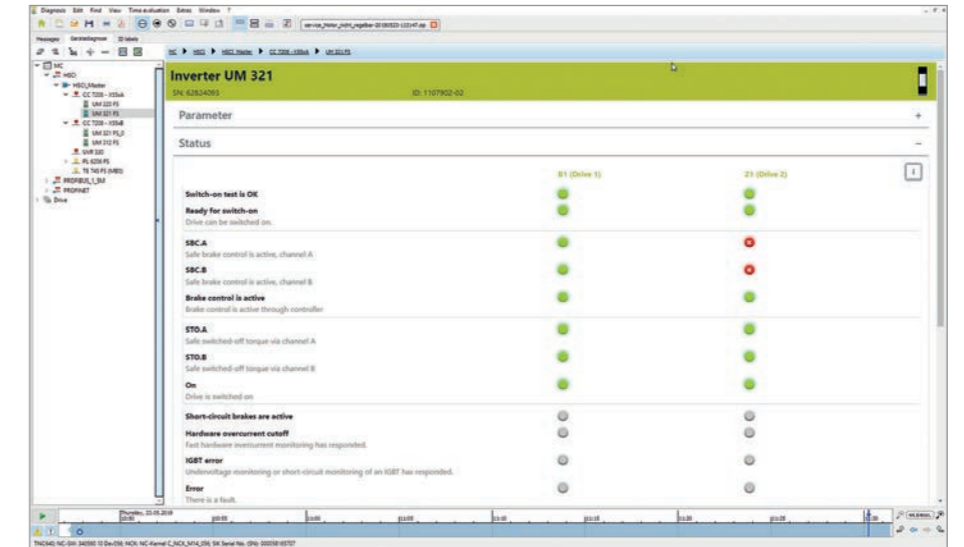
TNCscope	Registration required
TNC7	NC SW 81762x-16 or later
TNC 640	NC SW 34059x-01 or later
TNC 620	NC SW 34056x-01/73498x-01/81760x-01 or later
TNC 320	NC SW 34055x-01/771851-01 or later
Installation by machine manufacturers or servicing providers	
For more information , see the <i>Information for the Machine Tool Builder</i> brochures	

Software

TNCdiag: for evaluating status information and diagnostic information

Based on the **HFL** interface (**HEIDENHAIN Fibre Link**) between the components of the Gen 3 drives, detailed diagnostic possibilities are offered during initial setup and later servicing. To this end, the visualization and presentation of the available diagnostic data have been improved in the **TNCdiag** software. TNCdiag provides all of the relevant information and statuses of the components in the control system, all the way down to the encoders. This depiction can be component- or axis-specific and is optimized for touch operation. TNCdiag is available as an application on the control for displaying live data and as a PC tool for evaluating diagnostic data, including offline from service files.

- Status and diagnostic information about the HEIDENHAIN components (drive electronics, encoders, input/output devices, etc.) connected to the control
- History of the recorded data
- Replacement of DriveDiag for Gen 3



Graphically supported, dynamic display of status signals

TNCdiag	Registration required
TNC 640	NC SW 34059x-10 or later
TNC 620	NC SW 81760x-07 or later
TNC 320	–
Installation by machine manufacturers or servicing providers	
For more information , see the <i>Information for the Machine Tool Builder</i> brochures	

Software

TNCopt: for the initial setup of digital control loops

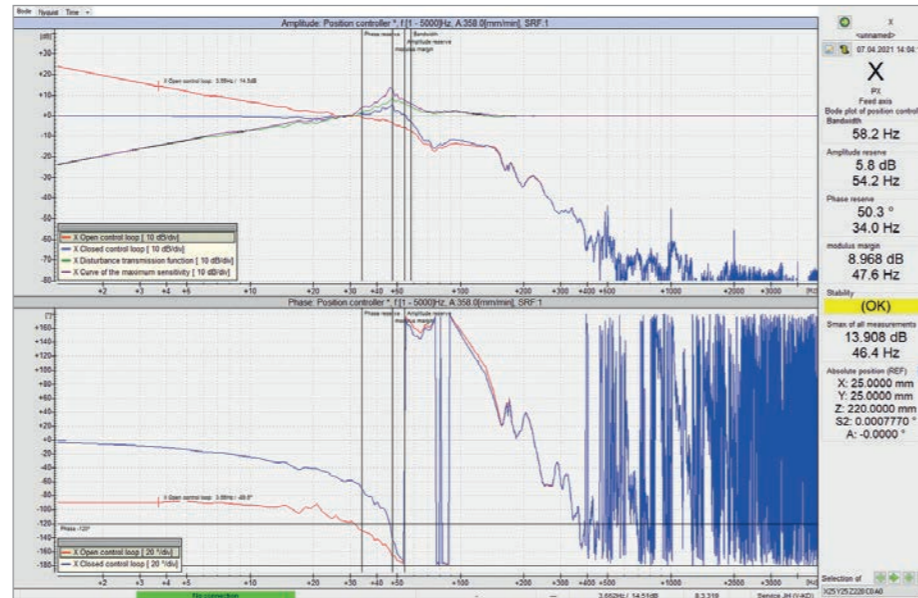
Machine tools with HEIDENHAIN controls must always meet the highest performance standards (surfaces, accuracy, fast and efficient machining, stable machining processes, etc.).

As part of this, exceptional performance is required of the drive system with servo motors and spindles. That's why HEIDENHAIN makes efficient and intelligent closed-loop technology its highest priority. Of particular importance is having the correct control-loop optimization and function settings in the control.

TNCopt conveniently helps you keep an overview and follow the proper sequence while setting up the axes.

Functions:

- Initial setup/optimization of the cascade control
- Compensation of peripheral machine influences (friction, hysteresis, etc.)
- System identification
- Dynamic Efficiency/Dynamic Precision
- Measuring functions (circular interpolation test, etc.)
- Automated Controller Tuning (ACT)



Software

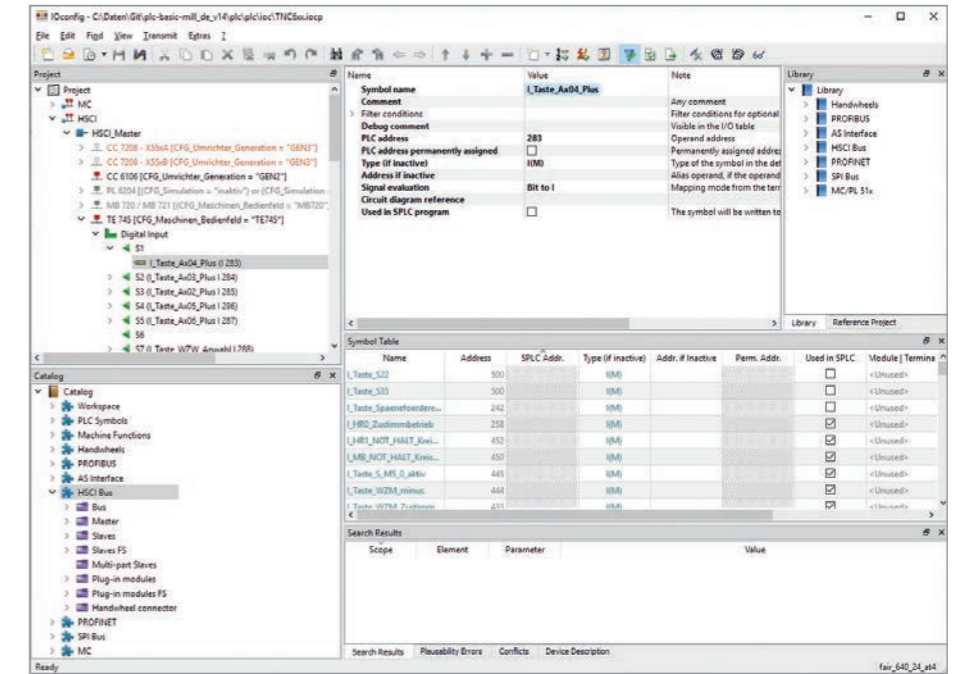
IOconfig: configuration software for inputs and outputs

Modern machine tools are becoming ever more complex and extensively equipped. For this reason, IOconfig allows you to configure all bus systems for HEIDENHAIN controls with all symbolic PLC operands. All required information is stored in a project file (.iocp).

Functions:

- All information is included in one project file
- All device descriptions are contained in the project file
- Generation of symbol definition files for PLC and SPLC
- Comprehensive option handling for including optional elements in the project
- Simulation mode for direct overview of active and inactive elements
- Convenient operation thanks to modern user interface
- Easy exchange of modules
- Symbol table for straightforward management of PLC symbols
- Definition of PLC start addresses for all elements
- Catalog for device descriptions
- Library for frequently used elements and branches
- Reference project for transferring elements from other projects
- Embedded in PLCdesign as of PLCdesign 3.1

With IOconfig 3, new bus systems are supported by simply adding further bus plug-ins. IOconfig currently includes plug-ins for SPI, PROFIBUS, PROFINET, and AS interface, as well as MC/PL51x and handwheels. The current device description files and the HEIDENHAIN PROFIBUS PL are also included in delivery.



TNCopt	Registration required
TNC7	NC SW 81762x-16 or later
TNC 640	NC SW 34059x-04 or later
TNC 620	NC SW 81760x-01 or later
TNC 320	—
Installation by machine manufacturers or servicing providers	
For more information , see the <i>Information for the Machine Tool Builder</i> brochures	

IOconfig	Registration required
TNC7	NC SW 81762x-16 or later
TNC 640	NC SW 34059x-01 or later
TNC 620	NC-SW 34056x-01/73495x-01/81760x-01 or later
TNC 320	NC-SW 77185x-01 or later
Installation by machine manufacturers or servicing providers	
For more information , see the <i>Information for the Machine Tool Builder</i> brochures	

Software

TNCkeygen: enabling keys for software options

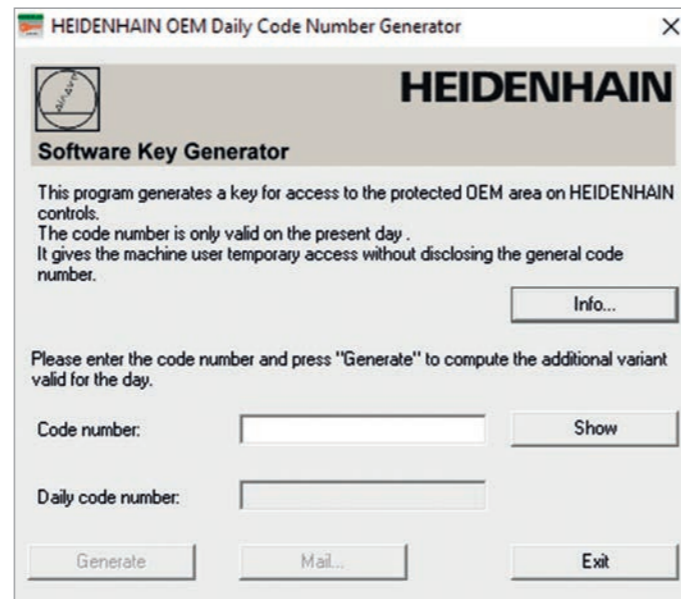
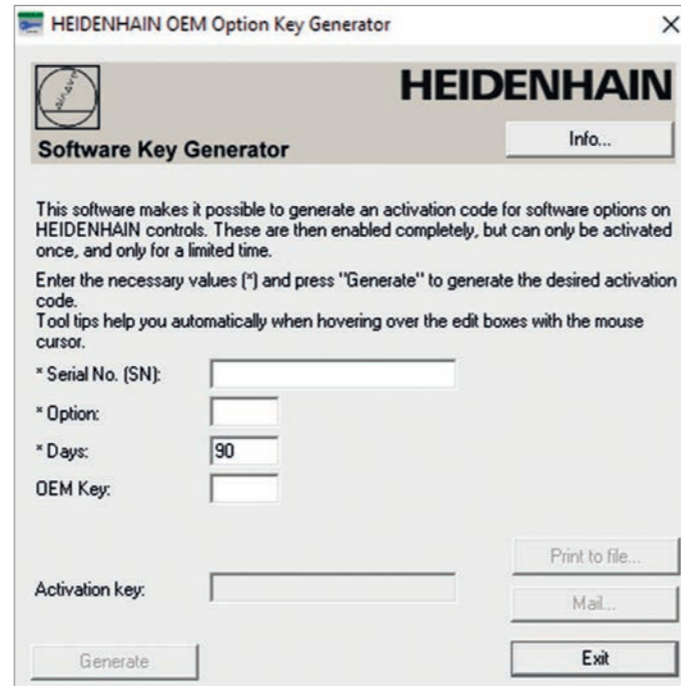
TNCkeygen is a collection of PC software tools for generating enabling keys for HEIDENHAIN controls for a limited period of time.

OEM key generator

The OEM key generator PC tool makes it possible to generate an enabling key for software options on HEIDENHAIN controls. The selected option is enabled for a limited time (10 to 90 days). It can only be enabled once. You can generate the desired enabling key by entering the SIK number, the option to be enabled, the duration, and a manufacturer-specific password. This allows you to test the available options at your own pace without being forced to buy them.

OEM daily key generator

This application generates an enabling key for the protected machine-manufacturer area on HEIDENHAIN controls. This key gives you access on the same day the key is generated.



Software Key Generator	Registration required
TNC7	NC SW 81762x-16 or later
TNC 640	NC SW 34059x-01 or later
TNC 620	NC SW 34056x-03/73498x-01/81760x-01 or later
TNC 320	NC SW 34055x-05/771851-01 or later
Installation by machine manufacturers	
For more information , see the <i>Information for the Machine Tool Builder</i> brochures	

Software

BMXdesign

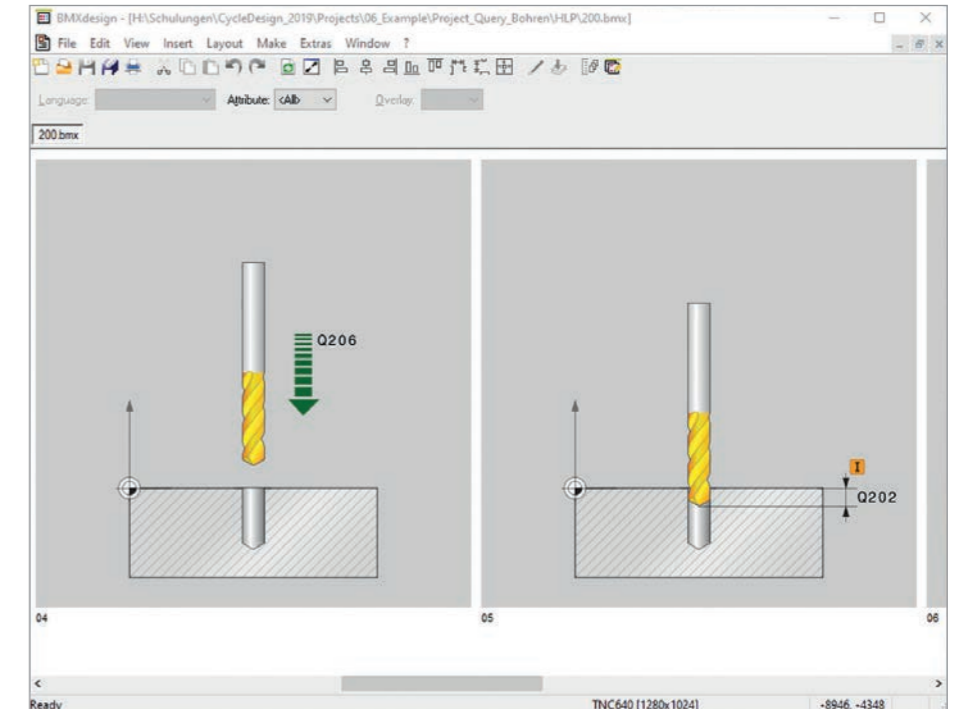
BMXdesign allows you to interactively create BMX project files and finished BMX files.

BMX files contain graphical and text elements that can be combined by the control into variants during program run. In this manner, help graphics or soft keys with language-sensitive texts or status-sensitive variants can be combined into just a single file.

BMX files are described in a BMX project file (*.BPJ). BMXdesign generates the final BMX file from the BPJ file.

Available functions:

- Interactive creation of BMX project files (WYSIWYG)
- Display of BMX files as on the control
- Adding and editing text and graphic elements
- Positioning of text fields with the mouse
- Automatic alignment of fields
- Adding of variants
- Replacement of database IDs with plain-language texts from multi-lingual files
- Printout with page preview
- Generation of BMX files
- Export function as bitmap file (*.BMP)
- Extracting graphic elements from BMX files
- Integration of PLCtext for managing BMX files
- Integration in PLCdesign



BMXdesign	Registration required
TNC7	NC SW 81762x-16 or later
TNC 640	NC SW 34059x-01 or later
TNC 620	NC SW 34056x-01/73498x-01/81760x-01 or later
TNC 320	NC SW 34055x-01/771851-01 or later
Installation by machine manufacturers	
For more information , see the <i>Information for the Machine Tool Builder</i> brochures	

Software

Programming station

What's the purpose of a programming station?

As easy as it is to write a part program on a TNC 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 programming station lets you program the same way you would at the machine but away from all the noise and distractions.

Creating programs

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

Testing programs created offline

Naturally, you can also test programs written on a CAM system. The high-resolution program verification graphics help you easily spot contour damage and hidden details, even in complex 3D programs.

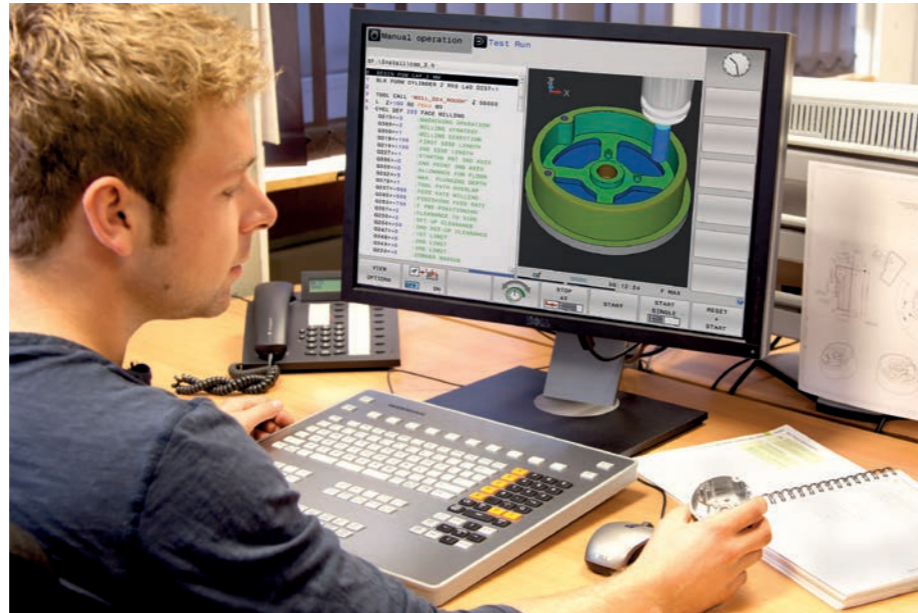
Training with the TNC programming station

Since the programming station is based on the same software as the TNC, it is ideal for apprentice and advanced-level training. Programming is performed on the original keyboard unit. Even the Test Run mode runs exactly as it would on the machine. This gives the trainee the confidence needed to operate a real machine. The programming station is also optimally suited for teaching TNC programming skills in schools, enabling programming in HEIDENHAIN Klartext and even ISO code.

The workstation

The programming station software runs on a PC and differs only marginally from a real TNC on a machine. You can still work with a TNC operating panel, except that it now includes the soft keys normally built into the monitor. The TNC operating panel is connected to the USB port of your PC, allowing your PC to display the familiar TNC interface. Alternatively, you can also operate the programming station without a keyboard. In this case, operation is performed with a virtual keyboard displayed along with the TNC control panel. It gives you the TNC's most important dialog activation keys.

Which programming stations are available?



Programming stations are available for all current TNC controls. A further version for operation with the VBox virtualization software is available as an option. The programming station DVDs include the programming station software, the necessary drivers, and the current User's Manuals in all available languages.

Programming station	TNC 640	TNC 620 / TNC 320
Demo version	ID 1114029-xx	ID 1114030-xx
With TNC operating panel	ID 1113967-02	
With virtual keyboard		
Single station license	ID 1113924-03	
Network license, 1 station	ID 1125955-03	
Network license, 14 stations	ID 1113926-03	
Network license, 20 stations	ID 1113928-03	
Operating panel without software release module	ID 1113967-52	

Installation by the operator

For more information, see the *Programming Station for TNC Controls* brochure

Software

virtualTNC: control of virtual machines

The simulation of NC programs on the control has long been a part of HEIDENHAIN contouring controls. The **virtualTNC** solution makes it possible to use the TNC as a control component for machine simulation applications (virtual machines) on external computer systems.

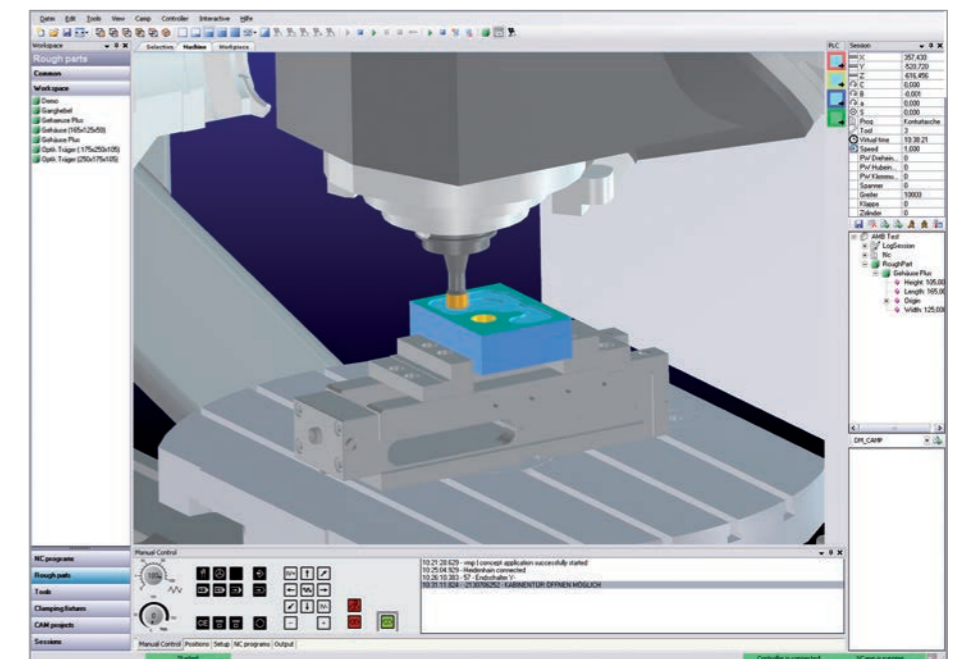
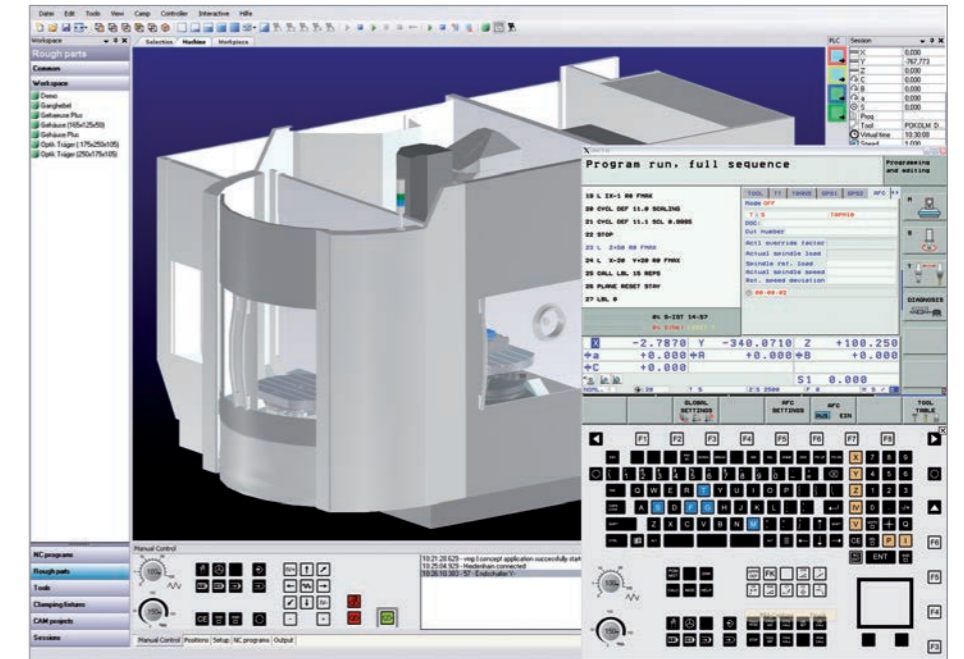
How a virtual machine with virtualTNC works

Machine simulation applications (virtual machines) can fully simulate production units, allowing real-world production processes to be optimized in advance. The virtualTNC software can control the axes of a virtual machine as if it were a real system. Operators program and operate the control just as they would an actual HEIDENHAIN TNC.

The virtualTNC solution is a programming station software application of the TNC that, thanks to a special interface, is able to ascertain the current axis positions of the running "virtual" control.

Connecting virtualTNC over HEIDENHAIN DNC

Software manufacturers who wish to simulate a production system can connect their virtual machine to virtualTNC through HEIDENHAIN DNC. The **COM component** (AxisStreaming object) required for the programming and configuration of the interface to virtualTNC and its interface description are included in the **RemoTools SDK** development package and the relevant help system.



virtualTNC is available for a fee	TNC 640
Single station license	ID 1113933-03
Network license, 1 station	ID 1122145-03
Network license, 14 stations	ID 1113935-03
Network license, 20 stations	ID 1113936-03

Installation by manufacturers of machine simulation applications

For more information, see the *HEIDENHAIN DNC* brochure

Hardware enhancements

HR: electronic handwheels

You can move TNC-controlled machine axes by simply pressing the axis direction keys. A simpler and more sensitive method, however, is to use an electronic handwheel from HEIDENHAIN.

In this case, the feed motor moves the axis slide in direct relation to the handwheel's rotation. For delicate operations, you can incrementally set the distance traveled per handwheel revolution.

HR 130 panel-mounted handwheels

The panel-mounted handwheels from HEIDENHAIN can be integrated into the machine operating panel or mounted at another location on the machine.

HR 510, HR 520, and HR 550 portable handwheels

The HR 510, HR 520, and HR 550 portable handwheels are particularly helpful when you work near the machine's working space. The axis keys and certain function keys are built into the housing. 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. The integrated display of the HR 520 and HR 550 handwheels keeps you immediately informed about the most important operating statuses. As a wireless handwheel, the HR 550 is ideal for use on large machines. When you aren't using the handwheel, you can simply place it in the HRA 551 FS handwheel holder (transceiver with integrated charging device).



HR 550



HR 520



HR 510



HR 130

Electronic handwheel	With/without detent	TNC7	TNC 640	TNC 620	TNC 320
		Minimum NC SW	Minimum NC SW	Minimum NC SW	Minimum NC SW
HR 130	ID 540940-01/540940-03	–	34059x-01	34056x-01/73498x-01/81760x-01	34055x-01
HR 510	ID 1120313-xx/1119971-xx	81762x-16	34059x-01	34056x-01/73498x-01/81760x-01	34055x-01
HR 510FS	ID 1119974-xx/1120311-xx	81762x-16	34059x-02	34056x-02/73498x-02/81760x-01	34055x-06
HR 520	ID 670303-xx/670302-xx	81762x-16	34059x-02	34056x-04/73498x-02/81760x-01	34055x-06
HR 520FS	ID 670305-xx/670304-xx	81762x-16	34059x-02	34056x-04/73498x-02/81760x-01	34055x-06
HR 550FS	ID 1183021-xx/1200495-xx	81762x-16	34059x-02	34056x-04/73498x-02/81760x-01	34055x-06
HRA 551 FS for HR 550 FS	ID 1119052-xx	81762x-16	34059x-02	34056x-04/73498x-02/81760x-01	34055x-06

Installation by the machine manufacturer

For more information, see the *Information for the Machine Tool Builder* brochures

Hardware enhancements

TS: workpiece touch probes

Workpiece touch probes from HEIDENHAIN help you reduce cost during shopfloor and serial production. Together with the TNC, touch probes can be used for manual or automatic setup, measuring, and inspection functions:

- Workpiece alignment
- Preset setting
- Workpiece measurement

The touch probes for workpiece measurement are inserted into the tool holder either manually or by a tool changer. Based on the machine, the touch probes can be equipped with various tool shanks. The stylus of a TS touch trigger probe is deflected upon contact with a workpiece surface. In that instant, the TS generates a triggering signal that, depending on the model, is transmitted to the control either by cable, radio transmission, or an infrared beam.

Cable-bound touch probes

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

TS 260: new generation; axial or radial cable connection

Wireless touch probes

With radio or infrared signal transmission for machines with automatic tool changes:

TS 460: new-generation standard touch probe for radio and infrared transmission, with compact dimensions, energy-saving mode, optional collision protection, and thermal decoupling

TS 740: high probing accuracy and repeatability, low probing forces, and infrared transmission

TS 760: highest probing accuracy and repeatability, low probing forces, infrared and radio transmission; for workpiece setup and measuring during machining with integrated workpiece cleaning; EnDat support; energy-saving standby mode.

Transceiver

Radio and infrared communication is established between the TS or TT touch probe and the SE transceiver.

SE 660 for radio or infrared transmission (hybrid technology); for both the TS 460 and TT 460.

SE 661 for radio or infrared transmission (hybrid technology); for both the TS 460 and TT 460; EnDat functionality for transmission of the switching state, diagnostic information, and additional data.

*The SE 661 is only for the TNC7, TNC 640 and TNC 620



Workpiece touch probes	TNC7	TNC 640	TNC 620	TNC 320
		Minimum NC SW	Minimum NC SW	Minimum NC SW
TS 260 ID 738283-xx	81762x-16	34059x-05	81760x-02	771851-02
TS 460 ID 1178530-xx	81762x-16	34059x-05	81760x-02	77185x-02
TS 740 ID 573757-xx	81762x-16	34059x-01	34056x-03/73498x-01/81760x-01	340551-05/771851-01
TS 760	81762x-16	34059x-11 SP5/ 34059x-12 SP5/34059x-16	81760x-07 SP5/81760x-08 SP5/81760x-16	–
TS 642 ID 653217-xx	81762x-16	34059x-09	81760x-06	77185x-06

Installation by the machine manufacturer

For more information, see the *Touch Probes for Machine Tools* brochure

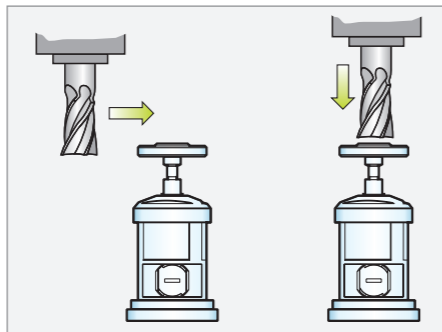
Hardware enhancements

TT: tool touch probes

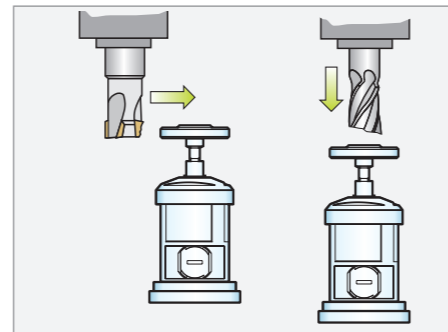
A key factor in obtaining consistently high production quality is, of course, the tool. This requires the exact measurement of tool dimensions and a cyclical inspection of the tool for wear, breakage, and tooth shape. For tool measurement, HEIDENHAIN offers the **TT triggering tool touch probes**.

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 a rotating or stationary tool (e.g., for individual tooth measurement), the contact plate is deflected and a triggering signal is transmitted directly to the TNC. The **TT 160** uses cable-bound signal transmission, whereas the **TT 460** uses radio or infrared transmission. This makes the latter particularly well suited for use on rotary and tilting tables.



TT tool touch probe
Tool length and radius measurement with stationary or rotating spindle



Tool wear measurement and tool breakage monitoring

Tool touch probes	TNC7 Minimum NC SW	TNC 640 Minimum NC SW	TNC 620 Minimum NC SW	TNC 320 Minimum NC SW
TT 160	ID 729763-xx	81762x-16	34059x-05	81760x-02
TT 460	ID 1192582-xx	81762x-16	34059x-05	81760x-02

Installation by the machine manufacturer

For more information, see the *Touch Probes for Machine Tools* brochure

Hardware enhancements

ITC: additional operating station

The additional ITC (Industrial Thin Client) operating stations from HEIDENHAIN are convenient additional remote station solutions for operating the machine or machine units such as tool-changing stations. With a remote operating design tailored to the TNC, the ITC can be readily connected via a standard Ethernet connection with a 100 m (max.) cable.

The **ITC 855** is a compact additional operating station for control systems with a 15-inch or 19-inch main screen. In addition to having an ASCII keyboard and a touchscreen, it is also equipped with the most important function keys of the TNC. The ITC 855 adjusts its resolution automatically to fit the size of the main screen. The soft keys are operated on the touchscreen.

Together with the TE 73x or TE 74x keyboard unit, the **ITC 362** (24-inch screen with multitouch operation) or **ITC 860** (19-inch touchscreen) each make up a full-fledged second operating station. Operation is identical to that of the TNC. The ITC 362 or ITC 860 can also be used as a second-screen solution for an extended workspace.

Convenient plug-and-play technology

As soon as the TNC detects an ITC, the control provides it with an up-to-date operating system. After the ITC has booted, the main screen is mirrored one-to-one. Thanks to this plug-and-play capability, configuration by the machine manufacturer is not needed. With a standard configuration of the X116 Ethernet interface, the TNC automatically integrates the ITC into its system.

You can switch between the TNC and the ITC either by direct takeover or in accordance with a configurable handover principle. Both the booting and shutdown processes are fully controlled by the TNC for maximum operational reliability.



ITC 860

ITC 855
As the "Extended Workspace"

Additional operating station	TNC7 Minimum NC SW	TNC 640 Minimum NC SW	TNC 620 Minimum NC SW	TNC 320 Minimum NC SW
ITC 362	ID 1346871-xx	81762x-16	34059x-07 SP2	–
ITC 855	ID 1370459-01	–	34059x-08	81760x-05
ITC 860	ID 1174935-01	–	34059x-07 SP2	–

Installation by the machine manufacturer

For more information, see the *Information for the Machine Tool Builder* brochures

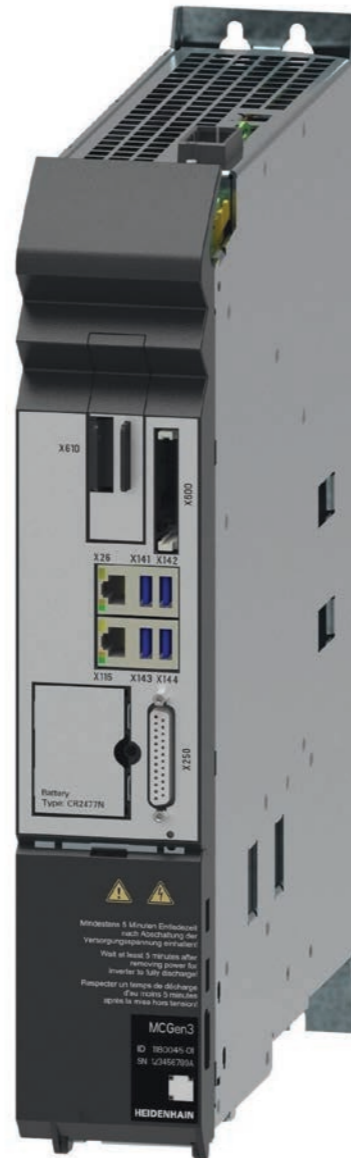
Hardware enhancements

IPC: industrial PC

With the **IPC 306** industrial PC, you can start and remotely operate Windows-based applications via the TNC's user interface, for example. The user interface is displayed on the control screen. Option 133 is required for this.

Since Windows runs on the industrial PC, Windows has no effect on the NC machining process. The IPC is connected to the NC main computer via Ethernet. A second screen is not needed, since the Windows applications are displayed on the TNC's screen via remote access.

In addition to the IPC 306, a separately ordered hard disk is required for operation. The operating systems Windows 8, 10, or 11 can be installed on this empty data medium.



IPC 306

Controls	TNC 640	TNC 620
Industrial PC	Prerequisite: Remote Desktop Manager (option 133)	
IPC 306 With 8 GB of RAM ID 1179966-01		
SSDR IPC 306 ID 1282884-51	Data medium for operating system	
Installation by the machine manufacturer		
For more information , see the <i>Information for the Machine Tool Builder</i> brochures		

Hardware enhancements

VT 121: vision system for tool inspection

The vision system for tool inspection consists of two components:

- Camera with two objectives (VT 121)
- Touch-operated PC software (VTC)

The vision system enables tool imaging during machining. Besides documenting tool status and wear, the following applications are possible as well:

- Tool inspection before critical machining steps
- Optimization of cutting parameters
- Optimization of NC programs
- Breakage inspection
- Tool inspection after service life

The camera takes close-up images of each tooth as well as detailed panoramic images of the entire tool circumference. During inspection with the VTC, the lighting angle can be varied for these panoramic images, enabling optimal illumination of individual teeth. Tools can also be imaged from below. The VTC can run automatically during unattended shifts (with cycles for the TNC7 or TNC 640). Via an interface to the TNC's tool table, the PC software can even lock tools as needed.

The vision system for tool inspection not only helps you avoid expensive damage to the tool, workpiece, and machine, but even the tool costs themselves can be reduced in the long term through targeted minimization of tool wear. The vision system also provides the following benefits:

- Automated imaging during machining
- Time savings through keeping the tool in the machine
- PC software (can be used on its own for later inspection)
- Compact system usable for even large tools
- Rugged design
- Controlled compressed-air cleaning

The sealed and highly rugged VT 121 vision system is designed to be installed inside the machine's work envelope. It requires compressed air only during tool clearing cycles. The vision system can be used regardless of whether cooling lubricant or

dry machining is performed. Compressed air from the integrated jets cleans the tools and the camera's cover glass. An optimal cleaning strategy ensures removal of nearly all chips. The vision system can be connected via its Gigabit Ethernet interface.



Vision system Tool Inspection	TNC7 Minimum NC SW	TNC 640 Minimum NC SW	TNC 620 Minimum NC SW	TNC 320 Minimum NC SW
VT 121 1249466-01	81762x-16	34059x-10	–	–
Installation by the machine manufacturer				
For more information , see the <i>VT 121 Vision System for Tool Inspection</i> Product Information document				

HEIDENHAIN

Mastering nanometer accuracy



HEIDENHAIN

DR. JOHANNES HEIDENHAIN GmbH

Dr.-Johannes-Heidenhain-Straße 5

83301 Traunreut, Germany

☎ +49 8669 31-0

☎ +49 8669 32-5061

✉ info@heidenhain.de

www.heidenhain.com



HEIDENHAIN
worldwide