HEIDENHAIN



Options and Accessories

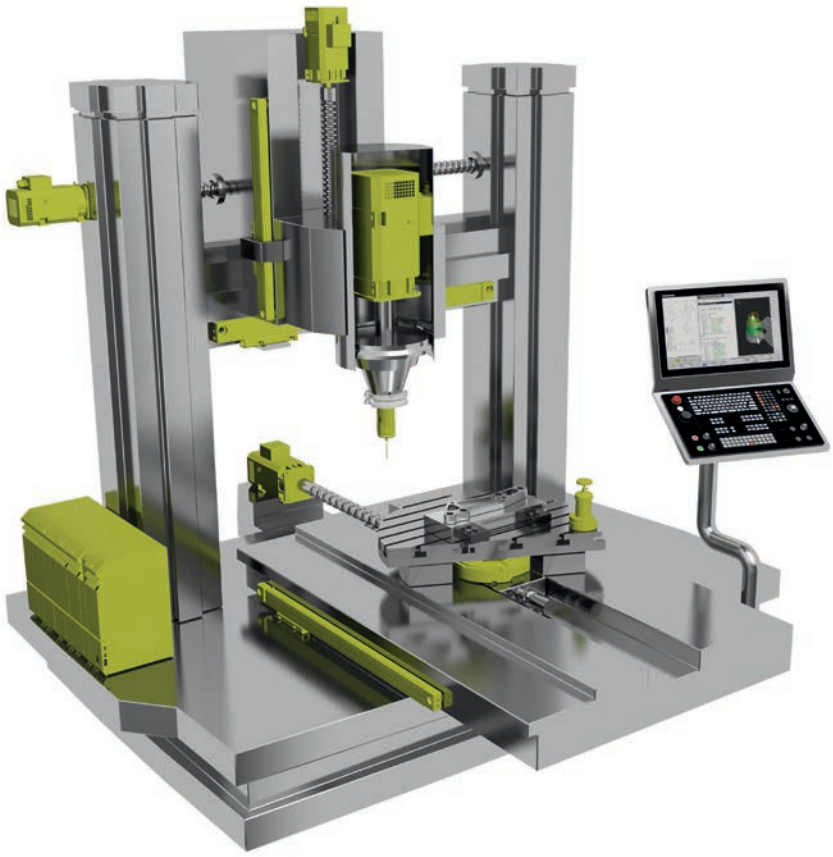
For TNC Controls

www.heidenhain.com/cnc-controls

02/2026

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. The software options are saved on the SIK plug-in board. The TNC7 and TNC7 basic can be operated with the present SIK or, in the future, with the new SIK2.

SIK2 is available starting with these software versions:
TNC7: 81762x-18 SP1
TNC7 basic: 81762x-18 SP1

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.

Windows 8 and Windows 10 are trademarks of Microsoft Corporation

Contents

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**
	Machine interfacing	**57**
Software		**64**
Hardware enhancements		**79**

Please also note the page references in the overview tables.

Overview

Option number SIK	Option number SIK2	Option	TNC 320	TNC 620	TNC 640	TNC7	TNC7 basic	Page
		Machining functions						
8	1-01-1	**Adv. Function Set 1** | Programming rotary-table machining • Programming cylindrical contours on an unrolled cylinder • Feed rate in mm/min or degrees/min	•	•	•	•	•	14
		Adv. Function Set 1 | Coordinate conversion • Tilting the working plane, PLANE function	•	•	•	•	•	15
		Adv. Function Set 1 | Interpolation • Circular in 3 axes with tilted working plane	•	•	•	•	•	15
9	4-01-1	**Adv. Function Set 2** | Interpolation • Linear in 5 axes	–	•	•	•	•	–
		Adv. Function Set 2 | 5-axis simultaneous machining • 3D tool compensation via surface normal vectors • Changing the swivel-head angle with the electronic handwheel during program run without affecting the tool tip position (TCPM = Tool Center Point Management) • Keeping the tool perpendicular to the contour • Tool radius compensation perpendicular to the tool direction • Manual traverse in the active tool-axis system	–	•	•	•	–	41
17	1-05-1	**Touch Probe Functions** | Streamlining setup and inspection with probing cycles • Workpiece misalignment compensation, preset setting • Automatic tool and workpiece measurement • Touch-probe input enabling for non-HEIDENHAIN systems	✓	•	✓	✓	•	16
19	–	**Advanced Programming Features** | Expanded programming capabilities • FK free contour programming • Canned cycles • Peck drilling, reaming, boring, counterboring, centering • Milling of internal and external threads • Clearing level and oblique surfaces • Complete 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	✓	•	✓	✓	✓	18
20	–	**Advanced Graphic Features** | Visualization for verification and program run • Plan view • Projection in multiple planes • 3D view	✓	•	✓	✓	✓	19
		Advanced Graphic Features | Detailed 3D visualization	✓	•	✓	✓	✓	20
21	4-02-1	**Adv. Function Set 3** | Superimposing handwheel positioning during program run	✓	•	✓	✓	•	42
		Adv. Function Set 3 | Calculating contours with tool-radius offset in advance (LOOK AHEAD)	✓	•	✓	✓	•	43

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

Option number SIK	Option number SIK2	Option	TNC 320	TNC 620	TNC 640	TNC7	TNC7 basic	Page
		Machining functions						
22	**–**	**Pallet Management** | Orchestrating automated pallet machining	–	✓	✓	✓	✓	21
40	**5-03-1**	**Collision Monitoring** | Preventing crashes with dynamic collision monitoring (DCM)	–	–	•	•	•	44
42	**1-03-1**	**CAD Import** | DXF converter, importing contours and machining positions from DXF files	•	•	•	•	•	23
		CAD Import | Importing of contours from 3D models	•	•	•	•	•	24
44	**1-06-1**	**Global PGM Settings** | Applying configurable settings to entire programs	–	–	•	•	–	46
45	**2-31-1**	**Adaptive Feed Contr.** | Adaptive Feed Control (AFC)	–	–	•	•	•	47
50	**4-03-1**	**Turning** | Mill-turning functions • Turning tool management • Tool radius compensation • Switching between milling and turning mode • Turning-specific contour elements • Turning cycle package	–	–	•	•	–	26
		Turning | Eccentric turning	–	–	•	•	–	27
92	**2-02-1**	**3D-ToolComp** | 3D radius offset based on tool angle (requires the **Adv. Function Set 2** software option)	–	–	•	•	–	52
93	**2-03-1**	**Ext. Tool Management** | Managing tools based on NC program requirements	•	•	•	•	•	29
96	**7-04-1**	**Adv. Spindle Interpol.** | Interpolating spindle | Interpolation turning, shaping, turning tool table	–	–	•	•	–	30
131	**7-02-1**	**Spindle Synchronism** | Synchronizing spindle motion • Synchronization of two or more spindles • Hobbing cycle (additionally required: the **Turning** option)	–	–	•	•	–	31
140	**5-03-2**	**Collision Monitoring v2** | DCM version 2 for crash prevention (automatically enables the **Collision Monitoring** software option)	–	–	–	•	•	45
145	**2-30-1**	**Active Chatter Contr.** | Active chatter control (ACC)	–	•	•	•	•	48
152	**1-04-1**	**CAD Model Optimizer** | Cleaning up 3D models for use on the control • Generate correct STL files from STEP files • Repair existing STL files	–	•	•	•	•	25
154	**2-05-1**	**Batch Process Mngr.** | Depiction of pallet management in a clear format	–	•	•	•	•	22
156	**4-04-1**	**Grinding** | Cycles for jig grinding and cylindrical grinding	–	–	•	•	–	32
157	**4-05-1**	**Gear Cutting** | Creating gears	–	–	•	•	–	33
158	**4-03-2**	**Turning v2** | Version 2 of the mill-turning functions (automatically enables the **Turning** option)	–	–	•	•	–	28
159	**1-07-1**	**Model Aided Setup** | Graphically supported workpiece setup	–	–	–	•	•	17
167	**1-02-1**	**Opt. Contour Milling** | Optimized trochoidal milling (OCM)	•	•	•	•	•	49

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

Overview

Option number SIK	Option number SIK2	Option	TNC 320	TNC 620	TNC 640	TNC7	TNC7 basic	Page
		Machine accuracy						
48	2-01-1	**KinematicsOpt** | Touch probe cycles for automatic calibration measurements on rotary axes	–	•	•	•	•	34
52	2-04-1	**KinematicsComp** | Correcting 3D spatial errors	–	–	•	•	–	35
141	2-20-1	**CrossTalk Comp.** | Compensation of axis couplings (CTC)	–	•	•	•	•	36
142	2-21-1	**Position Adapt. Contr.** | Position adaptive control (PAC)	–	•	•	•	•	37
143	2-22-1	**Load Adapt. Contr.** | Load adaptive control (LAC)	–	•	•	•	•	38
144	2-23-1	**Motion Adapt. Contr.** | Motion adaptive control (MAC)	–	•	•	•	•	39
146	2-24-1	**Machine Vibr. Contr.** | Vibration damping for machines (MVC)	–	•	•	•	•	40
155	5-02-1	**Component Monitoring** | Monitoring loads on machine components	–	•	•	•	•	50
168	5-01-1	**Process Monitoring** | Monitoring the production process based on a teach-in cut	–	–	–	•	–	51
		Communication						
18	3-03-1	**HEIDENHAIN DNC** | Communicating with external Windows applications via COM component	•	•	•	•	•	54
56 – 61	3-02-1*	**OPC UA NC Server** | Standardized interface for accessing TNC data and functions within a network	–	•	•	•	•	53
133	3-01-1	**Remote Desk. Manager** | Viewing and remotely operating external computers (e.g., Windows PCs)	•	•	•	•	•	56
–	3-04-01	**Process Tracking Interface (PTI)** | Interface for recording control signals and process signals.	–	–	–	•	•	55
		Machine interfacing						
0	6-01-1*	**Control Loop Qty. (Additional Axis 1-8)** | Additional control loops — Additional Axis 1	•	•	•	•	•	57
1		Additional Axis 2	•	•	•	•	•	
2		Additional Axis 3	–	•	•	•	•	
3		Additional Axis 4	–	•	•	•	•	
4		Additional Axis 5	–	–	•	•	–	
5		Additional Axis 6	–	–	•	•	–	
6		Additional Axis 7	–	–	•	•	–	
7		Additional Axis 8	–	–	•	•	–	

* This software option can be ordered multiple times in the desired quantity.
The control automatically takes all enablings into account.

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

Option number SIK	Option number SIK2	Option	TNC 320	TNC 620	TNC 640	TNC7	TNC7 basic	Page
		Machine interfacing						
24	**6-03-1**	**Gantry Axes** | Operating synchronized axes: gantry axes, tandem tables	•	•	✓	✓	•	59
46	**7-01-1**	**Python OEM Process** | Implementing Python OEM applications	•	•	•	•	•	60
49	**6-02-1**	**Double Speed Axes** | Double-speed control loops	–	•	•	•	•	61
77	**6-01-1***	**4 Additional Axes** | 4 additional control loops	–	–	•	•	•	57
78		**8 Additional Axes** | 8 additional control loops	–	–	•	•	–	
101 – 130	**–**	**OEM Option** | Enabling options provided by the OEM	–	–	•	–	–	62
135	**7-03-1**	**Synchronizing Functions** | Real-time coupling (RTC) for synchronizing axes and spindles	–	–	•	•	–	63
160	**6-30-1**	**Integrated FS: Basic** | Gen 3 exclusive: enabling of functional safety (FS) and four safe control loops	–	•	•	•	•	58
161	**6-30-2***	**Integrated FS: Full** | Gen 3 exclusive: enabling functional safety (FS) and the maximum number of functionally safe control loops (10 or more)	–	•	•	•	•	
162		**FS Control Loop Qty. (Add. FS Ctrl. Loop 1)** | Gen 3 exclusive: additional functionally safe control loop (no. 1)	–	•	•	•	•	
163		**FS Control Loop Qty. (Add. FS Ctrl. Loop 2)** | Gen 3 exclusive: additional functionally safe control loop (no. 2)	–	•	•	•	•	
164		**FS Control Loop Qty. (Add. FS Ctrl. Loop 3)** | Gen 3 exclusive: additional functionally safe control loop (no. 3)	–	•	•	•	•	
165		**FS Control Loop Qty. (Add. FS Ctrl. Loop 4)** | Gen 3 exclusive: additional functionally safe control loop (no. 4)	–	•	•	•	•	
166		**FS Control Loop Qty. (Add. FS Ctrl. Loop 5)** | Gen 3 exclusive: additional functionally safe control loop (no. 5)	–	•	•	•	•	
169		**FS Control Loop Qty. (Add. FS Full)** | Gen 3 exclusive: remaining functionally safe control loops enabled	–	•	•	•	•	

* This software option can be ordered multiple times in the desired quantity.
The control automatically takes all enablings into account.

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

Overview

Software	TNC 320	TNC 620	TNC 640	TNC7	TNC7 basic	Subject to a charge	Page
ConfigDesign | Configuration of the machine parameters	•	•	•	•	•		–
PCtoolsUpdateManager | Update manager for HEIDENHAIN software	•	•	•	•	•		–
PLCtext | Editing 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 (requires the **HEIDENHAIN DNC** software option)	•	•	•	•	•	•	54
TNCremo | Data transfer software	•	•	•	•	•		64
TNCremoPlus | Data transfer software with "live" screen	•	•	•	•	•	•	64
RemoteAccess | Software for remote diagnostics	•	•	•	•	•	•	65
StateMonitor | Collect and evaluate machine data	•	•	•	•	•	•	66
PLCdesign | PLC diagnostics for all HEIDENHAIN controls	•	•	•	•	•		68
KinematicsDesign | Creation of kinematic models for numerical controls	•	•	•	•	•		69
M3D Converter | Creation of high-resolution collision objects	•	•	•	•	•	•	69
CycleDesign | Incorporation of developed NC subprograms as cycles	•	•	•	•	•		70
TNCscope | Recording and evaluation of, for example, axis curves of HEIDENHAIN controls	•	•	•	•	•		71
TNCdiag | Evaluation of diagnostic information of digital motor systems	–	•	•	•	•		72
TNCopt | Initial setup of digital control loops	–	•	•	•	•		73
IOconfig | Configuration of, for example, handwheels	•	•	•	•	•		74
Software Key Generator / Web application | For enabling of software options	•	•	•	•	•		75
BMXdesign | Designing language-sensitive soft keys	•	•	•	•	•		76
Programming station	•	•	•	•	•		77
virtualTNC | For PC-based machine simulation	–	•	•	•	•	•	78

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

Hardware enhancements			TNC 320	TNC 620	TNC 640	TNC7	TNC7 basic	Must be adapted by the OEM	ID	Page
Handwheel	**HR 130** TTL; panel mounted	with detent w/o detent	•	•	•	•	•	No	540940-01 540940-03	79
	HR 180	with detent w/o detent	•	•	•	•	•	No	540940-16 540940-17	
	HR 510 portable handwheel	with detent w/o detent	•	•	•	•	•	No	1120313-xx 1119971-xx	
	HR 510 FS portable handwheel	with detent w/o detent	•	•	•	•	•	No	1119974-xx 1120311-xx	
	HR 520 portable handwheel with display	with detent w/o detent	•	•	•	•	•	No	670303-xx 670302-xx	
	HR 520 FS portable handwheel with display	with detent w/o detent	•	•	•	•	•	No	670305-xx 670304-xx	
	HR 550 FS portable radio handwheel with display	with detent w/o detent	•	•	•	•	•	No	1183021-xx 1200495-xx	
	HRA 551 FS handwheel holder for HR 550 FS		•	•	•	•	•	No	1119052-xx	
Workpiece touch probe	**TS 260** with cable		•	•	•	•	•	Yes	738283-xx	80
	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		–	•	•	•	•	Yes	1283267-xx	
Tool touch probe	**TT 160** with cable		•	•	•	•	•	Yes	729763-xx	81
	TT 460 with radio or infrared transmission		•	•	•	•	•	Yes	1192582-xx	
Vision system	**VT 121/VT 122** for tool inspection		–	–	•	•	•	Yes	1249466-01/ 1373589-xx	82
Additional operating station	**ITC 362** with 24-inch touchscreen		–	–	•	•	–	Yes	1346871-xx	84
	ITC 855 with 15-inch touchscreen and ASCII keyboard		–	•	•	–	–	Yes	1370459-xx	
	ITC 860 with 19-inch touchscreen; separate TNC keyboard required		–	–	•	–	–	Yes	1174935-xx	
	ITC 352 with 19-inch touchscreen		–	–	–	•	–	Yes	1374639-xx	
	ITC 342 with 16-inch touchscreen		–	–	–	–	•	Yes	1354570-xx	
Override controller	**OC 310**		–	–	–	•	•	Yes	1410803-xx	85
Industrial PC	**IPC 306** with 8 GB RAM		–	•	•	•	•	Yes	1179966-01	86
Tool breakage detector	**TD 110** tool breakage detector for machine tools		•	•	•	•	•	Yes	1377561-01	87

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

Overview
Dynamic Precision

dynamic + precision

The concept of Dynamic Precision comprises optional functions for HEIDENHAIN controls that effectively reduce the dynamic errors 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.

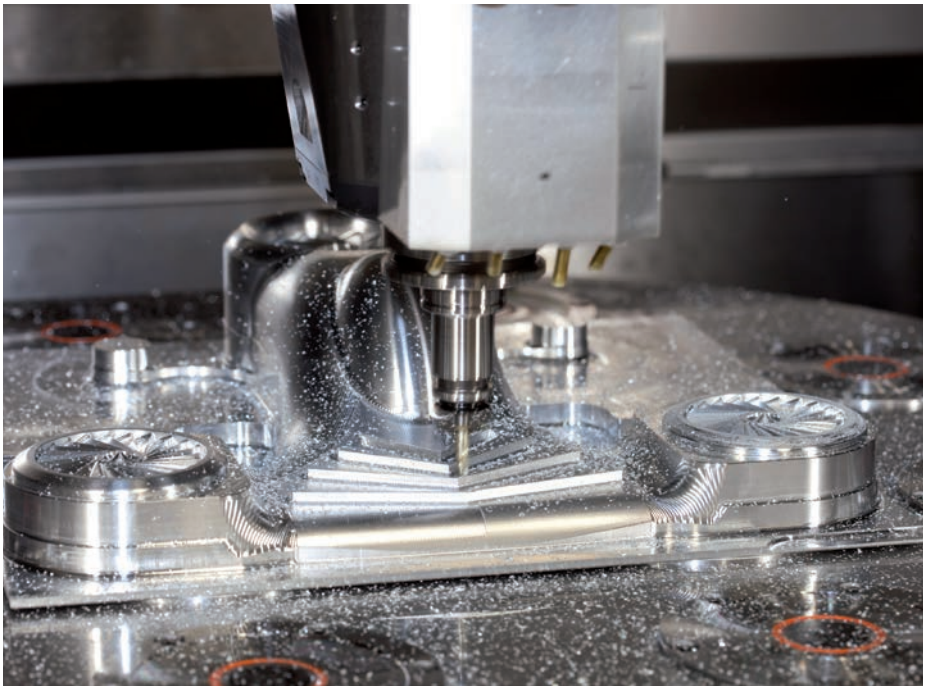
High precision and fast machining are the basis for increased productivity. Unit cost is therefore reduced without sacrificing accuracy or surface quality. Dynamic Precision even ensures that accuracy is maintained regardless of the operating time and load. As a result, feed rates no longer need to be 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 TNC7 basic	TNC 640	TNC 620	Page
Dynamic Precision	✓	✓	✓	
Cross Talk Comp. (CTC) Compensation of position errors through axis coupling	Option	Option	Option	36
Machine Vibr. Contr. (MVC) Damping of machine oscillations	Option	Option	Option	40
Position Adapt. Contr. (PAC) Position-dependent adaptation of control parameters	Option	Option	Option	37
Load Adapt. Contr. (LAC) Load-dependent adaptation of controller parameters	Option	Option	Option	38
Motion Adapt. Contr. (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

Dynamic Efficiency

With the concept of Dynamic Efficiency, HEIDENHAIN combines innovative TNC functions that help the user to make heavy machining and roughing more efficient while also enhancing process reliability. These software functions not only assist the user but also make the manufacturing process itself faster, more stable, and more predictable—in short, more efficient. Dynamic Efficiency helps boost the metal removal rate and reduce machining time.

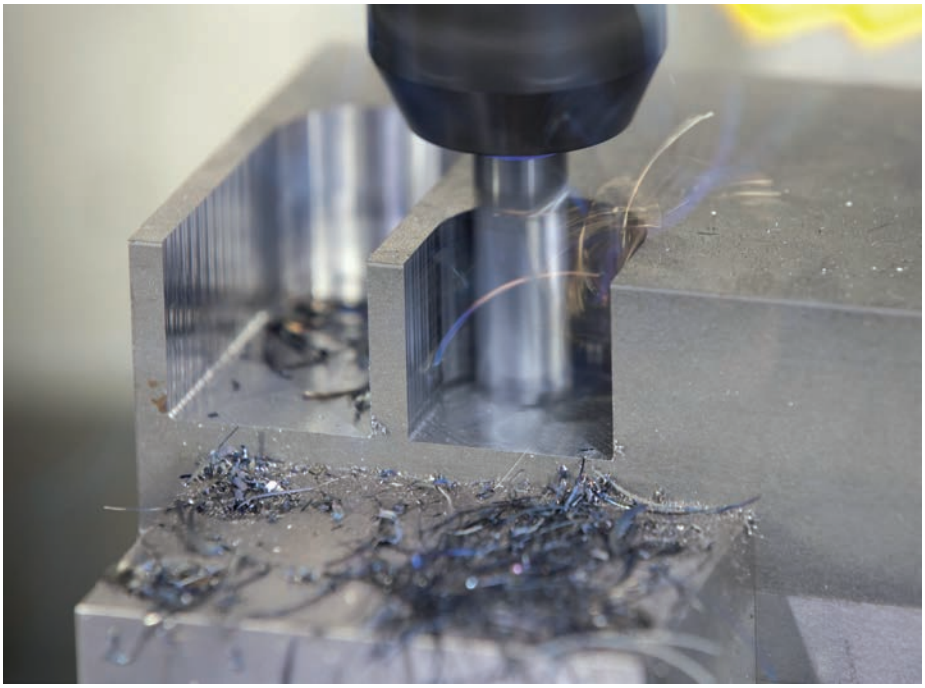
Dynamic Efficiency encompasses four software functions:

- 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 users 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 TNC7 basic	TNC 640	TNC 620	Page
Dynamic Efficiency	✓	✓	–	
Active Chatter Contr. (ACC) Active chatter control	Option	Option	Option	48
Adaptive Feed Contr. (AFC) Adaptive feed control	Option	Option	–	47
Trochoidal milling	Standard function	Standard function	Standard function	–
Opt. Contour Milling (OCM) Optimized contour machining	Option	Option	Option	49

Installation by the machine manufacturer or user

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

Connected Machining
Solutions for networked manufacturing

connected + machining

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 user 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 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) for monitoring and remote operation of machines, including older models.
- The Remote Desktop Manager software option enables access on the control to Windows PCs and the software installed on them.
- Firefox web browser for accessing web-based applications, such as ERP and MES clients, email clients, and HTML5 applications.



	TNC7 TNC7 basic	TNC 640	TNC 620	Page
Connected Machining	✓	✓	✓	
Remote Desk. Manager	Option	Option	Option	56
HEIDENHAIN DNC	Option	Option	Option	54
OPC UA NC Server	Option	Option	Option	53
Extended Workspace	Option	Option	–	84

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
- MTConnect
- Modbus TCP
- FOCAS

HEIDENHAIN
StateMonitor

Software options for use with StateMonitor

HEIDENHAIN
JobTerminal

JobTerminal manages job data, job times and job analyses.

HEIDENHAIN
MaintenanceManager

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

HEIDENHAIN
5 Signals

5 Signals records additional machine data via PLC signals.

HEIDENHAIN
5 Machines

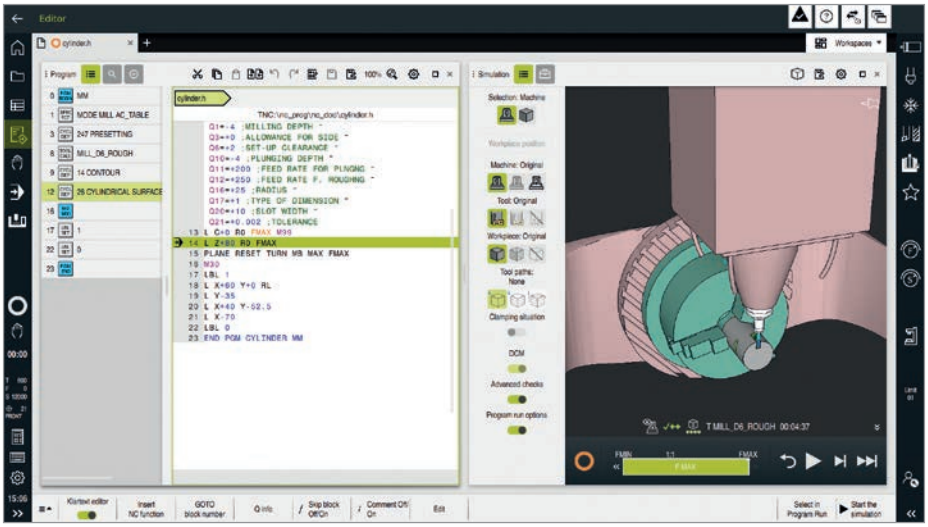
5 Machines allows you to connect an additional five machine tools.

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



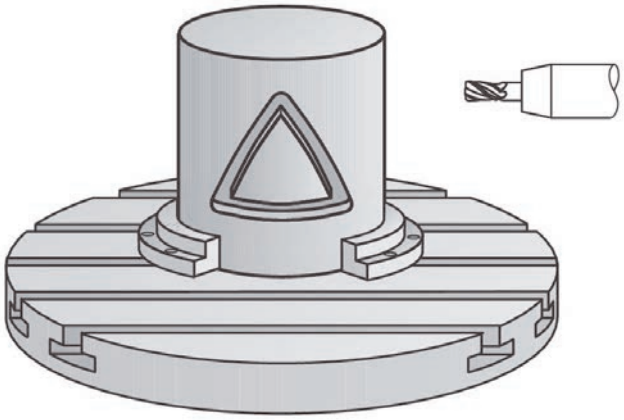
Programming and operation
Machining with a rotary table

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
Programming contours consisting of cycles, straight lines, and arcs on cylindrical surfaces using rotary and titling tables is easy with the TNC. Simply program the contour in the plane (regardless of axes) as if the cylinder surface were unrolled. The TNC then executes the programmed machining operation on the cylinder surface.

The TNC provides the following for cylinder surface machining:
- Slot milling (the slot width and tool diameter are identical)
- Guide-groove milling (the slot width is greater than the tool diameter)
- Ridge milling
- Outside contour milling
- NC function for cylinder kinematics: This function allows you to machine the cylinder surface with various NC functions (e.g., OCM cycles, pocket milling cycles, or path functions). This function allows you to perform milling and drilling operations on a cylindrical surface. A variety of standardized machining cycles can be used, thereby greatly simplifying the implementation of complex machining tasks on cylindrical geometries.



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.

Adv. Function Set 1	**SIK**	**SIK2**
	Option 8	1-01-1
TNC7/TNC7 basic/TNC 640/TNC 620 TNC 320	ID 617920-01 ID 536146-01	ID 1395831-01
TNC7 **TNC7 basic** **TNC 640** **TNC 620** **TNC 320**	NC SW 81762x-16 and later NC SW 81762x-18 and later NC SW 34059x-01 and later NC SW 34056x-01/73498x-01/81760x-01 and later NC SW 34055x-01/771851-01 and later	
Installation by the machine manufacturer		
For more information, see the TNC brochures		

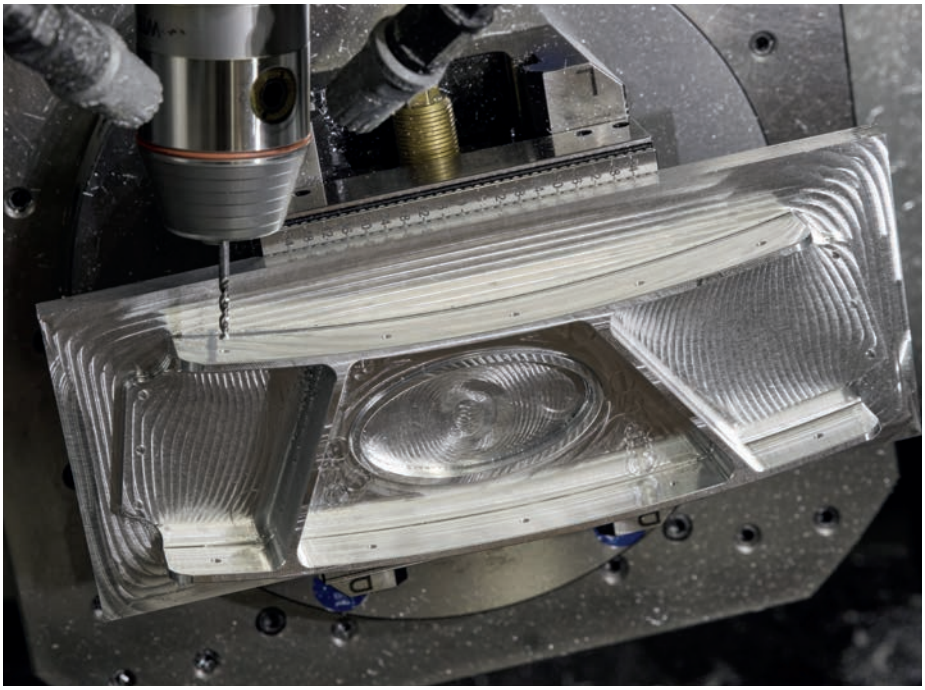
Programming and operation

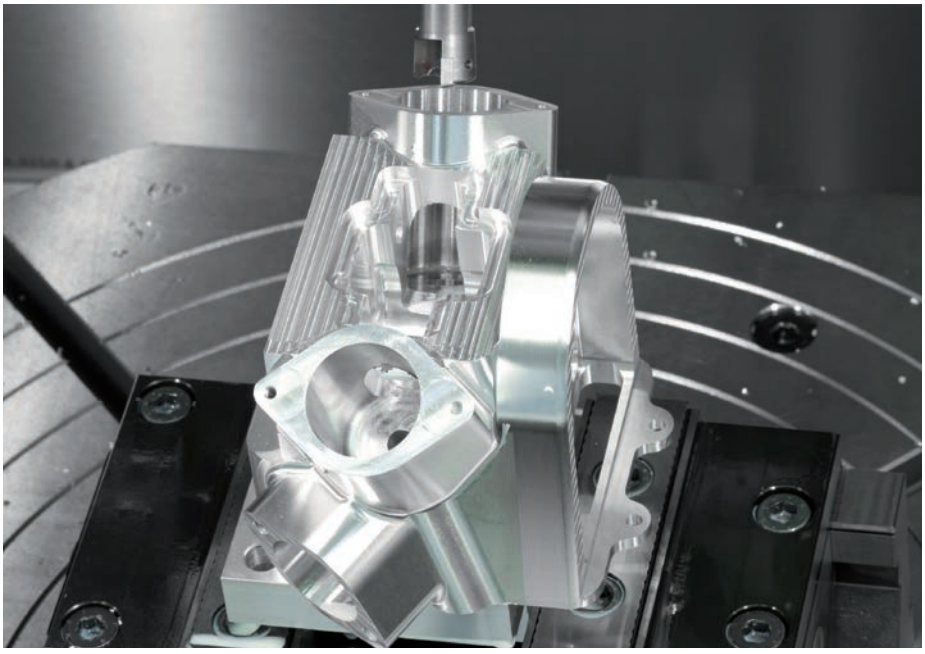
Coordinate conversion by tilting the working plane and using the PLANE function

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 program the machining operation as usual in the working plane—for example, in X/Y. The machine then runs the program in a plane that has been tilted with respect to the working 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.





Adv. Function Set 1	**SIK**	**SIK2**
	Option 8	1-01-1
TNC7/TNC7 basic/TNC 640/TNC 620 TNC 320	ID 617920-01 ID 536146-01	ID 1395831-01

TNC7	NC SW 81762x-16 and later
TNC7 basic	NC SW 81762x-18 and later
TNC 640	NC SW 34059x-01 and later
TNC 620	NC SW 34056x-01/73498x-01/81760x-01 and later
TNC 320	NC SW 34055x-01/771851-01 and 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

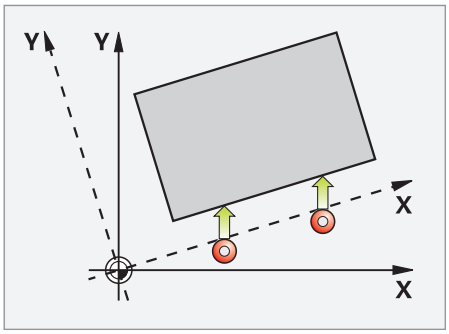
Workpiece alignment
The probing functions of the TNC reduce the need for tedious manual alignment of the workpiece:
- Workpiece clamped in any position
- The touch probe determines the actual clamping situation by probing a surface
- The TNC compensates for this misalignment with a "basic rotation," 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

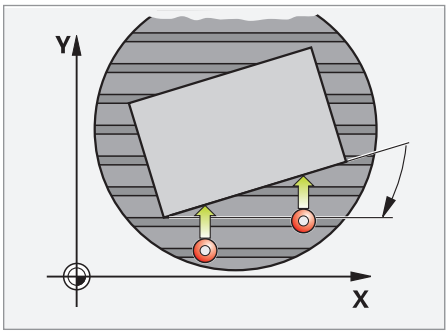
Preset setting
Being able to define the preset quickly and reliably reduces nonproductive time and increases machining accuracy. The TNC features probing cycles for automatically setting presets.

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.

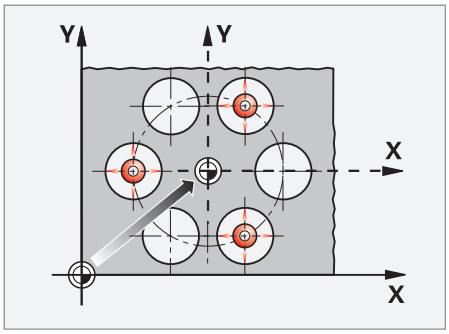
Tool measurement
In conjunction with the TT touch probes for tool measurement, the TNC makes it possible to automatically measure tools inside the machine. The measured tool length and radius are stored by the TNC in its central tool memory. In-process tool measurement allows you to immediately identify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

On the TNC 620, the touch probe cycles are available only with the **Touch Probe Functions** software option. This option is automatically enabled when using HEIDENHAIN touch probes featuring the EnDat interface (NC SW 81760x-06 and later). The TNC7, TNC 640 and TNC 320 already include the touch probe cycles as standard functions.

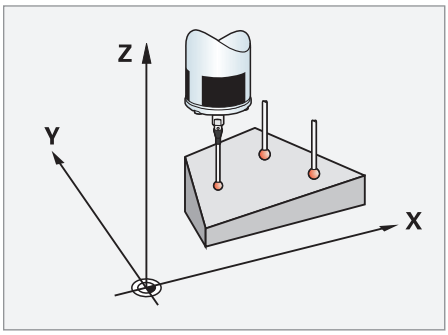




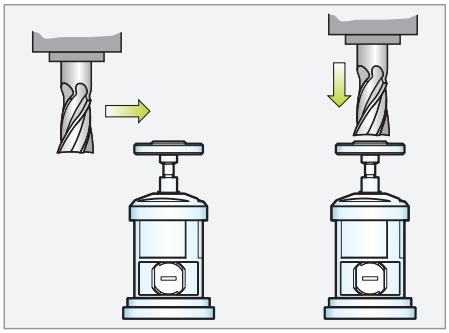
Compensating for workpiece misalignment by rotating the coordinate system or rotating the table

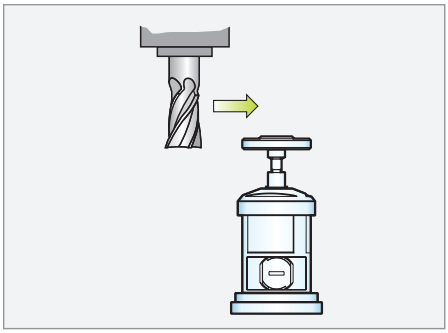


Preset setting



Workpiece measurement





Tool measurement

Touch Probe Functions	SIK	SIK2
	Option 17	1-05-1
	ID 634063-01	ID 1395851-01
TNC7	Standard function	
TNC7 basic	NC SW 81762x-18 and later	
TNC 640	Standard function	
TNC 620	NC SW 34056x-01/73498x-01/81760x-01 and later	
TNC 320	Standard function	
Installation by the machine manufacturer		
For more information, see the TNC brochures and the *Touch Probes for Machine Tools* brochure		

Programming and operation
Graphical 6D workpiece setup

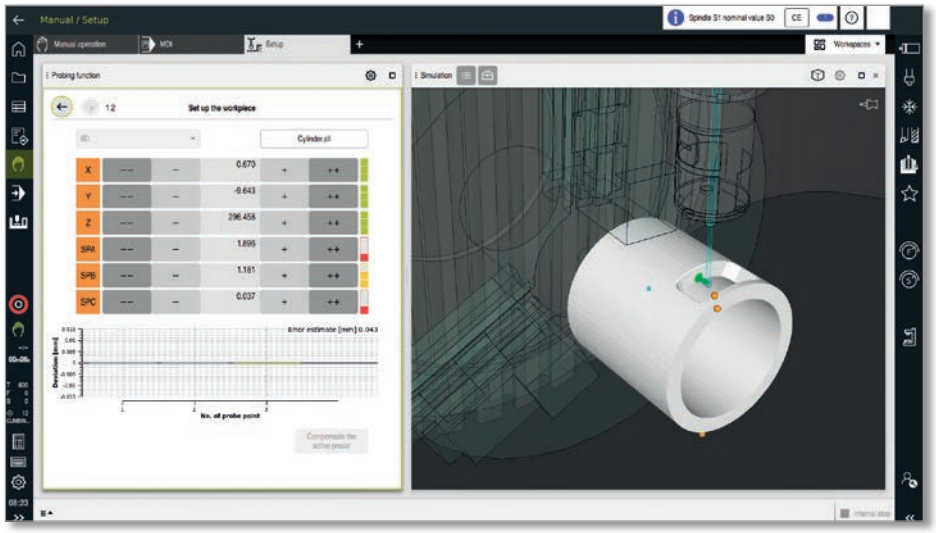
For single parts and small lot sizes without a special holder, the position of the workpiece blank almost always needs to be ascertained before machining. With this intuitive probing function, the TNC7 and TNC7 basic provide users with graphical guidance that makes workpiece setup fast, easy, and reliable. The exact position of the workpiece blank is measured in the machine's working space and reported to the control. This graphically supported measurement can be used for any workpieces. An accurate 3D model is all that is needed. You can simplify and clean up the 3D model of your workpiece using the CAD Model Optimizer software option in order to create a valid STL file for the measurement procedure. But you can also select an NC program within the function. The control generates the 3D model based on the workpiece blank defined in the NC program.*

The simulation view depicts a 3D model of the workpiece blank in the machine's work envelope. By default, the control positions the 3D model at the active workpiece preset.* After the user 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 user about the quality of the probing points for determining the position and orientation of the workpiece. The user can thus quickly see when the measurement of the actual position and orientation of the workpiece is complete. The user 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 user to align even complex workpiece blanks relative to pre-machined features, such as is needed for mold repair or 3D-printed workpiece blanks.

Error estimate diagram*
Every additional touch point gradually restricts the possible positioning of the workpiece and puts the 3D model closer to the actual position in the machine. For each probe point, the error estimate diagram shows the distance of the 3D model from the real workpiece.

* Software 81762x-19 or later



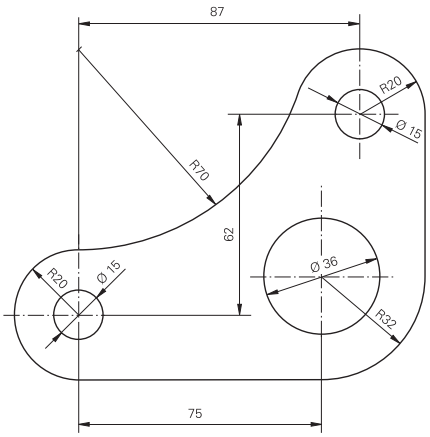
Model Aided Setup	SIK	SIK2
	Option 159	1-07-1
	ID 1364052-01	ID 1395855-01
TNC7	NC SW 81762x-17 and later	
TNC7 basic	NC SW 81762x-18 and later	
TNC 640	–	
TNC 620	–	
TNC 320	–	
Installation by the user		
For more information, see the *TNC7* brochure, and visit www.klartext-portal.com		

Programming and operation

Advanced programming functions, including FK free contour programming and canned cycles

FK free contour programming
Not all workpieces are suitably dimensioned for conventional NC programming. Thanks to the FK free contour programming function 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 solution, then the TNC programming graphics render the possible variants to choose from. On the TNC7 and TNC7 basic, graphical programming replaces FK programming as a standard feature.





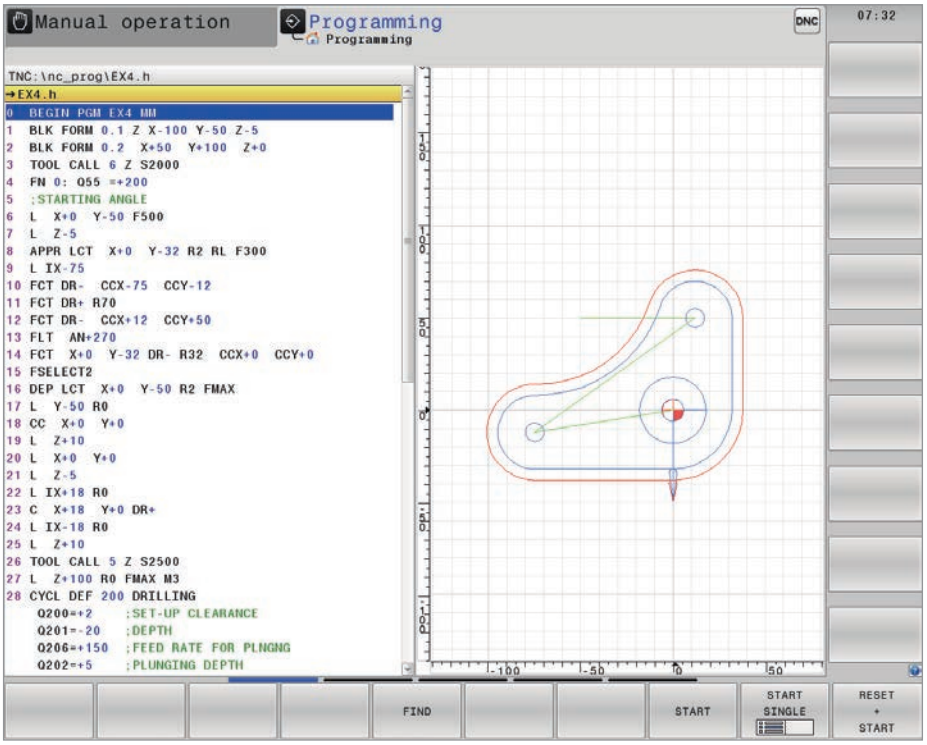
Standard cycles
In addition to the canned cycles for drilling and tapping (with or without a floating tap holder), the **Advanced Programming Features** software option also gives you cycles for thread milling, reaming, boring, and hole patterns, as well as milling cycles for plane-surface face milling 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** 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 supports you 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 user draws the desired contours directly on the touchscreen. The control then converts the drawing into a Klartext program. This function makes it very easy to modify existing programs.

* Available only for TNC7 and TNC7 basic



Programming graphics of the TNC 640

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

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 640, TNC 620 and TNC 320, and are described in their respective brochures.

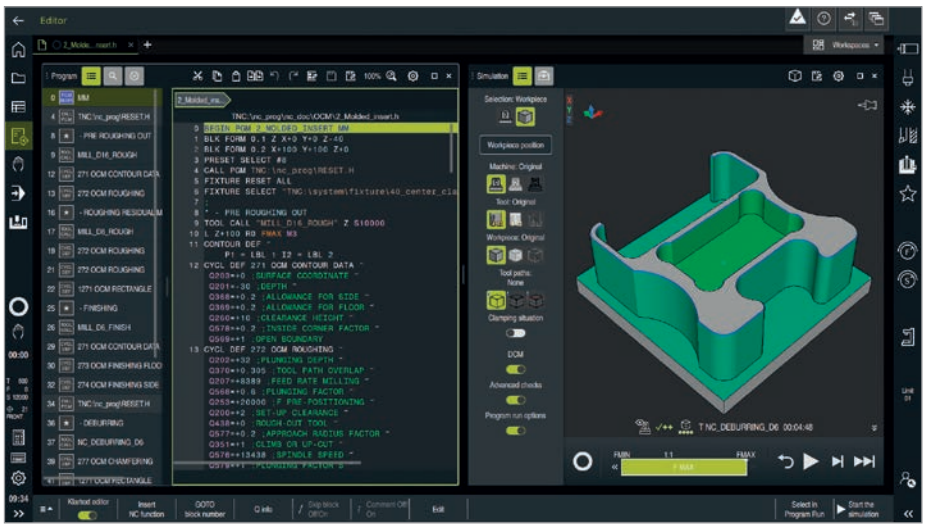
Other graphical views are available as options:

Simulation
To be on the safe side before running a part 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, TNC7 basic, 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 and TNC7 basic offer a function that enables a comparison between 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 on the TNC7



Program-run graphics of the TNC7

Advanced Graphic Features	**SIK** Option 20 ID 628253-01
TNC7	Standard function
TNC7 basic	Standard function
TNC 640	Standard function
TNC 620	NC SW 34056x-01/73498x-01/81760x-01 and later
TNC 320	Standard function
Installation by the user	
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.

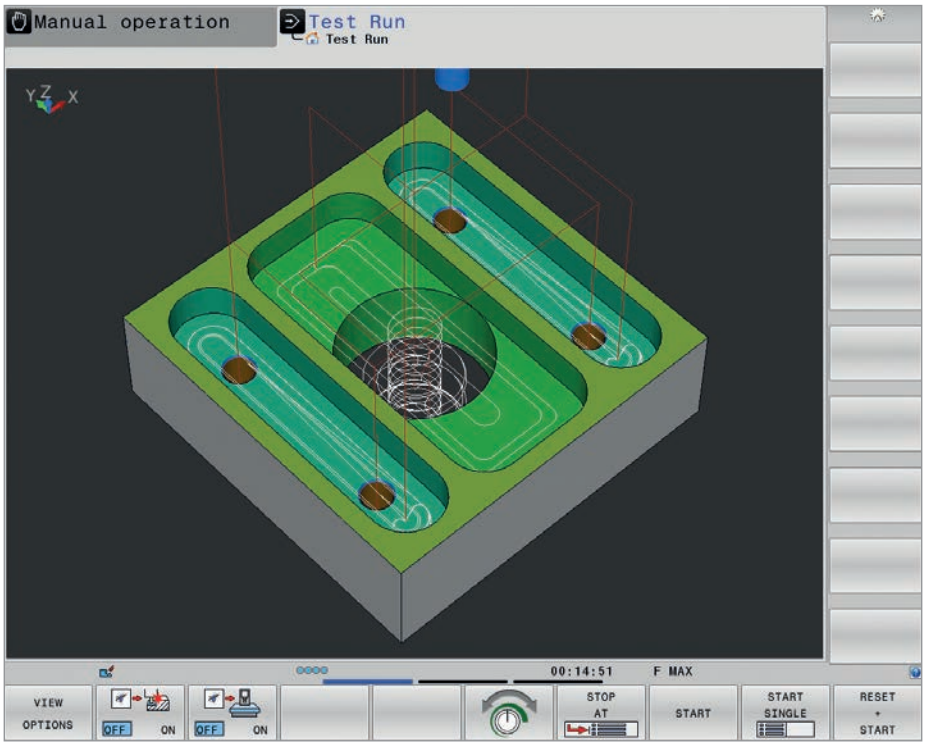
The high-resolution 3D view with strong zoom factors reveals even the tiniest program errors on the workpiece surface. For an even more precise view of the NC data analysis, you can also display the tool paths and their corresponding block numbers. Of particular help is the selectable workpiece transparency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

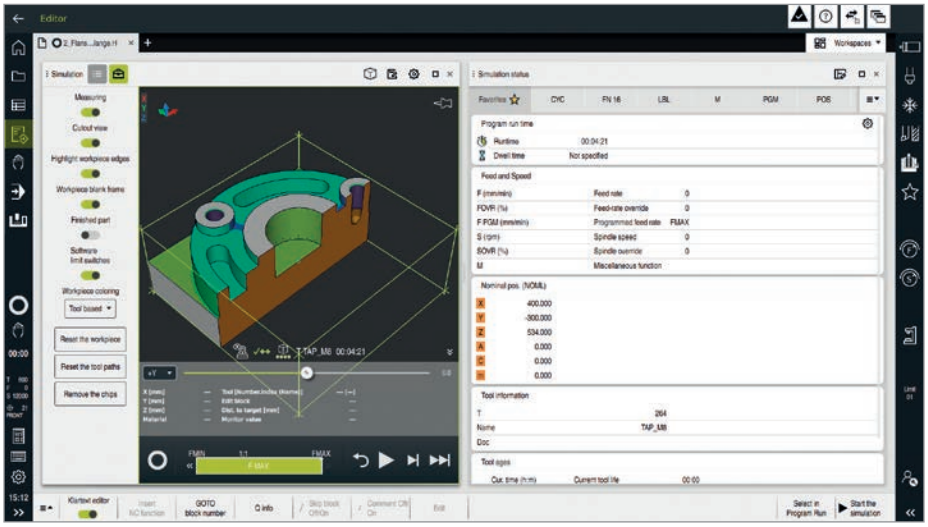
Enhanced graphics also make tool-specific machining operations easier to see: each machining operation performed with the same tool is shown in its own unique color. In 3D view, the TNC also includes a measuring function that displays the coordinates when you shift the focus within the graphics by touch.

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), the TNC7, and the TNC7 basic simulate not only material removal but also the complete 3D machine model, allowing the user to precisely evaluate processes in the machine envelope before machining. The machine's collision objects are configured and activated by the machine manufacturer.

The TNC7 and TNC7 basic, TNC 640 (with NC SW 34059x-11 and later) and TNC 620 (with NC SW 81760x-08 and 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 Programming Features	**SIK** Option 20 ID 628253-01
TNC7	Standard function
TNC7 basic	Standard function
TNC 640	Standard function in NC SW 34059x-04 and later
TNC 620	NC SW 81760x-01 and later
TNC 320	Standard function in NC SW 771851-01 and later
Installation by the user	
For more information, see the TNC brochures, and visit www.klartext-portal.com	

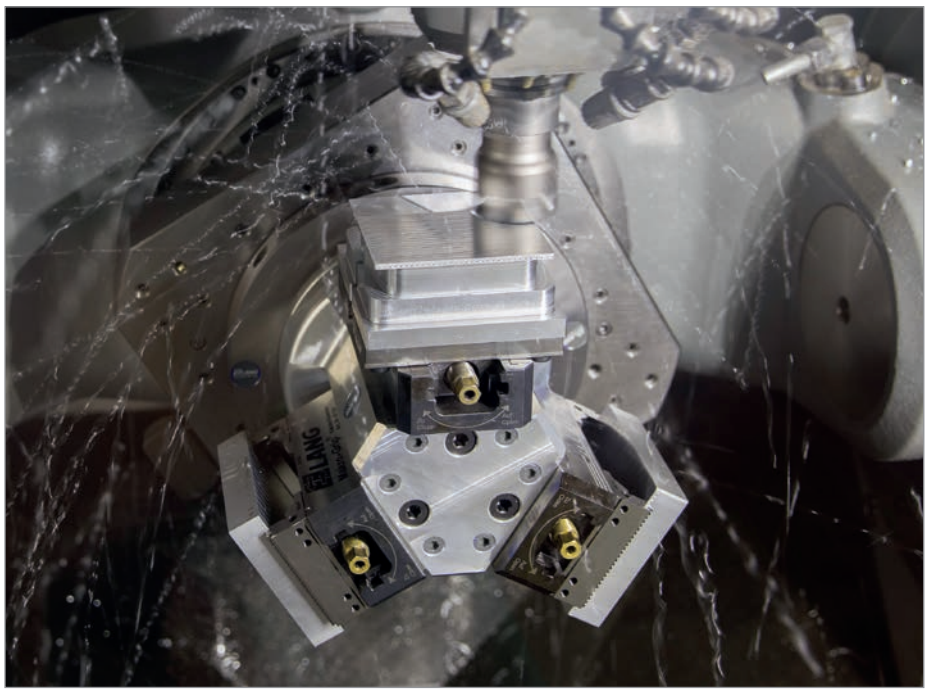
Programming and operation
Pallet management

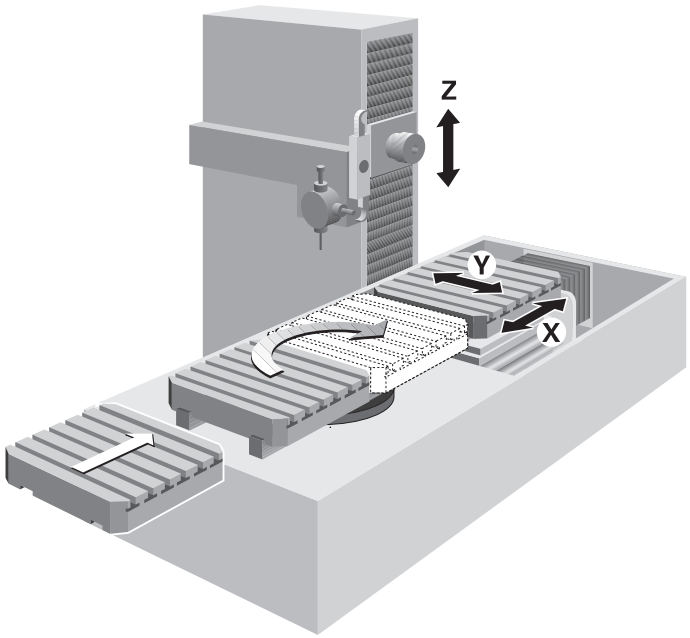
For any sequence of different workpieces loaded on pallets, the TNC is able to identify the proper machining program and datum shift.

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 user. 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 control can be either workpiece- or tool-oriented (with NC SW 34059x-08 and later on the TNC 640; with NC SW 81760x-05 and later on the TNC 620).





Pallet Management	**SIK** Option 22 ID 628255-01
TNC7	Standard function
TNC7 basic	Standard function
TNC 640	Standard function
TNC 620	Standard function
TNC 320	–

Installation by the machine manufacturer

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

Programming and operation

Batch Process Manager for organizing and fine tuning job sequences on the machine

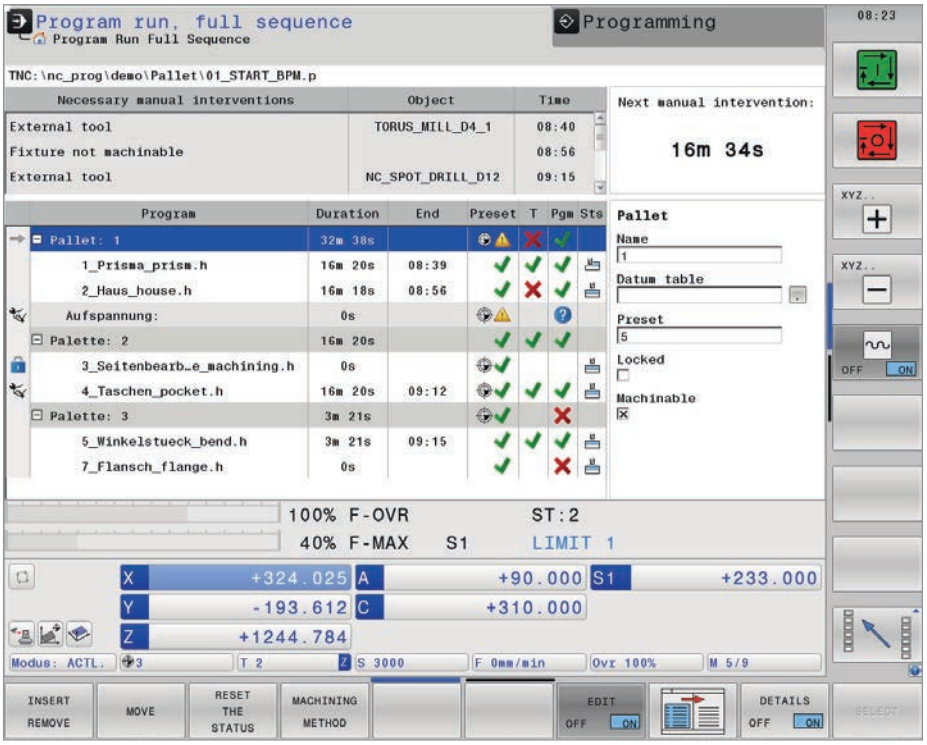
Batch Process Manager is a powerful function for pallet machining and series production. With its intuitive user interface, you can plan your production process and receive important information about upcoming machining operations.

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. Batch Process Manager thereby ensures 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 the **Pallet Management** software option. On controls with NC software versions up to 34059x-08 or 81760x-05, the **Extended Tool Management** software option must be enabled as well.



Batch Process Mngr.	**SIK**	**SIK2**
	Option 154	2-05-1
	ID 1219521-01	ID 1395860-01
TNC7	NC SW 81762x-16 and later	
TNC7 basic	NC SW 81762x-18 and later	
TNC 640	NC SW 34059x-08 and later	
TNC 620	NC SW 81760x-05 and later	
TNC 320	–	

Installation by the machine manufacturer

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

Programming and operation
CAD import for contours and machining positions from DXF files

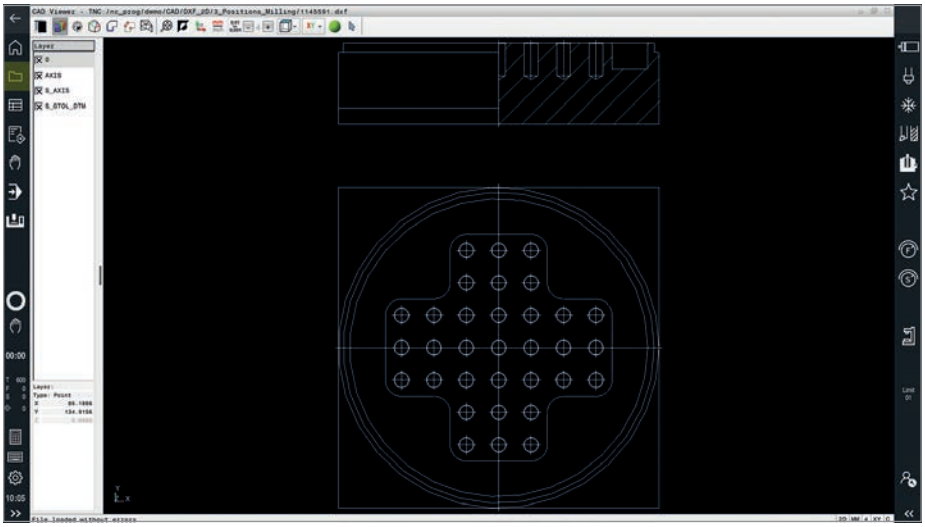
Why program contours when you already have the drawing as a DXF file? You can open DXF files directly on the TNC in order to extract contours or machining positions from them. This not only saves time otherwise spent on programming and testing but also provides assurance that the finished contour precisely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

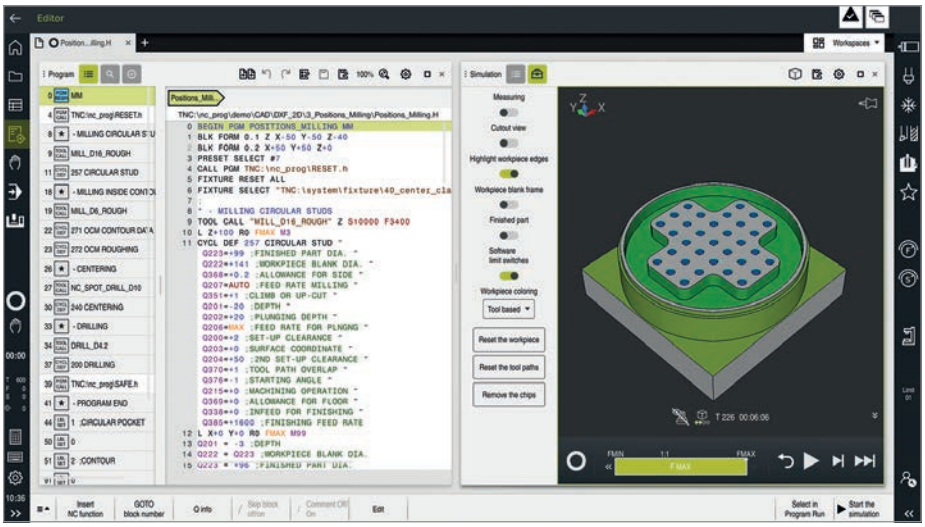
DXF files usually contain multiple layers used by the design engineer to organize the drawing. In order to minimize unneeded information on your screen during contour selection, you can hide all of the inessential layers in the DXF file with just a gesture. This feature requires an operating panel with 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 assists 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. As soon as you select a second element, the TNC detects your desired direction of machining, and starts automatic contour detection. 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 also 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** software option, you can also import contours and machining positions from 3D models.





CAD Import	**SIK**	**SIK2**
	Option 42	1-03-1
	ID 526450-01	ID 1395847-01
TNC7	NC SW 81762x-16 and later	
TNC7 basic	NC SW 81762x-18 and later	
TNC 640	NC SW 34059x-02 and later	
TNC 620	NC SW 73498x-02/81760x-01 and later	
TNC 320	NC SW 771851-01 and later	
Installation by the user		
For more information, see the TNC brochures, and visit www.klartext-portal.com		

Programming and operation
CAD import for contours from 3D models

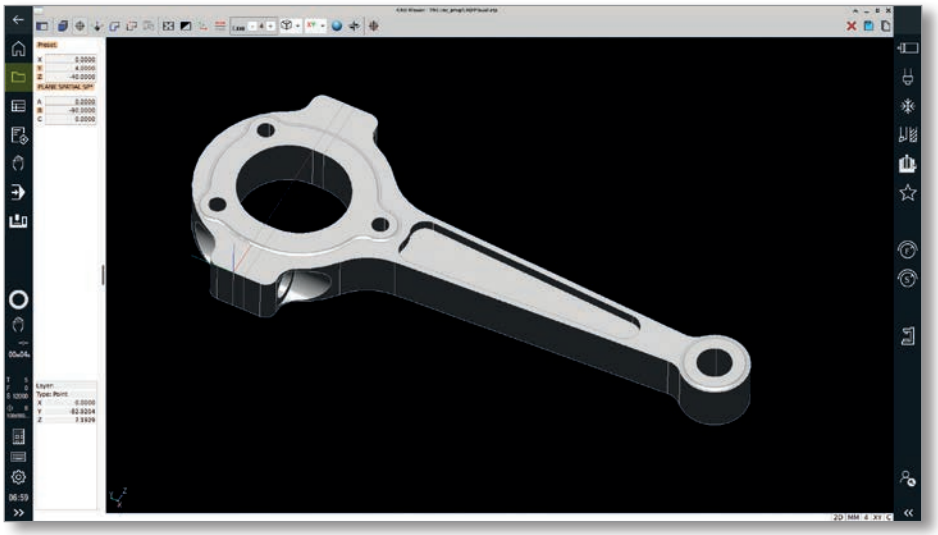
CAD Viewer enables the display of 2D and 3D models right on the TNC (e.g., DXF, Step, STL, or IGES files). Using the CAD Import software option, you can easily incorporate contours and machining positions from these CAD files directly into your Klartext program. You can thereby reduce the amount of programming 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 with a 3D basic rotation of the 3D model and set a datum with the appropriate 3D rotation in the desired working plane.

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

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

The CAD Import software 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.



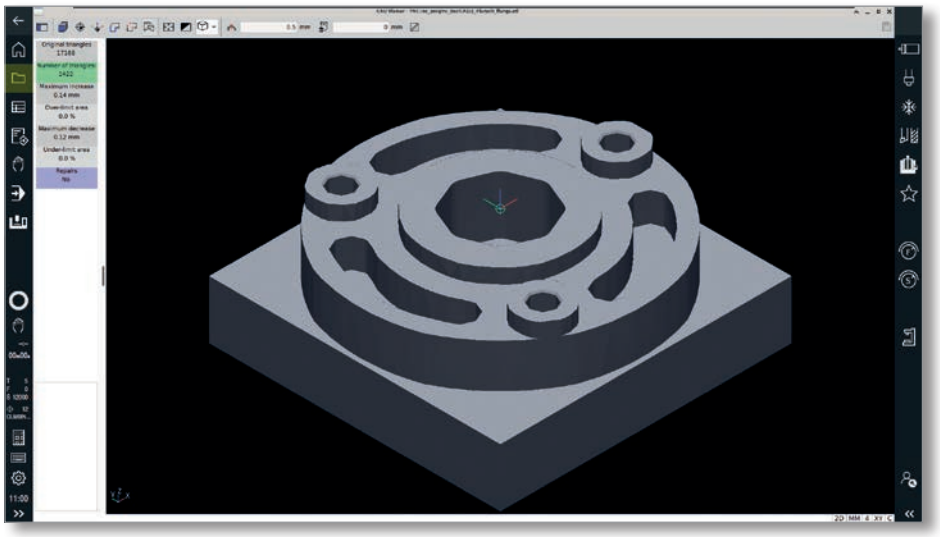
CAD Import	**SIK**	**SIK2**
	Option 42	1-03-1
	ID 526450-01	ID 1395847-01
TNC7	NC SW 81762x-16 and later	
TNC7 basic	NC SW 81762x-18 and later	
TNC 640	NC SW 34059x-08 and later	
TNC 620	NC SW 81760x-05 and later	
TNC 320	NC SW 771851-05 and later	
Installation by the user		
For more information, see the TNC brochures, and visit www.klartext-portal.com		

Programming and operation
Optimization of CAD models

The **CAD Model Optimizer** software option offers the user 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 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. With this software option you can also adapt 3D models of your fixtures before setting them up. The **Collision Monitoring v2** software option makes it possible to monitor fixtures in the machine envelope and thereby prevent collisions.

Using the **CAD Model Optimizer** software option, you can also correct small errors in the workpiece model before setting up the part using the 6D Model Aided Setup function.



CAD Model Optimizer	**SIK**	**SIK2**
	Option 152	1-04-1
	ID 1353918-01	ID 1395849-01
TNC7	NC SW 81762x-16 and later	
TNC7 basic	NC SW 81762x-18 and later	
TNC 640	NC SW 34059x-16 and later	
TNC 620	NC SW 81760x-16 and later	
TNC 320	–	
Installation by the user		
For more information, see the TNC brochures, and visit www.klartext-portal.com		

Programming and operation
Mill-turning functions for milling and turning on the same machine

The control provides 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 head
With a facing head, you can 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 head, the spindle provides the rotational motion while an axis integrated into the facing head laterally extends the 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.



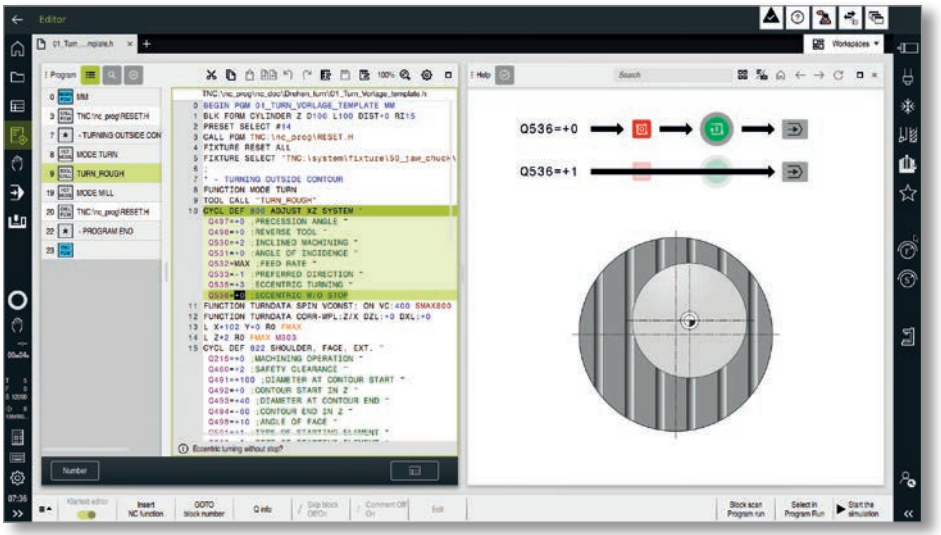
Turning	**SIK**	**SIK2**
	Option 50	4-03-1
	ID 634608-01	ID 1395877-01
TNC7	NC SW 81762x-16 and later	
TNC7 basic	–	
TNC 640	NC SW 34059x-01 and later	
TNC 620	–	
TNC 320	–	

Installation by the machine manufacturer

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

Programming and operation
Mill-turning functions for 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, the **Synchronizing Functions** software option must be enabled.

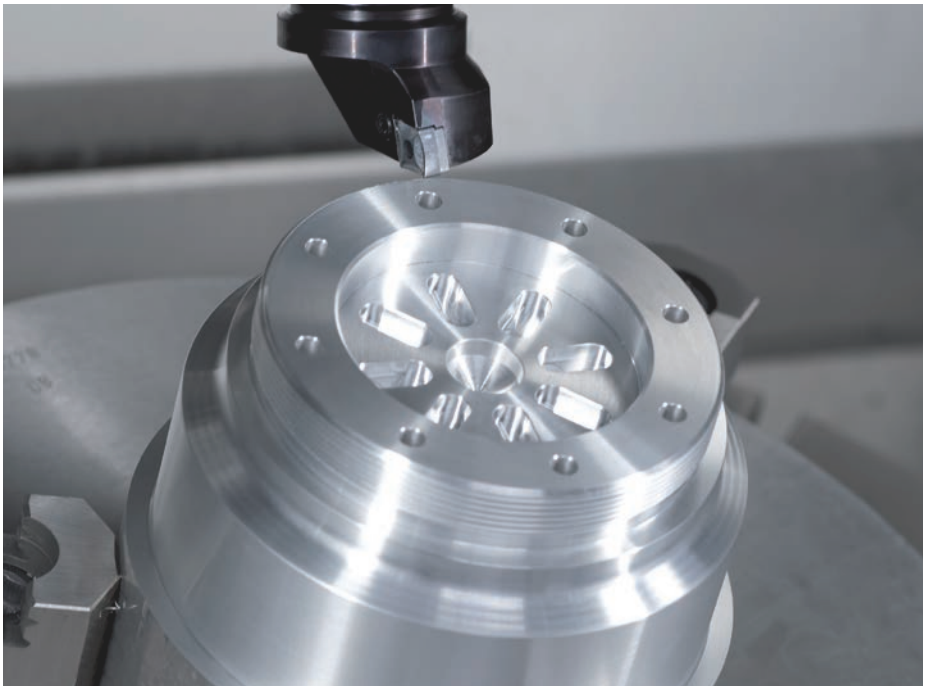
Turning	**SIK**	**SIK2**
	Option 50	4-03-1
	ID 634608-01	ID 1395877-01
TNC7	NC SW 81762x-16 and later	
TNC7 basic	–	
TNC 640	NC SW 34059x-04 and later	
TNC 620	–	
TNC 320	–	
Installation by the machine manufacturer		
For more information, see the TNC brochures, and visit www.klartext-portal.com		

Programming and operation
Mill-turning functions for simultaneous turning

This software 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 software version 11 and 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 and later).



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

* Advanced Function Set Turning additionally requires the **Turning** software option

Turning v2*	**SIK** Option 158 ID 1432155-01	**SIK2** 4-03-2 ID 1395877-02
TNC7 **TNC7 basic** **TNC 640** **TNC 620 / TNC 320**	NC SW 81762x-16 and later – NC SW 34059x-17 and later –	
Installation by the machine manufacturer		
For more information, see the TNC brochures, and visit www.klartext-portal.com		

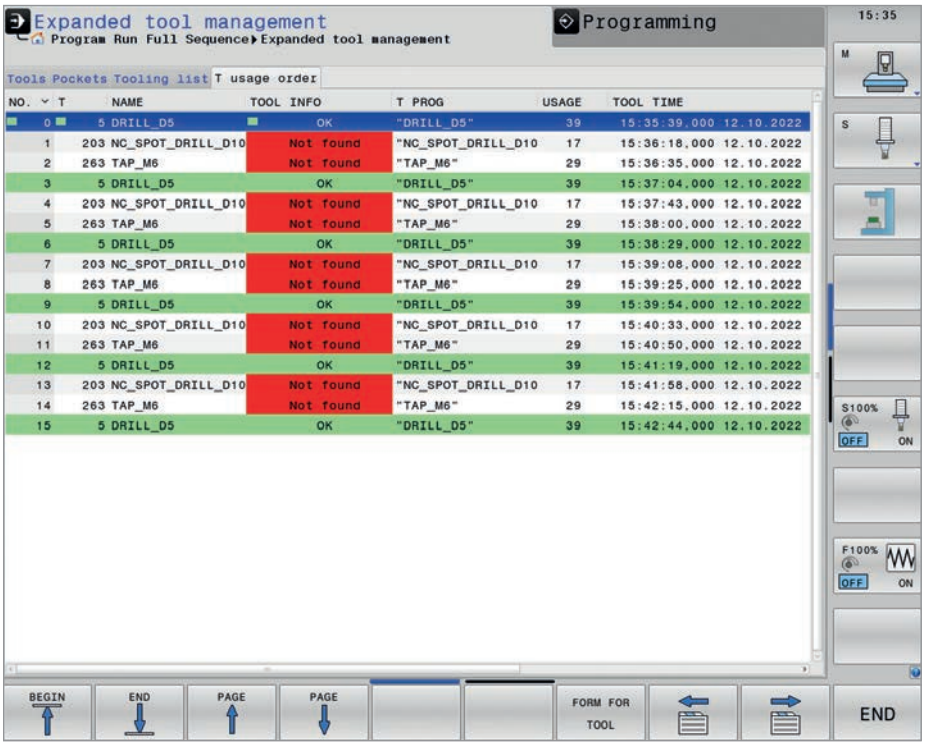
* **Turning v2** contains the **Turning** software option

Programming and operation
Extended tool management

The **Ext. Tool Management** software option expands tool management to include the "Tooling list" and "T usage order" tables. The "Tooling list" table shows the tool requirements of the NC program or pallet to be run. The "Tooling list" table 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.

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



Ext. Tool Management	**SIK**	**SIK2**
	Option 93	2-03-1
	ID 676938-01	ID 1395858-01
TNC7	NC SW 81762x-16 and later	
TNC7 basic	NC SW 81762x-18 and later	
TNC 640	NC SW 34059x-01 and later	
TNC 620	NC SW 81760x-03 and later	
TNC 320	NC SW 771851-02 and later	
Installation by the machine manufacturer		
For more information, see the TNC brochures, and visit www.klartext-portal.com		

Programming and operation
Interpolating spindle

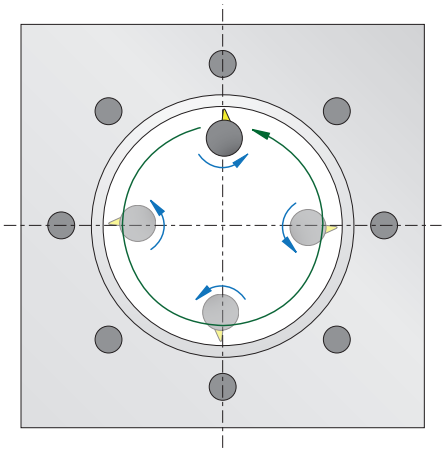
The intelligent coupling of the tool spindle with the linear axes allows the control to execute precise and efficient machining processes with the following functions:
- Interpolation turning
- Correction of turning tools
- Turning tool table
- Shaping*

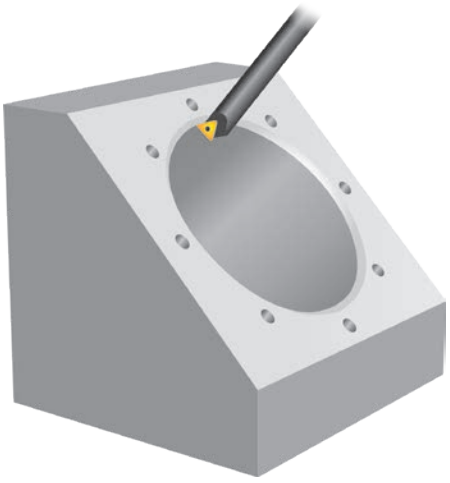
The interpolating spindle enables you to execute a planing or turning operation also on machines without rotary table.

Interpolation turning (option)
Although you can create ring slots, recesses, tapers, or any turning contour using conventional turning operations, you can also take advantage of interpolation turning. In interpolation turning, the tool executes a circular movement using the linear axes. For outside turning, the cutting edge is oriented towards the circle center, and for internal turning, away from the center. By varying the circle radius and the axial position, any rotationally symmetric body can be produced, even in a tilted working plane. This cycle can be used only for finishing work and does not permit roughing operations with multiple passes.

Shaping*
When creating premium-quality surfaces with special requirements, such as sealing surfaces, a tool that is continuously engaged is usually the best choice. Shaping is a further development of planing, a kinematically identical classic production method. It uses tools that are very similar to turning tools. This makes it possible for the operator to very easily implement this technology in any working plane. The tool spindle does not even need to be programmed separately, but is rather positioned automatically by the TNC, which always keeps the cutting edge perpendicular to the contour. This significantly reduces the amount of programming necessary. All axes are servo-controlled, based on the defined feed rate, in order to produce workpiece surfaces with a consistently high quality.

* Only available for TNC7 SIK2 with software NC 81762x-19 or later





Adv. Spindle Interpol.	**SIK**	**SIK2**
	Option 96	7-04-1
	ID 751653-01	ID 1395892-01
TNC7	NC SW 81762x-16 and later	
TNC7 basic	–	
TNC 640	NC SW 34059x-05 and later	
TNC 620	–	
TNC 320	–	

Installation by the machine manufacturer

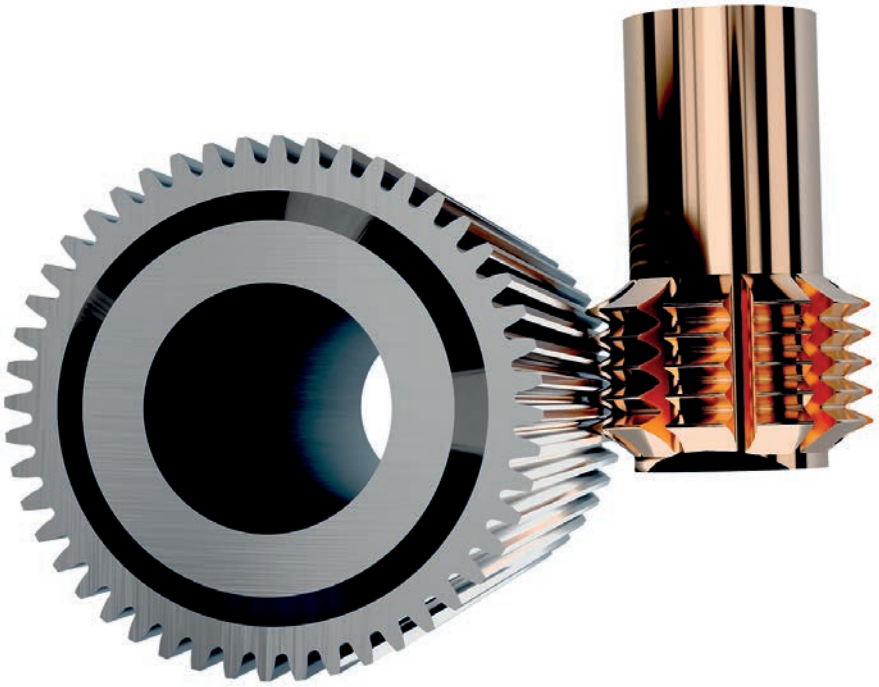
For more information, see the TNC brochures, and visit www.klartext-portal.com

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.

When using the **Turning** or **Turning v2** software option in combination with **Spindle Synchronism**, the control offers Cycle 880 GEAR HOBBING, 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 and 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	**SIK** Option 131 ID 806270-01	**SIK2** 7-02-1 ID 1395890-01
TNC7 **TNC7 basic** **TNC 640** **TNC 620** **TNC 320**	NC SW 81762x-16 and later – NC SW 34059x-05 and later – –	
Installation by the machine manufacturer		
For more information, see the TNC brochures		

Programming and operation
Grinding and dressing functions

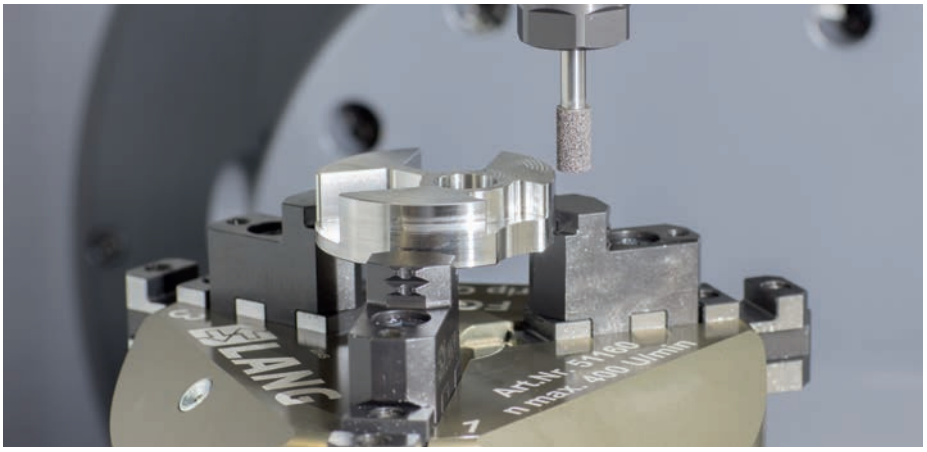
With the TNC7 and TNC 640, you can even perform grinding operations on your machine. Cycles for jig grinding, cylindrical grinding*, and dressing allow you to program the respective function with great convenience. The TNC7 and TNC 640 can also superimpose a tool-axis reciprocating stroke onto the programmed movements. Optimized tool management assists the user 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.

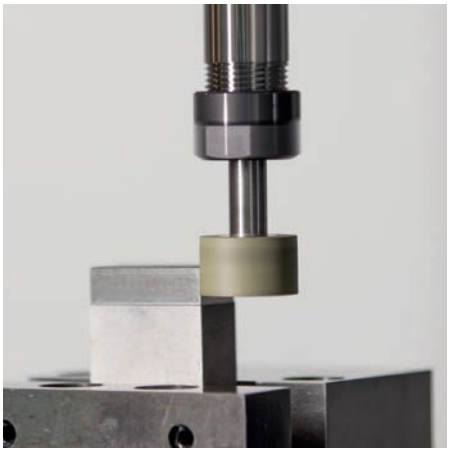
Jig grinding
Jig grinding is the grinding of a 2D contour. On a milling machine, jig grinding is primarily used for finishing holes or pre-machined contours using a grinding tool. The grinding contour can be programmed in the defined grinding cycles and then machined.

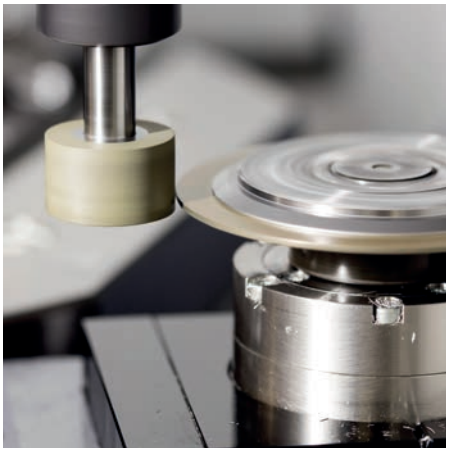
Cylindrical grinding*
Cylindrical grinding is a machining process for the high-precision machining of cylindrical or conical surfaces, in which the workpiece rotates and is machined by a rotating grinding wheel. This function enables reliable high levels of accuracy and a high surface quality for your workpieces. Similar to integrated turning, setup processes that cost valuable time are eliminated. The TNC7 offers comprehensive standard cycles for defining multiple machining steps with reciprocating and feed strokes on a contour element. You can also use it to manage the complex process of approaching and retracting, including tilting axis movements.

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	SIK	SIK2
	Option 156	4-04-1
	ID 1237232-01	ID 1395878-01
TNC7	NC SW 81762x-16 and later	
TNC7 basic	–	
TNC 640	NC SW 34059x-10 and later	
TNC 620	–	
TNC 320	–	

Installation by the machine manufacturer

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

* Only available for TNC7 with software 81762x-19 or later.

Programming and operation
Creating gears in a single setup

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

With the **Gear Cutting** software option, however, you can completely machine spur and helical gears in a single setup using skiving or hobbing operations. If you wish to use this software option in turning mode, then the **Turning** or **Turning v2** software option 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. You can also, for example, use an individual tooth flank line to produce crowning of the tooth flank (starting with NC software 18).

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 software 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	**SIK** Option 157 ID 1237235-01	**SIK2** 4-05-1 ID 1395879-01
TNC7	NC SW 81762x-16 and later	
TNC7 basic	–	
TNC 640	NC SW 34059x-09 and 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 for easy calibration measurements 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** software option 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.

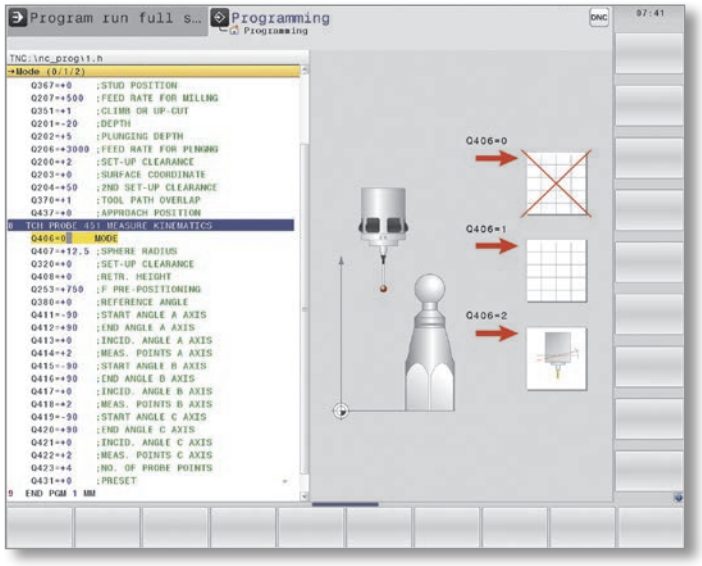
Of course, a detailed log file is also available containing the actual measured values, the measured and optimized dispersion (for the static tilting accuracy), and the actual compensation values. Together with the **KinematicsComp** software option, the **KinematicsOpt** software option permits compensation 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	**SIK**	**SIK2**
	Option 48	2-01-1
	ID 630916-01	ID 1395856-01
TNC7	NC SW 81762x-16 and later	
TNC7 basic	NC SW 81762x-18 and later	
TNC 640	NC SW 34059x-01 and later	
TNC 620	NC SW 34056x-03/73498x-01/81760x-01 and later	
TNC 320	–	

Installation by the machine manufacturer

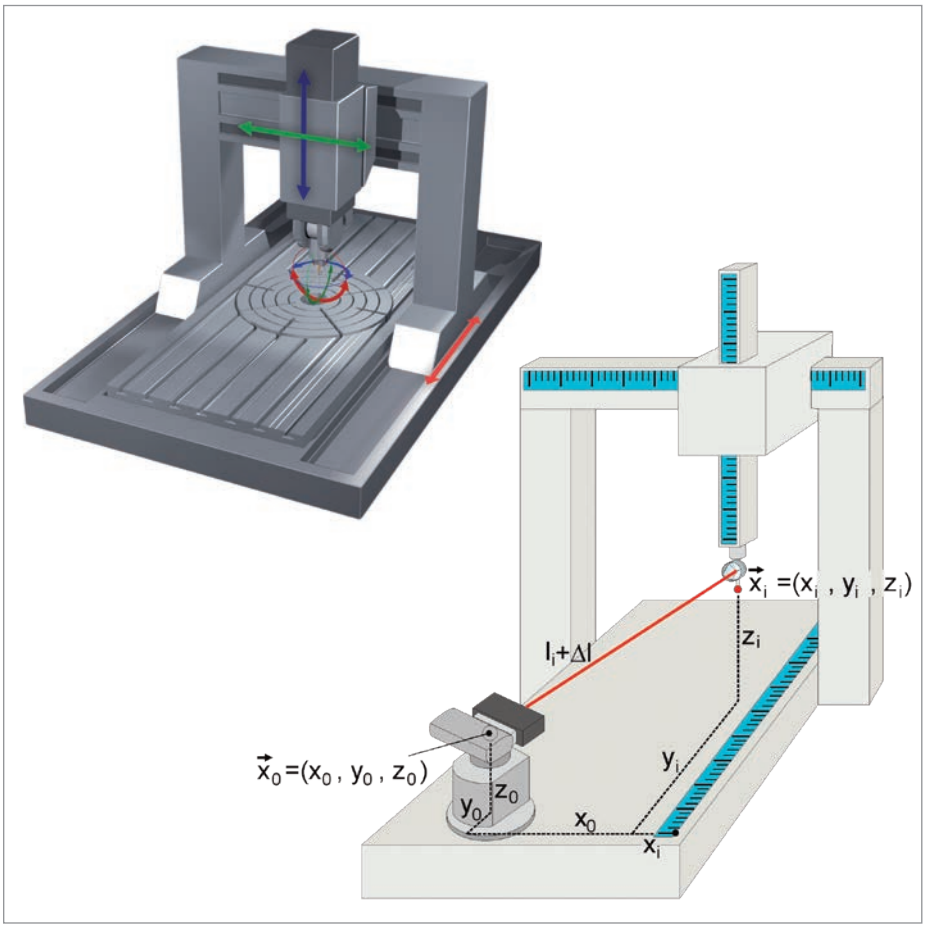
For more information, see the TNC brochures, and visit www.klartext-portal.com

Machine accuracy
KinematicsComp for correcting 3D spatial errors

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 to be neglected is thermal expansion, which can cause highly complex changes in the geometry of machine components.

The **KinematicsComp** software option gives machine manufacturers a way to significantly improve a 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. The **KinematcsOpt** software option uses the **KinematicsComp** software option to compensate for the misalignment of rotary axes and for the grid compensation of rotary swivel heads.



Determining geometric deviations with a laser-based coordinate measuring device (Source: PTB notification 117)

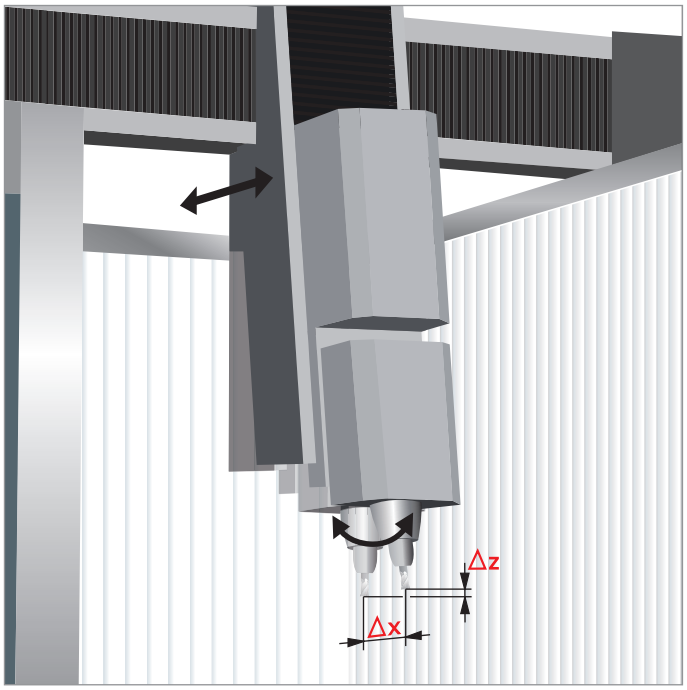
KinematicsComp	**SIK**	**SIK2**
	Option 52	2-04-1
	ID 661879-01	ID 1395859-01
TNC7	NC SW 81762x-16 and later	
TNC7 basic	–	
TNC 640	NC SW 34059x-05 and later	
TNC 620	–	
TNC 320	–	
Installation by the machine manufacturer		
For more information, see the TNC brochures, and visit www.klartext-portal.com		

Machine accuracy
Correcting position errors due to axis coupling (CTC)

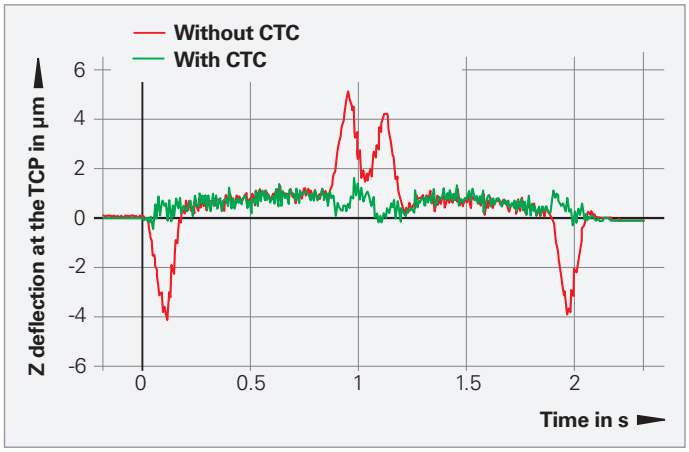
During dynamic acceleration processes, a machine tool's structure undergoes forces capable of deforming machine components, resulting in errors at the tool center point (TCP). The dynamic acceleration of an axis causes not only axial deformations but also deformations that are lateral to the direction 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 **CrossTalk Compensation** software 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 **CrossTalk Compensation** software 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; servo lag within the tolerance band

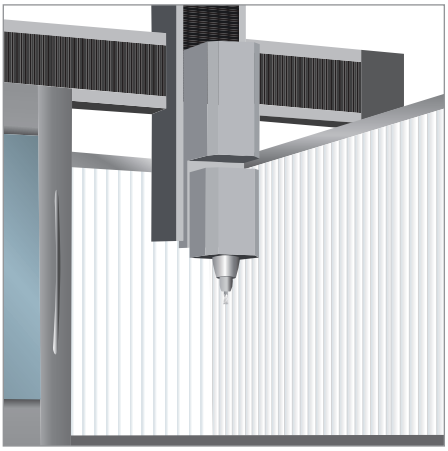
dynamic + precision

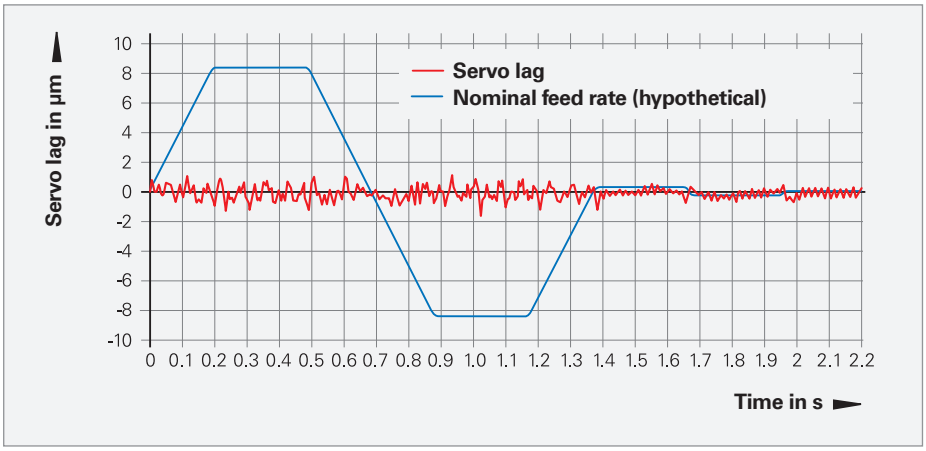
CrossTalk Comp.	SIK	SIK2
	Option 141	2-20-1
	ID 800542-01	ID 1395862-01
TNC7	NC SW 81762x-16 and later	
TNC7 basic	NC SW 81762x-18 and later	
TNC 640	NC SW 34059x-02 and later	
TNC 620	NC SW 34056x-04/73498x-02/81760x-01 and later	
TNC 320	–	
Installation by the machine manufacturer		
For more information, see the TNC brochures and the *Dynamic Precision* Technical Information document, and visit www.klartext-portal.com		

Machine accuracy
Position-dependent adaptation of control parameters (PAC)

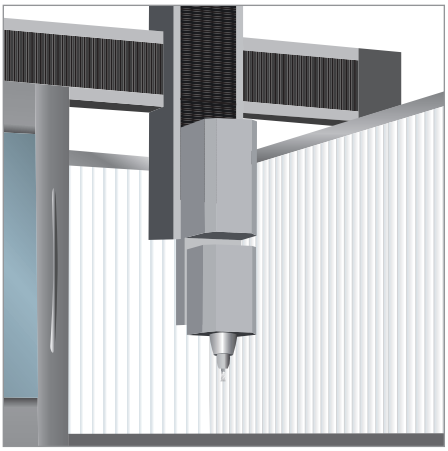
Depending on the positions of the axes in the working space, a machine's kinematics give rise to variable dynamic behavior that can adversely affect servo-control stability.

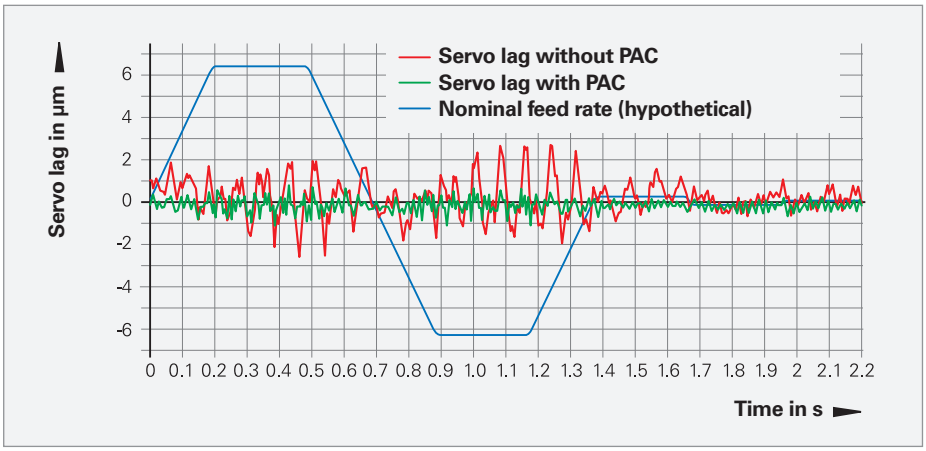
To take full advantage of the machine's dynamic performance, the **Position Adaptive Control** software option enables changes to machine parameters based on position, thus permitting assignment of the respective optimal loop gain to defined interpolation points. Additional position-dependent filter parameters can be defined in order to further increase control loop stability.





Servo control optimized for Z = 0; servo lag within the tolerance band (±1 µm)





Servo control at Z = –500
- Without PAC: clearly visible oscillations and servo lag (±3 µm) outside of the tolerance band
- With active PAC: servo lag within the tolerance band (±1 µm)

dynamic + precision

Position Adapt. Contr.	**SIK**	**SIK2**
	Option 142	2-21-1
	ID 800544-01	ID 1395863-01
TNC7	NC SW 81762x-16 and later	
TNC7 basic	NC SW 81762x-18 and later	
TNC 640	NC SW 34059x-02 and later	
TNC 620	NC SW 34056x-04/73498x-02/81760x-01 and later	
TNC 320	–	

Installation by the machine manufacturer

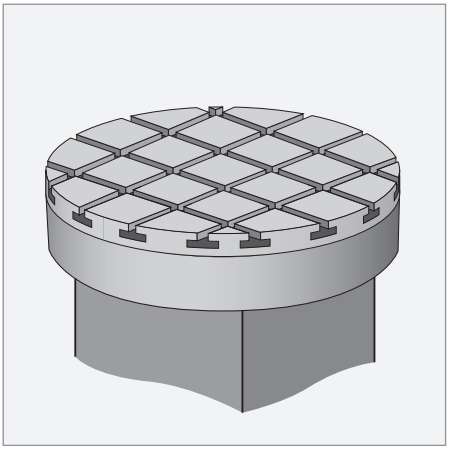
For more information, see the TNC brochures and the *Dynamic Precision* Technical Information document, and visit www.klartext-portal.com

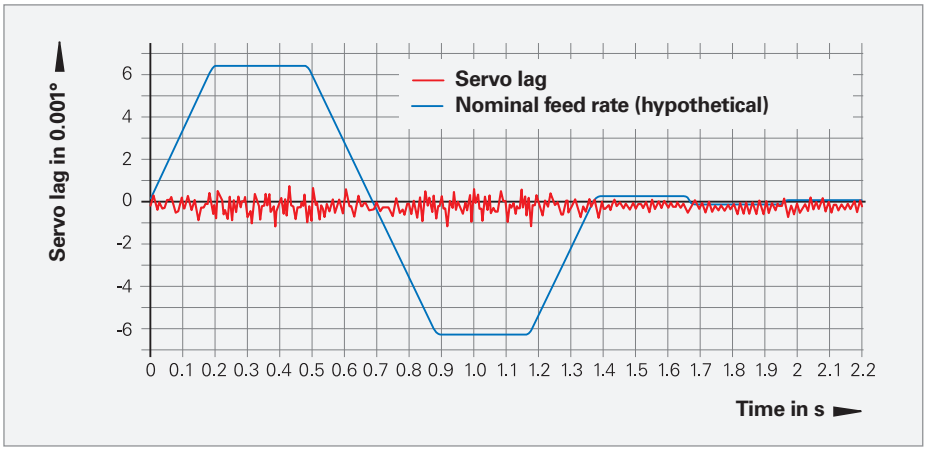
Machine accuracy
Load-dependent adaptation of control parameters (LAC)

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

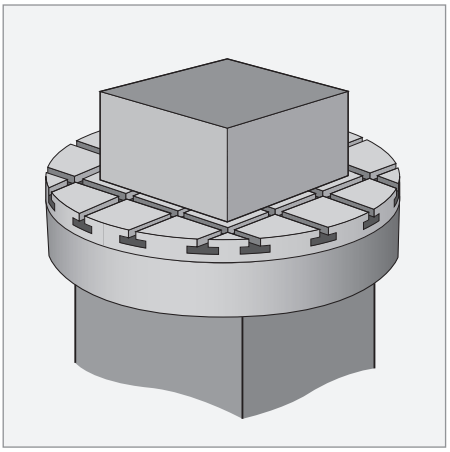
The **Load Adaptive Control** software option enables the control to automatically ascertain the current workpiece mass or mass moment of inertia and the acting frictional forces. In order to ensure an optimum response to changes in machine behavior at different loads, adaptive feedforward controls can be activated, and the control loop gain can be changed based on the load.

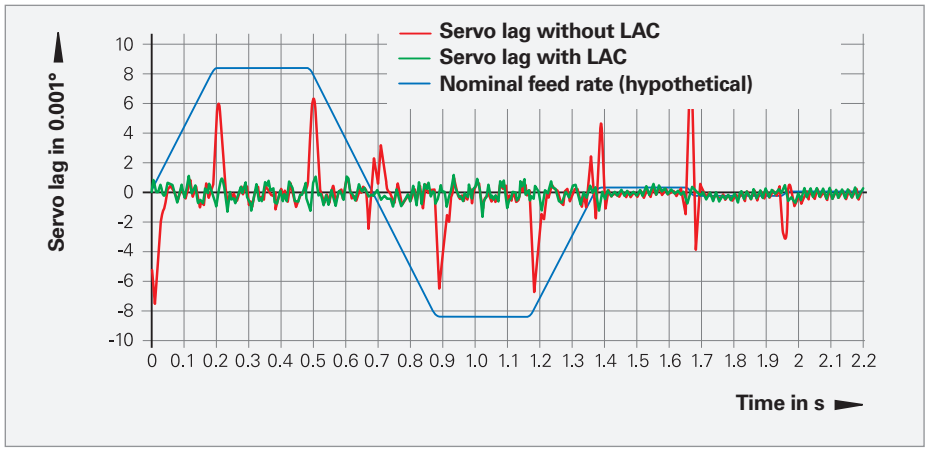
In order to allow rapid adjustment in response to sudden changes in the load (e.g., from loading and unloading the workpiece), Cycle 239 is provided for ascertaining the current load status.





Optimal feedforward control for rotary tables without additional load and with servo lag within the tolerance band (±0.001°)





With change in load
- Without LAC: with unchanged feedforward control, the servo lag (±0.008°) is outside of the tolerance band
- With LAC: with feed-forward control and active LAC, the servo lag is within the tolerance band (±0.001°)

dynamic + precision

Load Adapt. Contr.	**SIK**	**SIK2**
	Option 143	2-22-1
	ID 800545-01	ID 1395864-01
TNC7	NC SW 81762x-16 and later	
TNC7 basic	NC SW 81762x-18 and later	
TNC 640	NC SW 34059x-02 and later	
TNC 620	NC SW 34056x-04/73498x-02/81760x-01 and later	
TNC 320	–	

Installation by the machine manufacturer

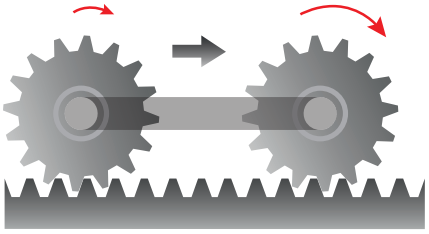
For more information, see the TNC brochures and the *Dynamic Precision* Technical Information document, and visit www.klartext-portal.com

Machine accuracy
Motion-dependent adaptation of control parameters (MAC)

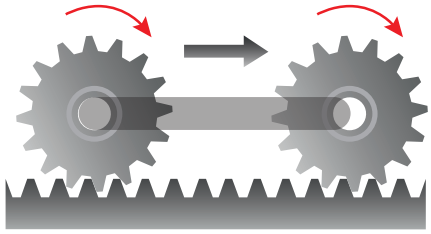
In addition to the position-dependent adaptation of control parameters through the **Position Adaptive Control** software option, the **Motion Adaptive Control** software option provides a way of changing machine parameters based on other input values such as speed or motor acceleration. This motion-dependent adaptation of the control parameters makes it possible, for example, to realize a velocity-dependent adaptation of the kv factor on motors that exhibit stability changes at different traversing velocities.

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

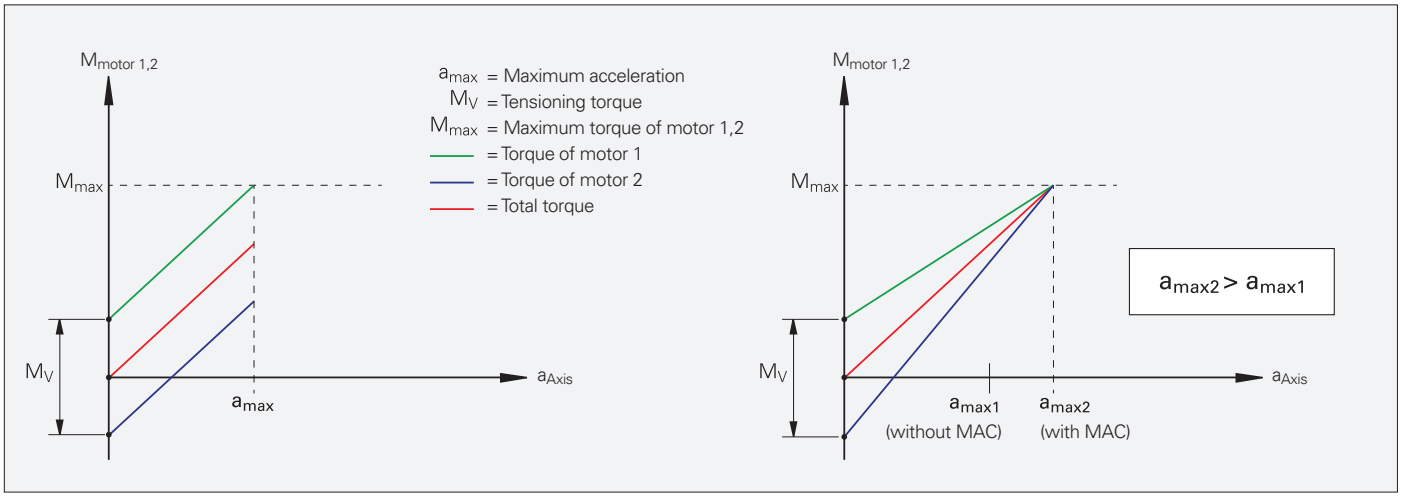
With this software option, this configuration makes it possible to attain a significantly higher maximum acceleration at rapid traverse; for example, through parameter-based reduction of the tensioning torque at increasing acceleration.



Without MAC



With MAC



dynamic + precision

Motion Adapt. Contr.	**SIK**	**SIK2**
	Option 144	2-23-1
	ID 800546-01	ID 1395865-01
TNC7	NC SW 81762x-16 and later	
TNC7 basic	NC SW 81762x-18 and later	
TNC 640	NC SW 34059x-02 and later	
TNC 620	NC SW 34056x-04/73498x-02/81760x-01 and later	
TNC 320	–	
Installation by the machine manufacturer		
For more information, see the TNC brochures and the *Dynamic Precision* Technical Information document, and visit www.klartext-portal.com		

Machine accuracy
Active vibration damping (MVC)

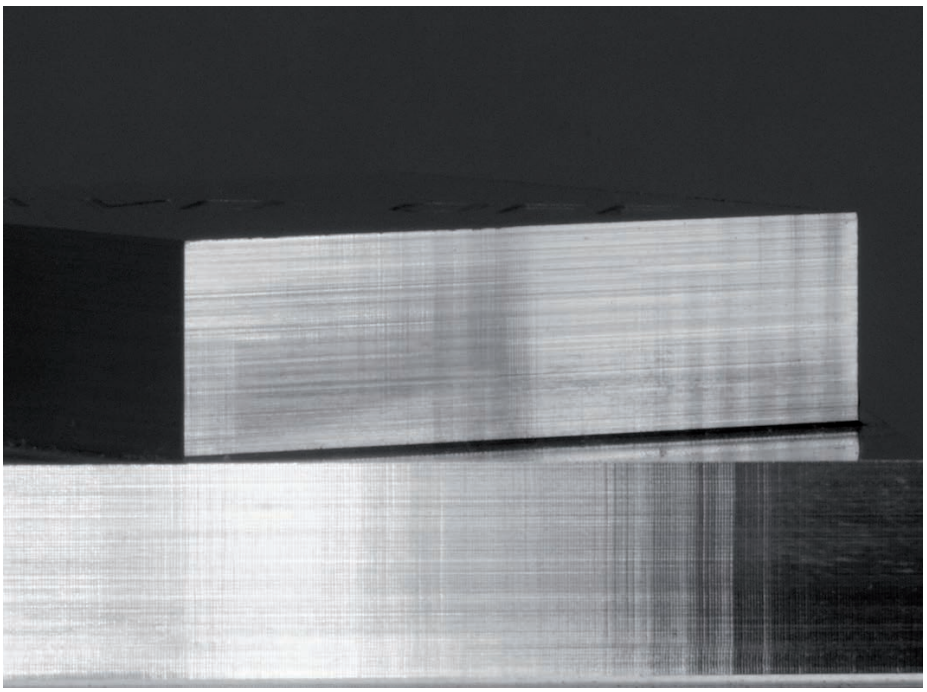
In the presence of low-frequency vibrations on a machine tool, inclined or curved surfaces can often exhibit surface-finish problems in the form of visible shadows or inconsistent contrast. Peaks as small as 1 µm or smaller may be visually noticeable on a workpiece. These disturbances often necessitate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

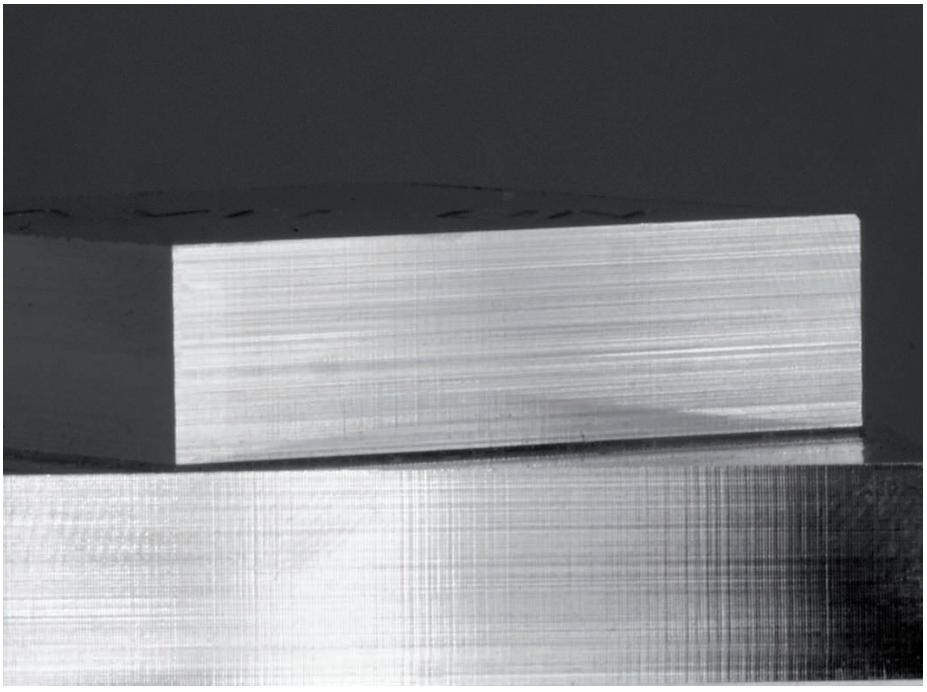
Low-frequency disturbances are often caused by elasticity in the drive train, such as vibrations between the motor and the slides, or machine setup vibrations, in which high accelerations of the machine axes cause disturbances in the machine tool through its fastening elements or connected base.

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

The **Machine Vibration Control** software option Machine Vibration Control (MVC) suppresses low-frequency vibrations via the control's control loop. To this end, this software option provides the Active Vibration Damping (AVD) and Frequency Shaping Control (FSC) functions. The software option works in two ways: it causes a clean workpiece surface because any vibrations that would be visible on the workpiece surface are suppressed. MVC 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 Vibr. Contr.	**SIK**	**SIK2**
	Option 146	2-24-1
	ID 800548-01	ID 1395869-01
TNC7	NC SW 81762x-16 and later	
TNC7 basic	NC SW 81762x-18 and later	
TNC 640	NC SW 34059x-04 and later	
TNC 620	NC SW 34056x-04/73498x-02/81760x-01 and later	
TNC 320	–	

Installation by the machine manufacturer

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

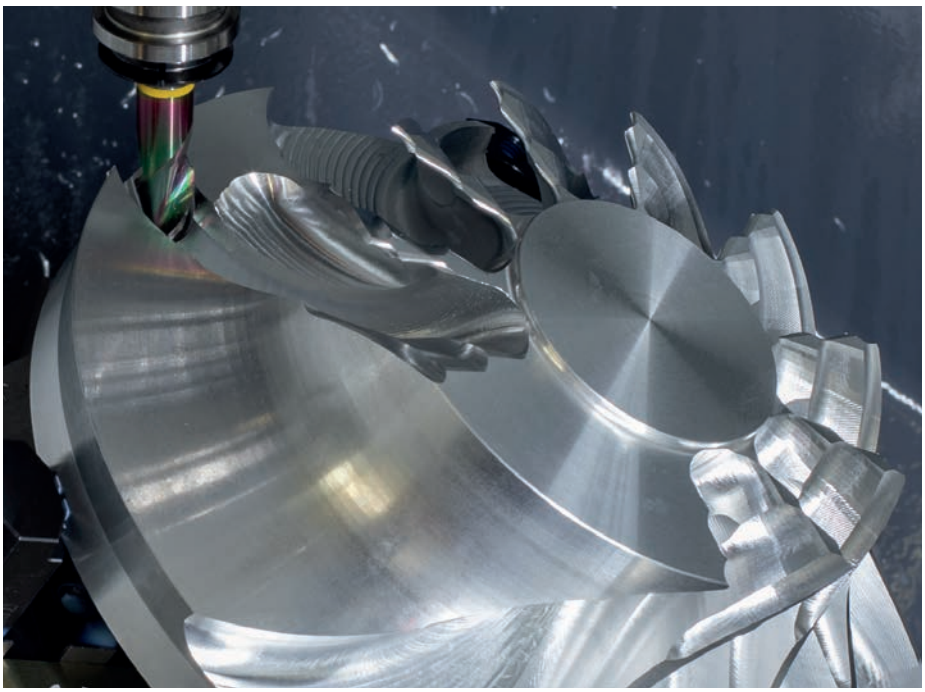
dynamic + precision

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.



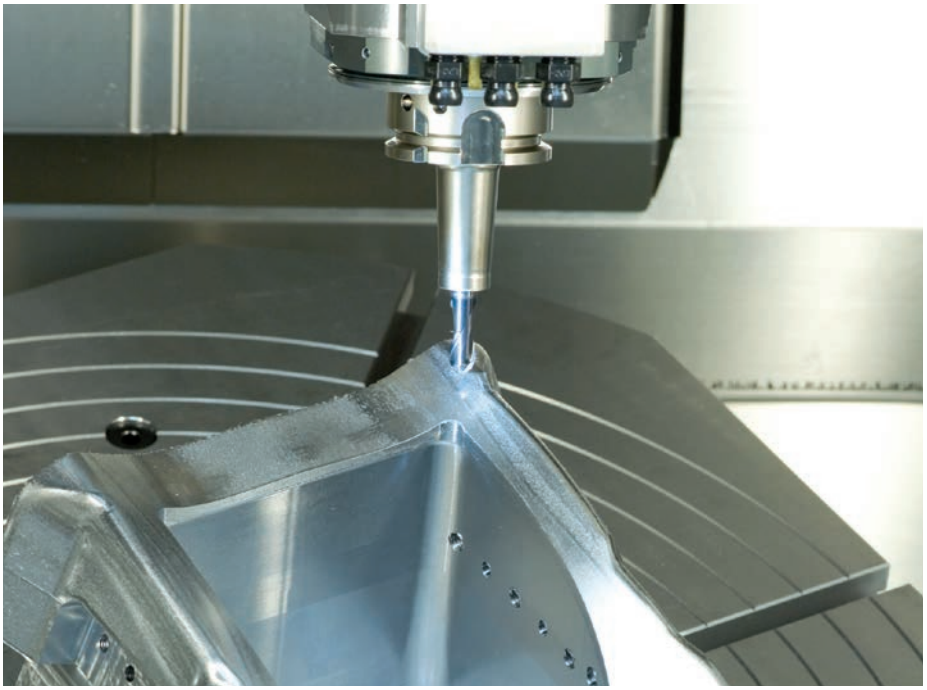


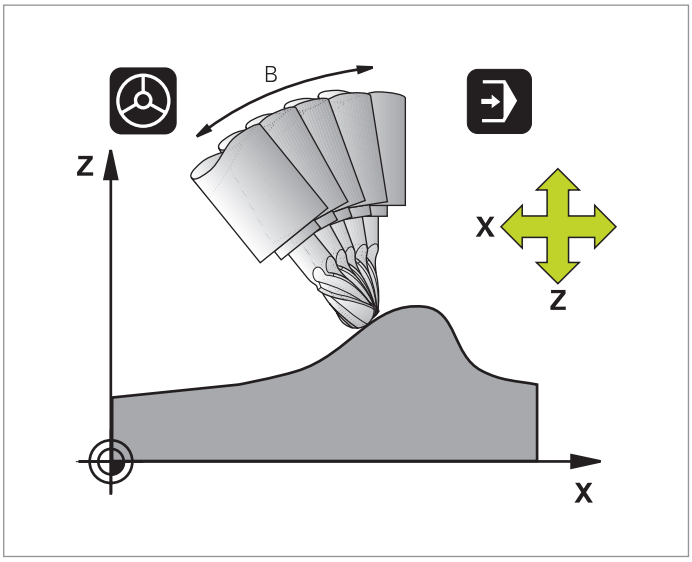
Adv. Function Set 2	**SIK**	**SIK2**
	Option 9	4-01-1
	ID 617921-01	ID 1395875-01
TNC7	NC SW 81762x-16 and later	
TNC7 basic	–	
TNC 640	NC SW 34059x-01 and later	
TNC 620	NC SW 34056x-01/73498x-01/81760x-01 and later	
TNC 320	–	
Installation by the machine manufacturer		
For more information, see the TNC brochures		

Machining functions

Handwheel superimpositioning: superimpose handwheel positioning during program run

The Handwheel Superimpositioning 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 superimpositioning to adjust the offset compensation in linear axes without having to change the NC program.

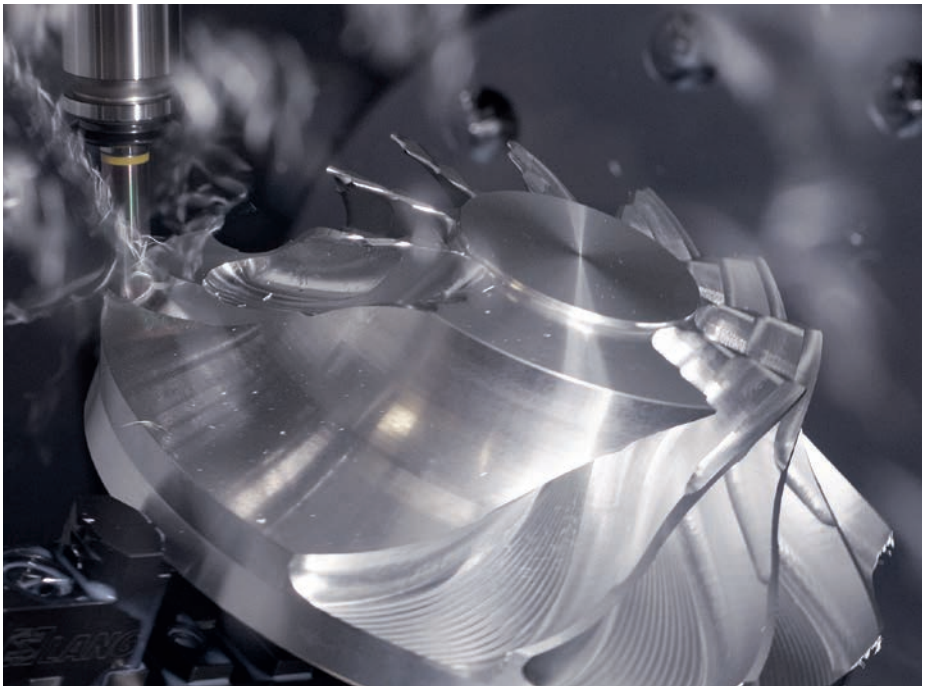


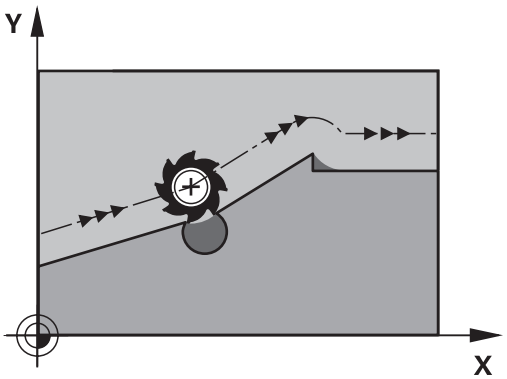


Adv. Function Set 3	**SIK**	**SIK2**
	Option 21	4-02-1
	ID 628254-01	ID 1395876-01
TNC7	Standard function	
TNC7 basic	NC SW 81762x-18 and later	
TNC 640	Standard function	
TNC 620	NC SW 34056x-01/73498x-01/81760x-01 and later	
TNC 320	Standard function	
Installation by the machine manufacturer		
For more information, see the TNC brochures		

Machining functions
Calculating contours with tool-radius offset in advance

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.





Adv. Function Set 3	**SIK**	**SIK2**
	Option 21	4-02-1
	ID 628254-01	ID 1395876-01
TNC7	Standard function	
TNC7 basic	NC SW 81762x-18 and later	
TNC 640	Standard function	
TNC 620	NC SW 34056x-01/73498x-01/81760x-01 and 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. This makes collision monitoring a valuable function that relieves the user and protects the machine from damage.

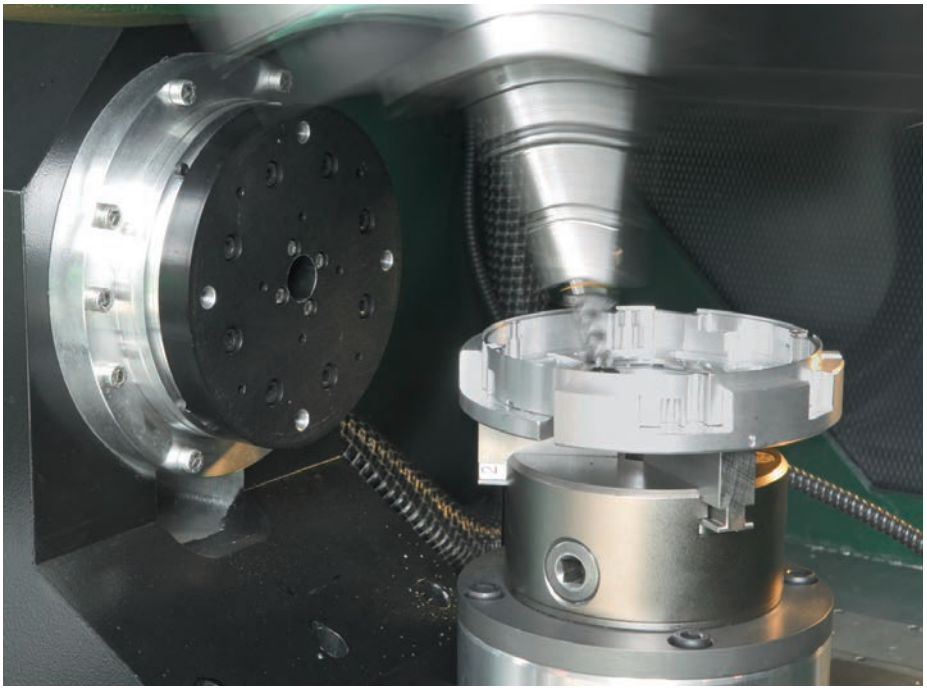
In cases such as these, the user is assisted by the **Collision Monitoring** (DCM) software option of the TNC. The control interrupts the machining process whenever a collision is imminent, thus increasing overall safety. This, in turn, helps prevent machine damage and any resulting costly downtime. Unattended shifts also become more reliable.

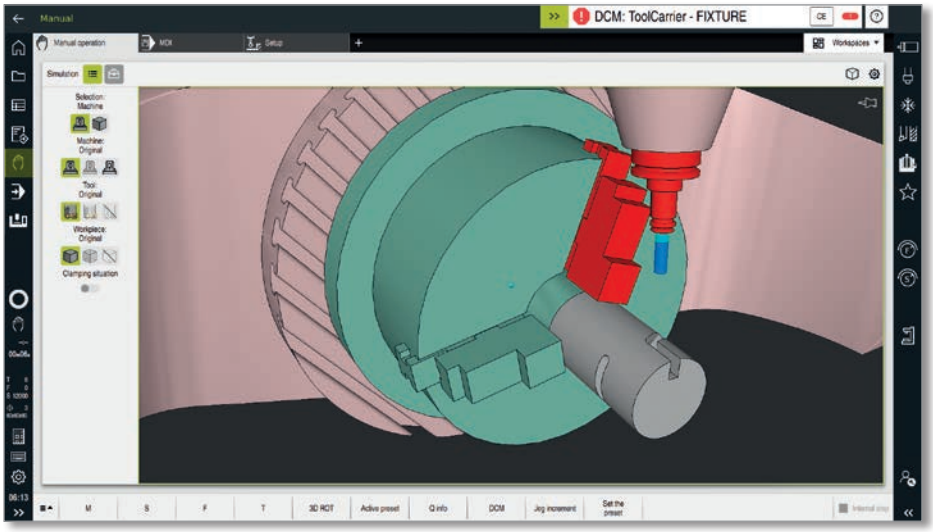
Collision monitoring works not only in automatic mode but is also active in manual mode. If, for example, the user is on a collision course during workpiece setup, the TNC detects this, stops axis movement, and issues an error message. Before actually machining a part, you can also check for collisions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

Of course, the TNC also shows the user 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 TNC7, TNC7 basic, and the TNC 640 (with NC SW 34059x-05 and later) 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 TNC7, TNC7 basic, and the TNC 640 (with NC SW 34059x-11 and later), clamping setups can be incorporated in *.cfg, *m3d, or *.stl formats, including via Klartext commands.





Collision Monitoring	SIK	SIK2
	Option 40	5-03-1
	ID 526452-01	ID 1395882-01
TNC7	NC SW 81762x-16 and later	
TNC7 basic	NC SW 81762x-18 and later	
TNC 640	NC SW 34059x-02 and later	
TNC 620	–	
TNC 320	–	

Installation by the machine manufacturer

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

Machining functions
Dynamic collision monitoring (DCM) version 2

The **Collision Monitoring v 2** software option (DCM v 2) includes all functions of the **Collision Monitoring** software option. The expanded collision monitoring of the control not only prevents tools from colliding with machine components, but also with fixtures. The control 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. The **CAD Model Optimizer** software option allows you to simplify or refine the 3D model of your fixture, thereby enabling you to create a valid STL file for collision monitoring.



With DCM v2 it is possible to machine right up to the fixture. If necessary, you can also reduce the standard 2 mm distance between the tool and the fixture. You easily program the monitored distance in the NC program.

Any tool shapes can be incorporated as 3D models (STL files). This provides the following advantages:
- Monitoring of the tool shaft for collisions
- Ability to protect tools of any shape from collisions with fixtures and machine components
- Realistic material-removal simulation
- Use of tools with any measurement reference point (e.g., backward deburrers)

The collision protection provided by DCM v2 is only as good as the 3D models of the collision objects. For reliable machine operation, the digital data must match the physical reality. The import function of the OPC UA NC Server provides tool presetters and tool databases with functions for importing the 3D models of the tool and tool holder directly into the control. The model validation feature checks the 3D models during importing, thus ensuring maximum reliability of **Collision Monitoring** and the simulation (starting with NC software version 18).

Collision Monitoring v2	**SIK**	**SIK2**
	Option 140	5-03-2
	ID 1432154-01	ID 1395882-02
TNC7	NC SW 81762x-16 and later	
TNC7 basic	NC SW 81762x-18 and 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 in mid-program. A clearly structured form is provided for this. Upon restarting, 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:

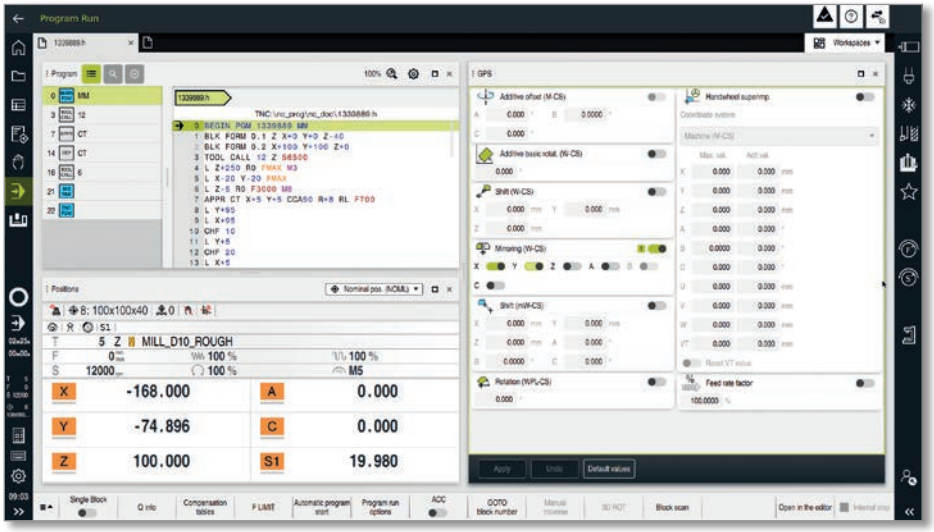
- Additional rotation based on basic rotation or 3D basic rotation
- Shifting the workpiece preset in a single axis
- Mirroring individual axes
- Additional shift of an already shifted workpiece datum
- Rotation about the active tool axis
- Handwheel superimposition
- Feed rate factor



Handwheel superimpositioning is possible in various coordinate systems:

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

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



Global PGM Settings	SIK	SIK2
	Option 44	1-06-1
	ID 576057-01	ID 1395852-01
TNC7	NC SW 81762x-16 and later	
TNC7 basic	–	
TNC 640	NC SW 34059x-08 and later	
TNC 620	–	
TNC 320	–	
Installation by the machine manufacturer		
For more information, see the TNC brochures		

Machining functions
Adaptive feed control (AFC)

The **Adaptive Feed Control** software option (AFC) automatically regulates the feed rate of the TNC, taking into consideration the respective 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
Adaptive Feed Control 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.



dynamic + efficiency

Adaptive Feed Contr.	**SIK**	**SIK2**
	Option 45	2-31-1
	ID 579648-01	ID 1395871-01
TNC7	NC SW 81762x-16 and later	
TNC7 basic	NC SW 81762x-18 and later	
TNC 640	NC SW 34059x-02 and later	
TNC 620 / TNC 320	–	

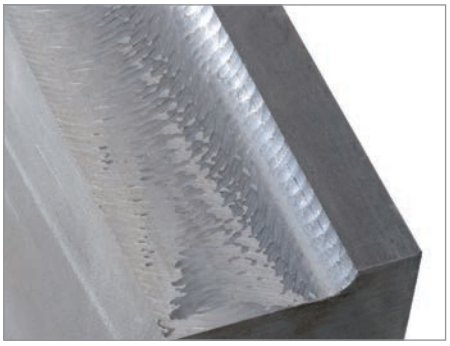
Installation by the machine manufacturer

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

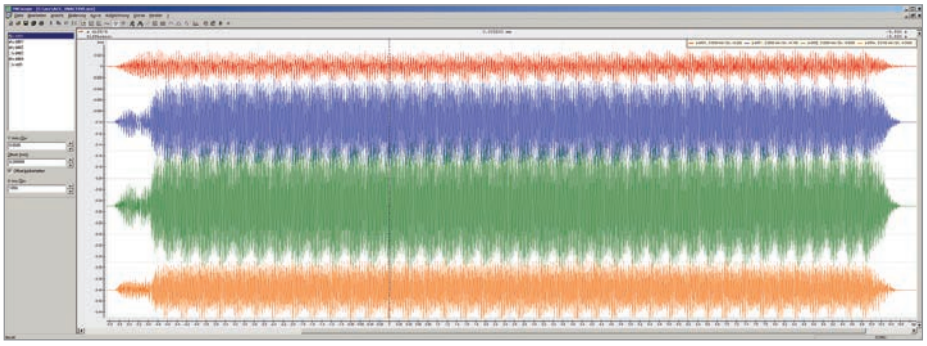
Machining functions
Active chatter control (ACC)

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.

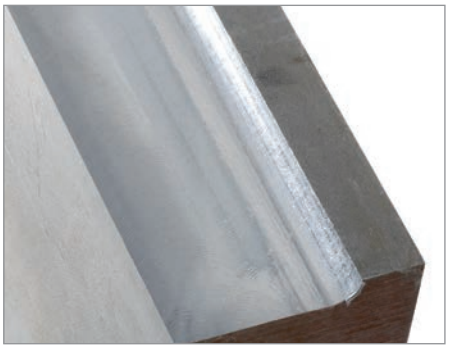
To reduce chatter susceptibility, HEIDENHAIN now offers the highly effective **Active Chatter Control** software option (ACC). This control function is particularly advantageous when used during heavy machining. Thanks to ACC, substantially higher removal rates are possible. Depending on the type of 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.



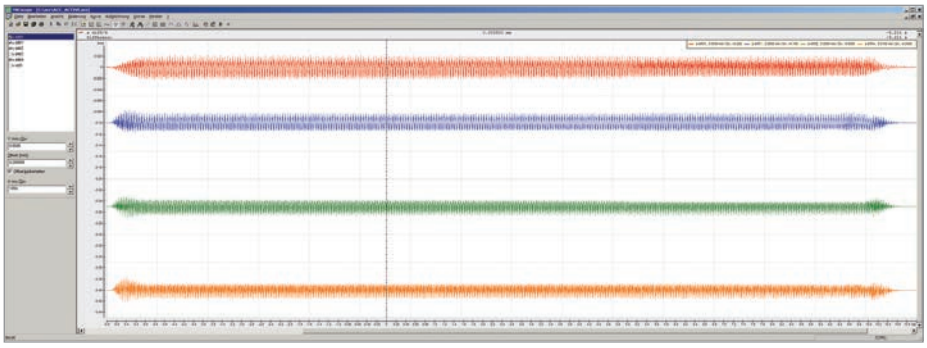
Heavy machining without ACC



Servo lag without ACC



Heavy machining with ACC



Reduced servo lag with ACC

Active Chatter Contr.	**SIK**	**SIK2**
	Option 145	2-30-1
	ID 800547-01	ID 1395870-01
TNC7	NC SW 81762x-16 and later	
TNC7 basic	NC SW 81762x-18 and later	
TNC 640	NC SW 34059x-02 and later	
TNC 620	NC SW 34056x-04/73498x-02/81760x-01 and later	
TNC 320	–	

Installation by the machine manufacturer

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

dynamic + efficiency

Machining functions
Optimized roughing processes with 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 sizable portion of the total machining time.

Process reliability and maximum removal rates require optimal adaptation of the cutting data to the characteristics of the tool and workpiece material. The **Optimized Contour Milling** software option (OCM) gives the user a cutting data calculator that draws on a comprehensive integral materials database. The user 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. Simply program the contour as usual in Klartext, or leverage the convenient **CAD Import** software option. There you can extract contours or machining positions from your 3D model. The control then automatically calculates the complex movements required for maintaining consistent process conditions. During roughing, OCM takes void areas into account, thus allowing the machining time to be considerably shortened (with NC software version 16 or later).

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

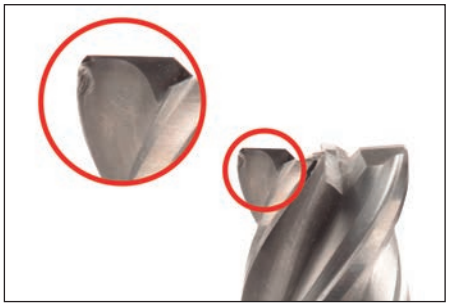
OCM is an effective, reliable, and convenient way to improve your throughput:
- Create programs on the shop floor for any pocket or island
- Significantly higher machining speed
- Considerable reduction in tool wear
- More chips in less time

OCM provides practical cycles for roughing, side-wall finishing, and floor finishing, as well as cycles for standard shapes, chamfering and deburring. (TNC 640: 34059x-11; TNC 620: 81760x-08).

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

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

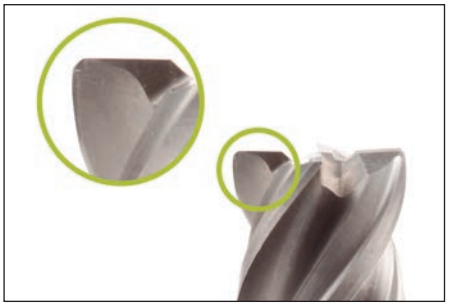
Tool: VHM end mill Ø 10 mm
Workpiece material: 1.4104



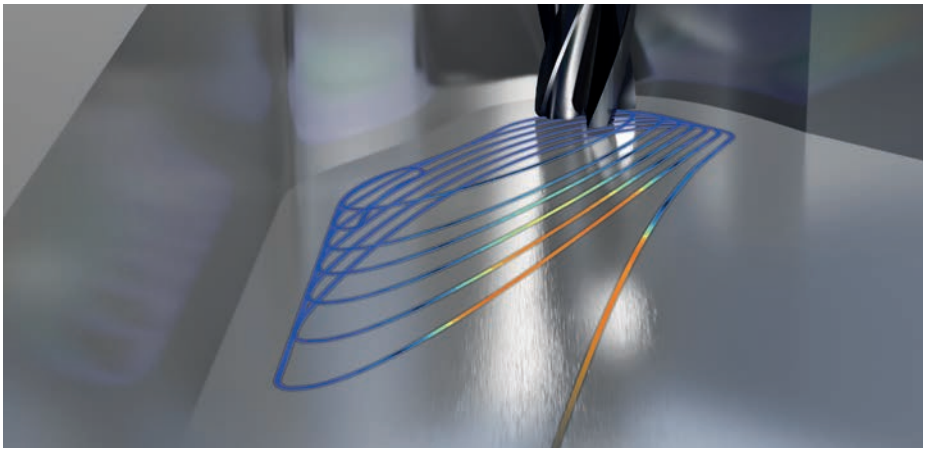
Tool after two parts

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

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



Tool after six parts



Opt. Contour Milling	**SIK**	**SIK2**
	Option 167	1-02-1
	ID 1289547-01	ID 1395833-01
TNC7	NC SW 81762x-16 and later	
TNC7 basic	NC SW 81762x-18 and later	
TNC 640	NC SW 34059x-10 and later	
TNC 620	NC SW 81760x-07 and later	
TNC 320	NC SW 771851-18 and later	
Installation by the machine manufacturer		
For more information, see the TNC brochures, and visit www.klartext-portal.com		

dynamic + efficiency

Machining functions
Monitoring loads on machine components

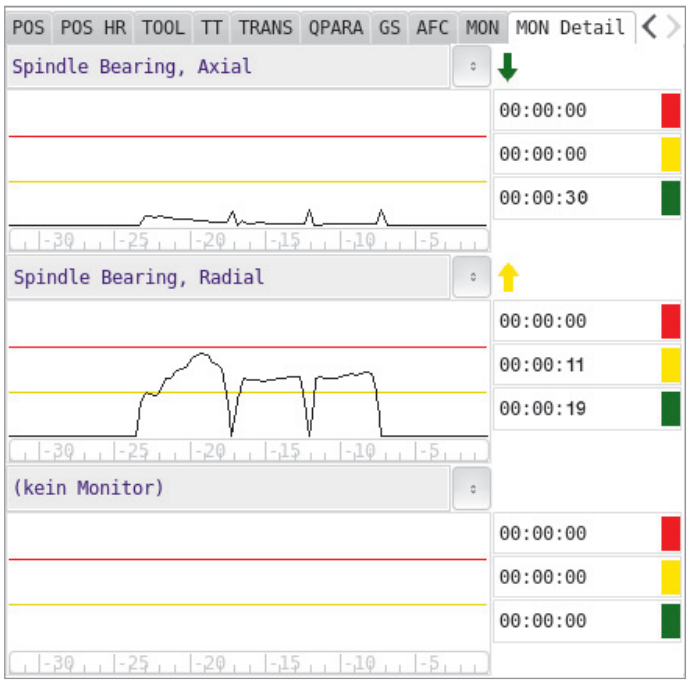
Overload often leads to damaged machine components, and therefore to machine downtime. For example, the spindle bearing is often subject to a very high load in machining strategies that have been optimized for maximum efficiency, and can therefore sustain undetected damage. The **Component Monitoring** software option 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 and TNC7 basic use this software 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 downtime. 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 the software option. 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, TNC7 basic, TNC 640 (with NC SW 34059x-10 or later), and TNC 620 (with NC SW 81760x-07 or later)





Component Monitoring	**SIK**	**SIK2**
	Option 155	5-02-1
	ID 1226833-01	ID 1395881-01
TNC7	NC SW 81762x-16 and later	
TNC7 basic	NC SW 81762x-18 and later	
TNC 640	NC SW 34059x-09 and later	
TNC 620	NC SW 81760x-06 and later	
TNC 320	–	
Installation by the machine manufacturer		
For more information, see the *Information for Machine Manufacturers* brochures		

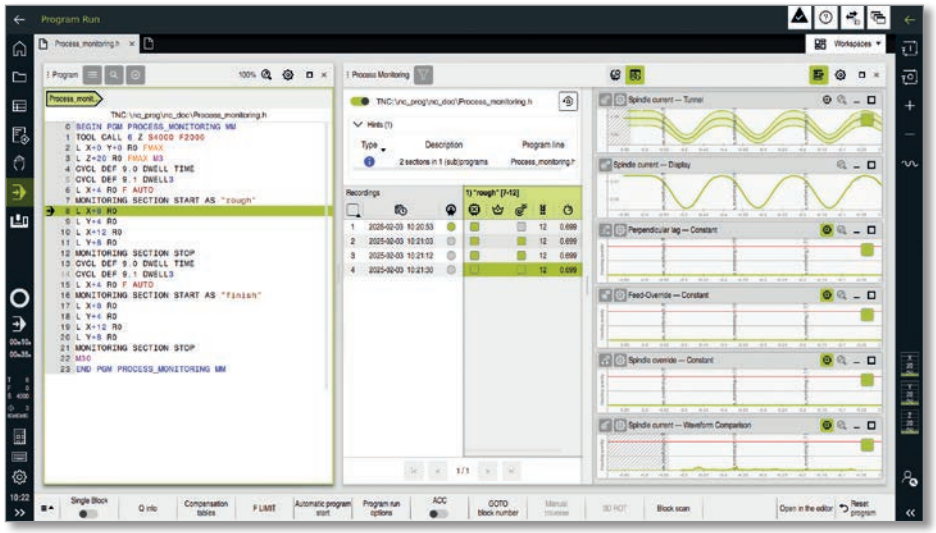
Machining functions
Monitoring the production process based on a teach-in cut

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 preventing 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 user can save time and costs with the fully-integrated process monitoring feature from HEIDENHAIN. With its intelligent process monitoring capability, the control detects disturbances early in the machining process and ensures reliable results. Errors such as tool breakage, skipped, or faulty pre-machining of a workpiece, deviations in the material, or changed workpiece positions are detected immediately.

The control monitors predefined process sections during program run. A monitoring task consists of the following:
- Signal: Define whatever is to be monitored (e.g., spindle current)
- Procedure: Define how the signal is analyzed (e.g., waveform comparison)
- Reaction: Define how the control reacts to deviations (e. g., stopping in the event of tool breakage)

Every monitoring task is graphically depicted to enable easy setup and tracking. The parameterization can be stored as a template and carried over to other machines.

Machining is followed by quality assurance. The user designates the workpiece as either a conforming or noncomforming part. As soon as 10 conforming parts have been identified, they are automatically used as a reference. The TNC7 then monitors all subsequent machining and reacts to deviations. 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 be automatically started with a replacement tool, thereby maintaining productivity despite disruptions. To do this, the process monitoring function uses internal control signals and requires no 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	**SIK** Option 168 ID 1432152-01	**SIK2** 5-01-1 ID 1395880-01
TNC7 **TNC7 basic** **TNC 640** **TNC 620** **TNC 320**	NC SW 81762x-16 and later – – – –	
Installation by the machine manufacturer		
For more information, see the *TNC7* brochure, and visit www.klartext-portal.com		

Machining functions
3D-ToolComp for 3D radius offset based on tool angle

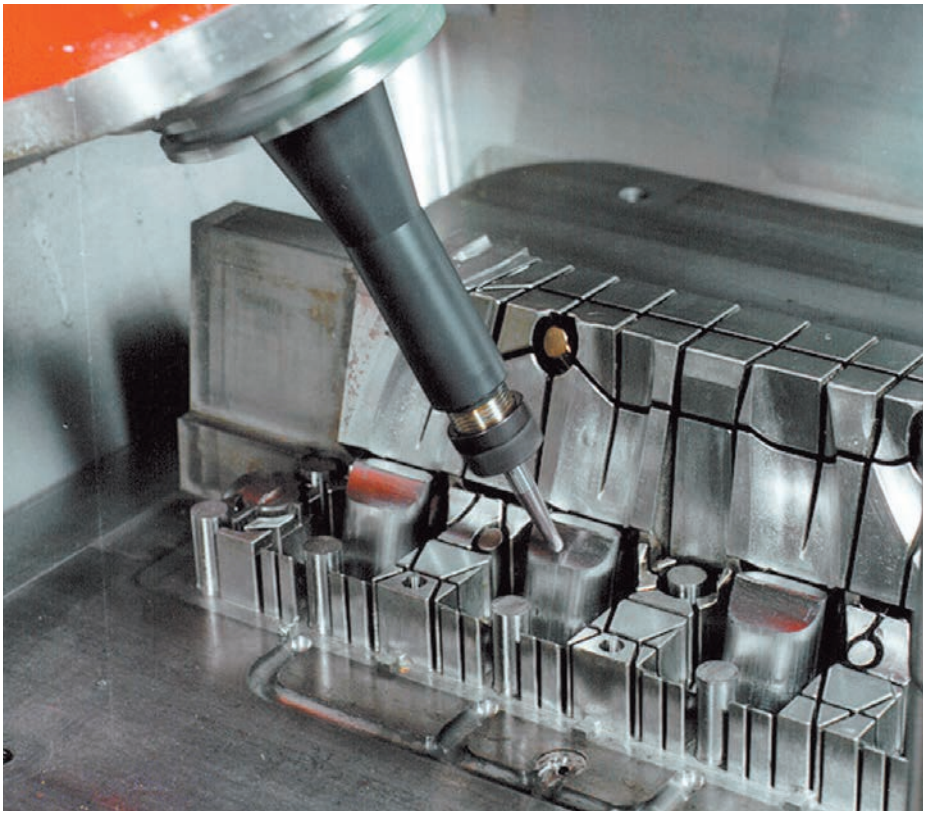
3D-ToolComp is a powerful software option for three-dimensional tool radius compensation. In a compensation table, angle-dependent delta values are defined describing the tool's deviation from an ideal circular form (see figure). For use with a tool, this function requires surface normal vectors in the NC program, for which the **Advanced Function Set 2** software option must be enabled.

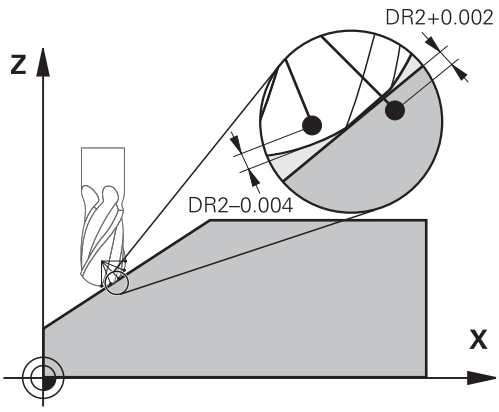
The TNC then corrects the radius value defined for the tool's current point of contact with the workpiece. In order to determine the point of contact exactly, the NC program must have been created with surface-normal blocks (LN blocks) by a CAM system. The surface-normal blocks specify the theoretical 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.

The **3D-ToolComp** software option is required for 3D calibration.





3D-ToolComp	SIK	SIK2
	Option 92	2-02-1
	ID 679678-01	ID 1395857-01
TNC7	NC SW 81762x-16 and later	
TNC7 basic	–	
TNC 640	NC SW 34059x-07 and later	
TNC 620	–	
TNC 320	–	

Installation by the user

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

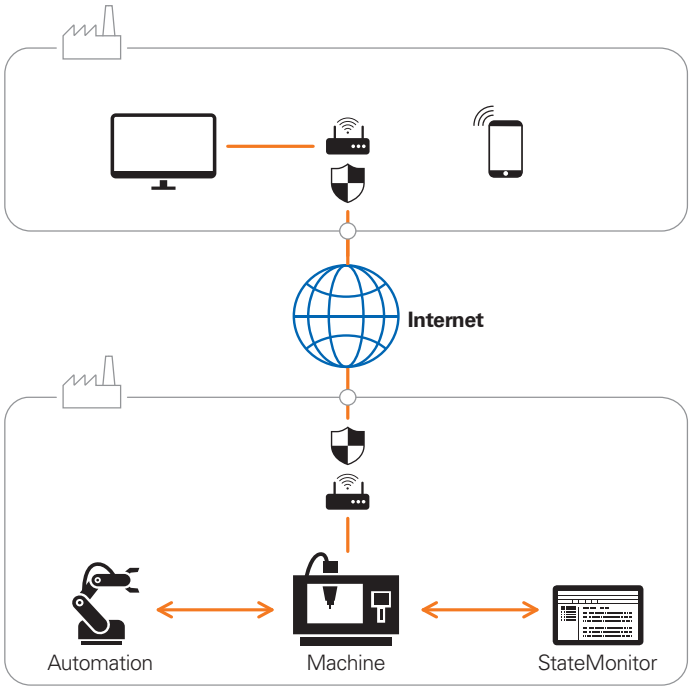
Communication
HEIDENHAIN OPC UA NC Server for standardized communication

The **OPC UA NC Server** software option offers an interface based on OPC UA for HEIDENHAIN controls. 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. When using a SIK2 and option 3-02-1, ten incoming connections are available. (SIK2 options can be purchased more than once.)

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	TNC7 Minimum NC SW	TNC7 basic Minimum NC SW	TNC 640 Minimum NC SW	TNC 620 Minimum NC SW
• Machine identification • Production data acquisition • Machine messages • Program sequence check	81762x-16	81762x-18	34059x-10	81760x-08
• File transfer • Remote management of file systems	81762x-16	81762x-18	34059x-11	81760x-08
• OEM enhancements (e.g., PLC)	81762x-16	81762x-18	34059x-16	81760x-16
• Tool data management	81762x-17	81762x-18	34059x-17	81760x-17
• Import of 3D tool models for DCM v2 • Other encryption algorithms • Various service functions	81762x-18	81762x-18	34059x-18	81760x-18

OPC UA NC Server	SIK	SIK2
	Options 56 to 61	3-02-1
	ID 1291434-01 to ID 1291434-06	ID 1395873-01
TNC7	NC SW 81762x-16 and later	
TNC7 basic	NC SW 81762x-18 and later	
TNC 640	NC SW 34059x-10 and later	
TNC 620	NC SW 81760x-08 and later	
TNC 320	–	
Installation by IT specialists		
For more information, visit www.heidenhain.com/products/software		

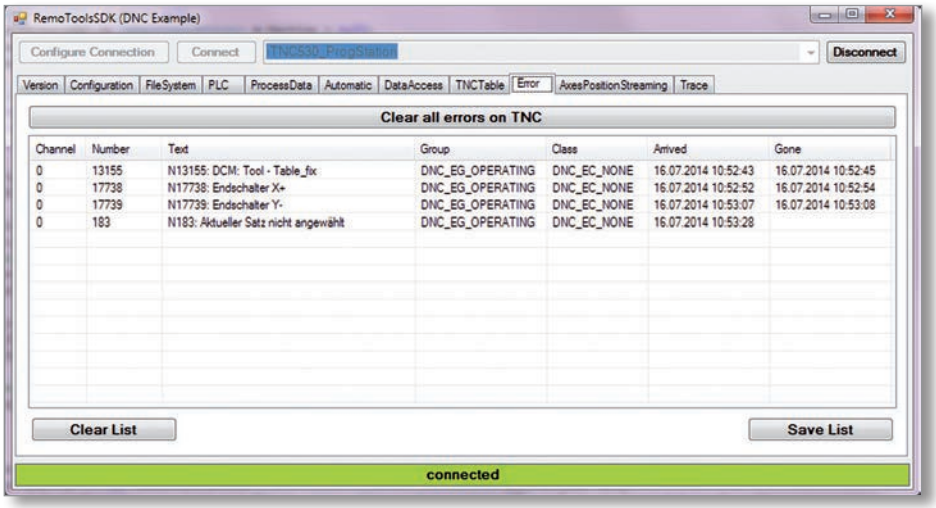
connected + machining

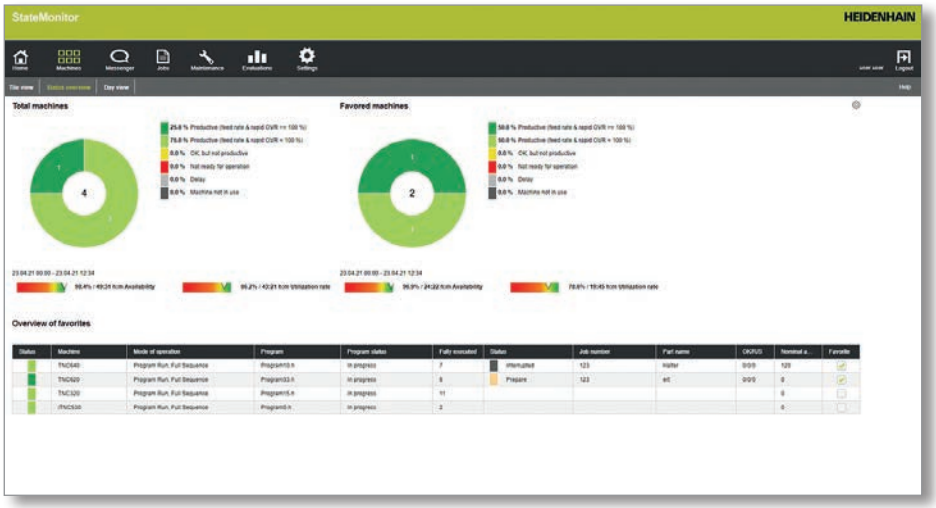
Communication
HEIDENHAIN DNC for communication over a COM component

Paperless job management requires the seamless exchange of data about all process steps in the production process. The **HEIDENHAIN DNC** software 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
- Standard 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.

HEIDENHAIN DNC	SIK	SIK2
	Option 18	3-03-1
	ID 526451-01	ID 1395874-01
RemoTools SDK	Accessory, ID 340442-xx	
TNC7	NC SW 81762x-16 and later	
TNC7 basic	NC SW 81762x-18 and later	
TNC 640	NC SW 34059x-01 and later	
TNC 620	NC SW 34056x-01/73498x-01/81760x-01 and later	
TNC 320	NC SW 34055x-01/771851-01 and later	
Installation by the user		
For more information, see the TNC brochures or visit www.heidenhain.com/products/software		

connected + machining

Communication
Process Tracking Interface (PTI)

With this software option, HEIDENHAIN offers a powerful function for the detailed recording and documentation of process data during machining. PTI permits live access to selected process signals during operation, including spindle speed, feed rate, or machine utilization, thus enabling the complete traceability of machining processes.

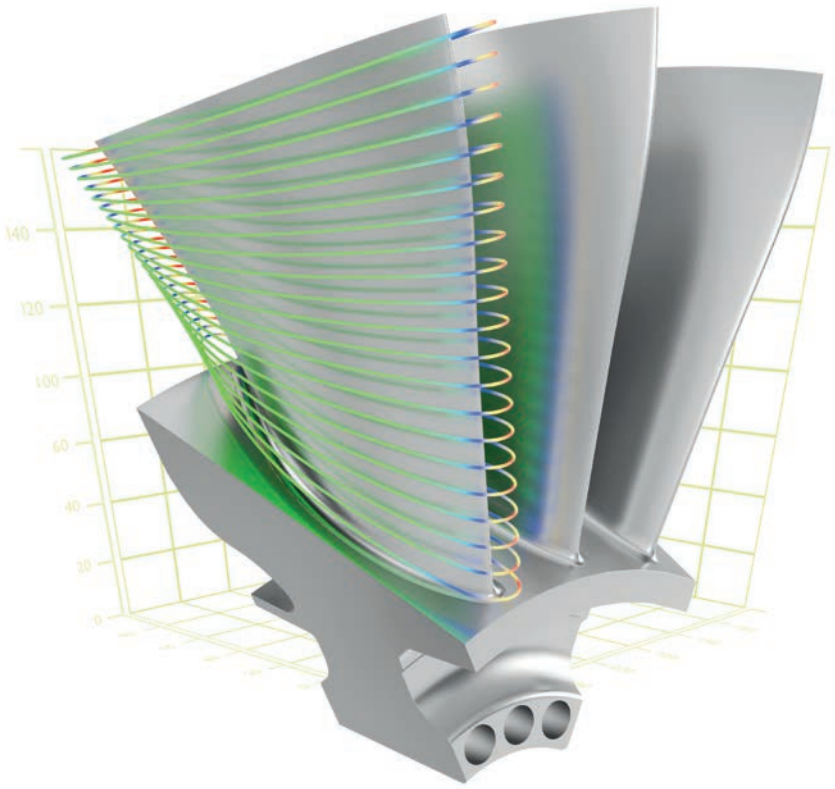
The data recorded with PTI can be used, for example, to create digital workpiece twins for quality assurance and to fulfill industry-specific documentation requirements (e.g., in the aerospace industry).

PTI is intended for experts and developers. It provides the basis for implementing one's own functionality using control signals to implement individualized process analyses, machine status monitoring, custom-developed solutions, and more. In constrast, Process Monitoring is a completed tool for end users that provides turnkey process monitoring functions.

PTI supports third-party software providers and machine manufacturers in developing their own solutions for data analysis, process monitoring, and machine status monitoring.

Overview of functions:
- Synchronous recording of up to 16 process signals simultaneously
- Two SIK2 options on the control each enable 8 recording channels
- Sampling interval of 3 ms for NC channels (recording of PLC channels follows the PLC clock)
- Suitable for use in productive operation
- Can be expanded to include PLC information
- For access, a C++ library is available with RemoTools SDK 4.0.

You can order this software option up to two times, each time selecting up to eight channels for simultaneous recording.



Process Tracking Interface (PTI)	**SIK**	**SIK2**
	–	3-04-1 ID 1433138-01
TNC7	NC SW 81762x-19 and later	
TNC7 basic	NC SW 81762x-19 and later	
TNC 640	–	
TNC 620	–	
TNC 320	–	
Installation by the user		
For more information, visit www.heidenhain.com/products/software		

Communication

Remote Desktop Manager for viewing and remotely operating 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** software option provides the user 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 user'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.

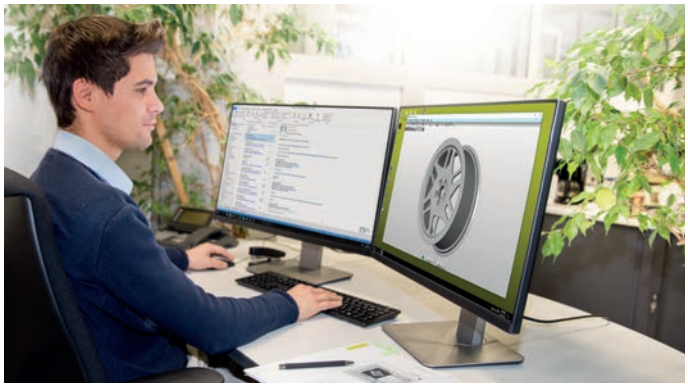


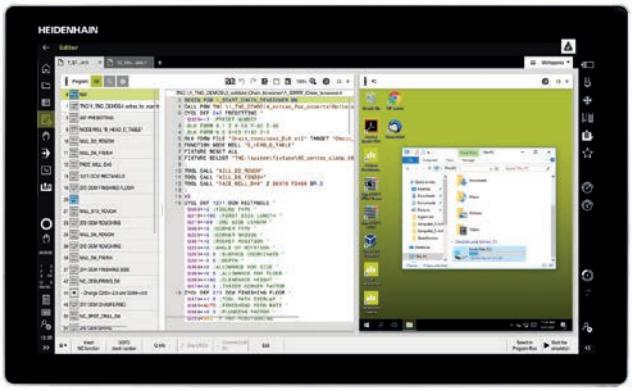
Extended Workspace Comfort

Extended Workspace Comfort
With Extended Workspace Comfort, you can add a second 19-inch HEROS screen next to or above the control's main screen. 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 control; 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 and TNC7 basic make it possible to seamlessly embed remote desktops or applications as a workspace or as a separate operating mode directly into the control's user interface. The workspaces are therefore able to display the content in an optimized way in the display areas selected by the user.





connected + machining

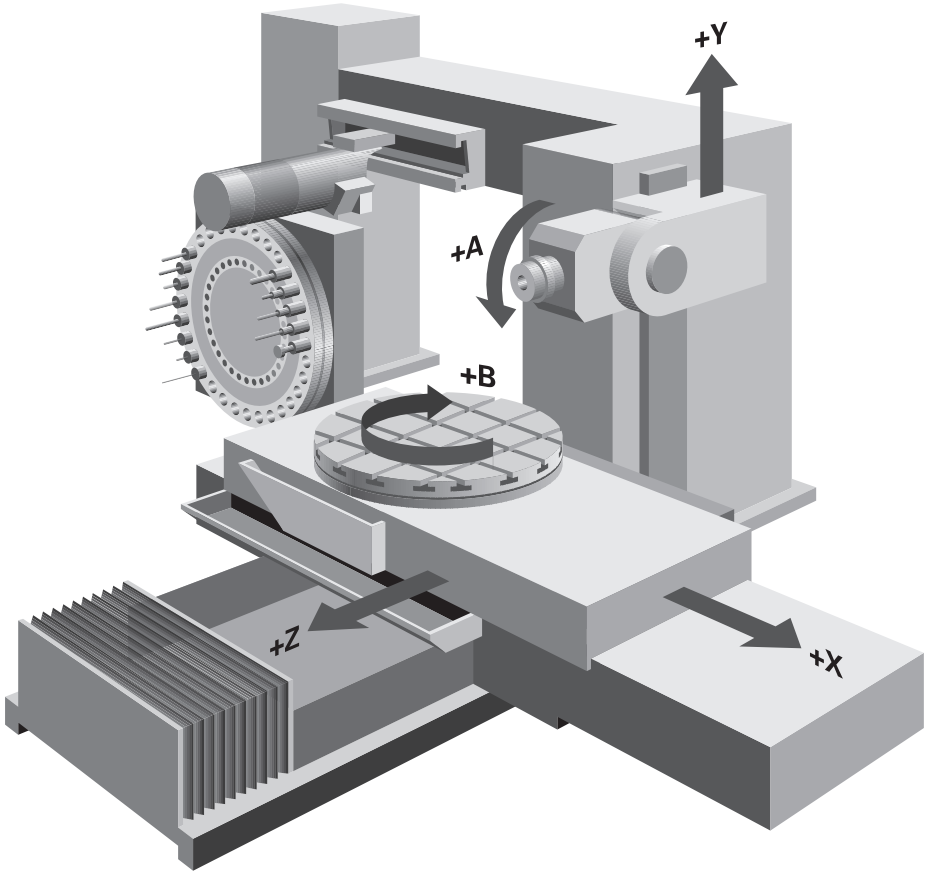
Remote Desk. Manager	SIK	SIK2
	Option 133	3-01-1
	ID 894423-01	ID 1395872-01
TNC7	NC SW 81762x-16 and later	
TNC7 basic	NC SW 81762x-18 and later	
TNC 640	NC SW 34059x-02 and later	
TNC 620	NC SW 81760x-02 and later	
TNC 320	NC SW 77185x-04 and later	
Installation by IT specialists		
For more information, see the TNC brochures or visit www.heidenhain.com/products/software		

Machine interfacing
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 at any time 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
- TNC7 basic: 8 control loops
- TNC 640: 24 control loops
- TNC 620: 8 control loops
- TNC 320: 6 control loops



Control Loop Qty.	SIK	ID	SIK2	ID
Additional Axis 1	Option 0	354540-01	6-01-1*	1395883-01
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		
4 Additional Axes	Option 77	634613-01	6-01-1*	1395883-01
8 Additional Axes	Option 78	634614-01		

TNC7	NC SW 81762x-16 and later
TNC7 basic	NC SW 81762x-18 and later
TNC 640	NC SW 34059x-01 and later
TNC 620	NC SW 34056x-01/73498x-01/81760x-01 and later
TNC 320	NC SW 34055x-01/771851-01 and later

Installation by the machine manufacturer

For more information, see the *Information for Machine Manufacturers* brochures

* This software option can be ordered multiple times in the desired quantity. The control automatically takes all enablings into account.

Machine interfacing
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.

Gen 3 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.

The machine manufacturer needs the **Add. FS Full** software option if safe control loops are to be enabled via single options and ten or more safe control loops are needed.
The **Add. FS Full** software option enables all remaining FS axes if the following requirements are fulfilled:
- The **Integrated FS: Basic** software option is enabled
- The **Add. FS Ctrl. Loop 1 to 5** software options are enabled

Option number		Option	Comment
SIK ID	**SIK2** ID		
160 1249928-01	**6-30-1** 1395886-01	Integrated FS: Basic	Gen 3 exclusive: enabling of functional safety and of four safe control loops
161 1249929-01	**6-30-2*** 1395887-01	Integrated FS: Full	Gen 3 exclusive: enabling of functional safety and of the maximum number of safe control loops
162 1249930-01		FS Control Loop Qty. (add. FS Ctrl. Loop 1)	Gen 3 exclusive: additional safe control loop (no. 1)
163 1249931-01		FS Control Loop Qty. (Add. FS Ctrl. Loop 2)	Gen 3 exclusive: additional safe control loop (no. 2)
164 1249932-01		FS Control Loop Qty. (Add. FS Ctrl. Loop 3)	Gen 3 exclusive: additional safe control loop (no. 3)
165 1249933-01		FS Control Loop Qty. (Add. FS Ctrl. Loop 4)	Gen 3 exclusive: additional safe control loop (no. 4)
166 1249934-01		FS Control Loop Qty. (Add. FS Ctrl. Loop 5)	Gen 3 exclusive: additional safe control loop (no. 5)
169 1319091-01		FS Control Loop Qty. (Add. FS Full)	Enabling all remaining FS control loops

* This software option can be ordered multiple times in the desired quantity. The control automatically takes all enablings into account.



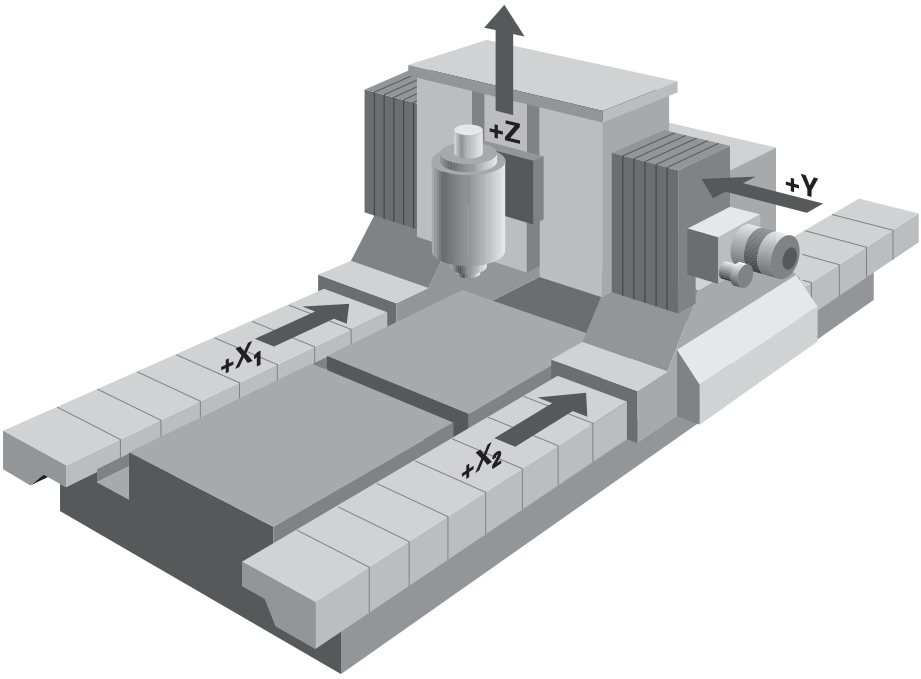
	Integrated FS: Basic **Integrated FS: Full** **FS Control Loop Qty. (Add. FS Ctrl. Loop 1-5)**	**Add. FS Full**
TNC7	NC SW 81762x-16 and later	NC SW 81762x-16 and later
TNC7 basic	NC SW 81762x-18 and later	NC SW 81762x-18 and later
TNC 640	NC SW 34059x-10 and later	NC SW 34059x-11 and later
TNC 620	NC SW 81760x-07 and later	NC SW 81760x-08 and later
Installation by the machine manufacturer		
For more information, see the Technical Information documents		

Machine interfacing
Operating 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	**SIK**	**SIK2**
	Option 24	6-03-1
	ID 634621-01	ID 1395885-01
TNC7	Standard function	
TNC7 basic	NC SW 81762x-18 and later	
TNC 640	Standard function	
TNC 620	NC SW 34056x-01/73498x-01/81760x-01 and later	
TNC 320	NC SW 34055x-01/771851-01 and later	
Installation by the machine manufacturer		
For more information, see the *Information for Machine Manufacturers* brochures		

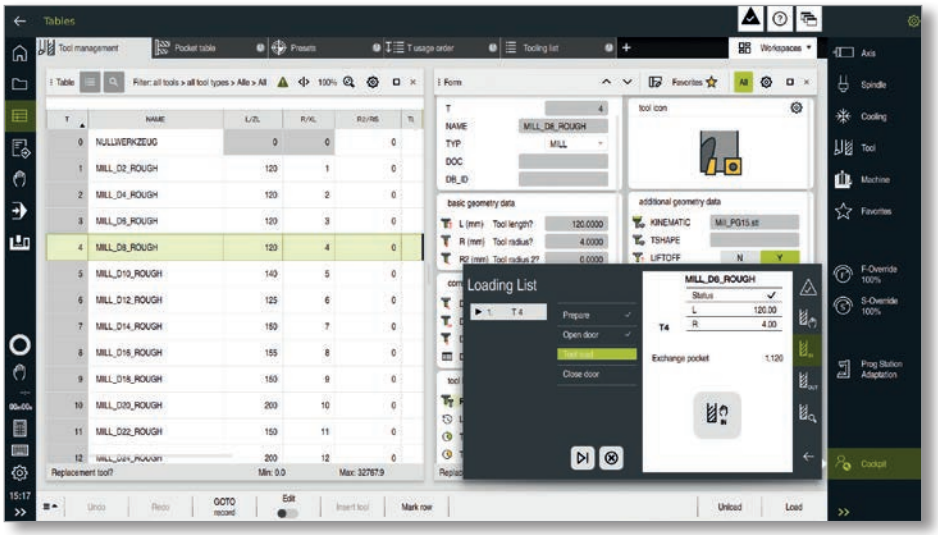
Machine interfacing
Python OEM Process for implementing special functions

The **Python OEM Process** software 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 masks) can also be executed without enabling the **Python OEM Process** software option. 10 MB of dedicated memory are reserved for this function. For more information, see the *Python in HEIDENHAIN Controls* Technical Manual.

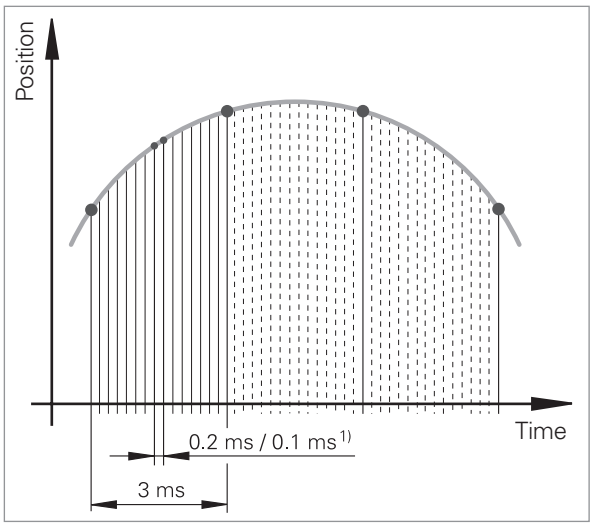


Python OEM Process	**SIK**	**SIK2**
	Option 46	7-01-1
	ID 579650-01	ID 1395889-01
TNC7	NC SW 81762x-16 and later	
TNC7 basic	NC SW 81762x-18 and later	
TNC 640	NC SW 34059x-01 and later	
TNC 620	NC SW 34056x-01/73498x-01/81760x-01 and later	
TNC 320	NC SW 34055x-04/771851-01 and later	
Installation by the machine manufacturer		
For more information, see the *Information for Machine Manufacturers* brochures		

Machine interfacing
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. The **Double Speed Axes** software option 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.



1) Single speed/Double speed (with the **Double Speed Axes** software option)

Control loop cycle times

Fine interpolation
Single-speed: 0.2 ms
Double-speed: 0.1 ms (with the **Double Speed Axes** software option)

Position controller
Single-speed: 0.2 ms
Double-speed: 0.1 ms (with the **Double Speed Axes** software option)

Speed controller
Single-speed: 0.2 ms
Double-speed: 0.1 ms (with the **Double Speed Axes** software option)

Current controller

f_{PWM}	T_{INT}
3333 Hz	150 µs
4000 Hz	120 µs
5000 Hz	100 µs
6666 Hz	75 µs (with the **Double Speed Axes** software option)
8000 Hz	60 µs (with the **Double Speed Axes** software option)
10 000 Hz	50 µs (with the **Double Speed Axes** software option)

Double Speed Axes	**SIK**	**SIK2**
	Option 49	6-02-1
	ID 632223-01	ID 1395884-01
TNC7	NC SW 81762x-16 and later	
TNC7 basic	NC SW 81762x-18 and later	
TNC 640	NC SW 34059x-01 and later	
TNC 620	NC SW 34056x-01/73498x-01/81760x-01 and later	
TNC 320	–	
Installation by the machine manufacturer		
For more information, see the *Information for Machine Manufacturers* brochures		

Machine interfacing
OEM options

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 user as options. A reserved area in the option menu (SIK) facilitates convenient and versatile enabling of these software options for the user.

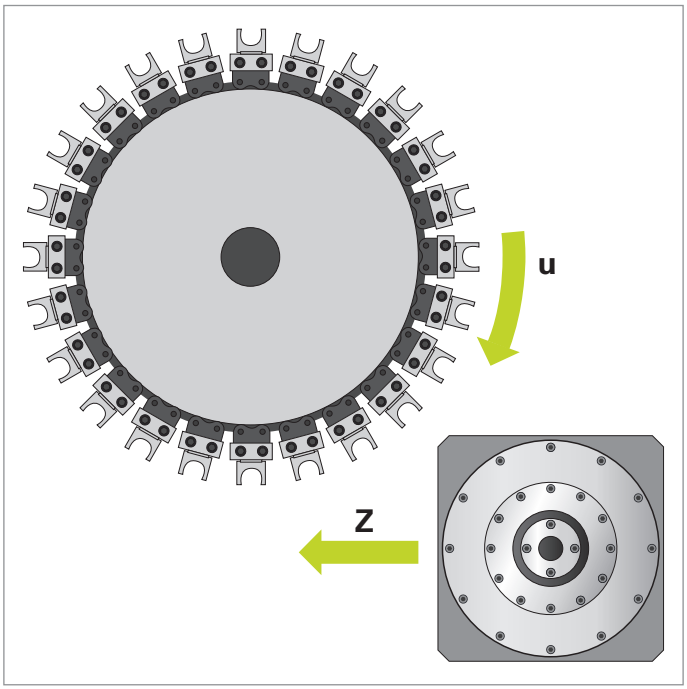
Options 101 to 130 provide thirty options that can be enabled by the machine manufacturer and enabled through his own PLC program. A particular benefit is the user's ability to enable options via the SIK menu without the need for on-site support by the machine manufacturer.

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

Machine interfacing

Real-time coupling function (RTC) 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.



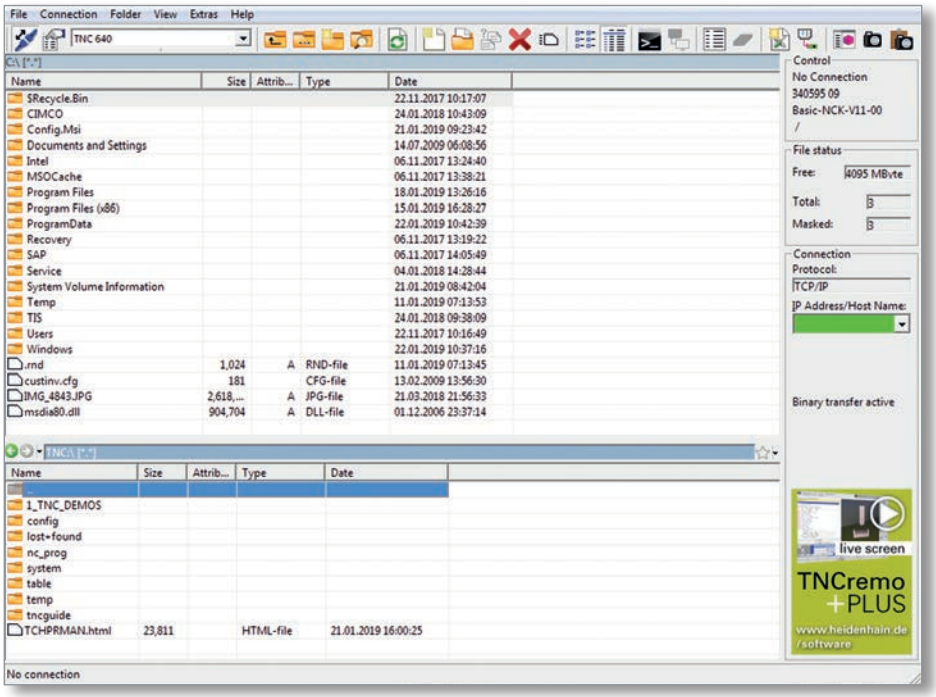
Synchronizing Functions	SIK	SIK2
	Option 135	7-03-1
	ID 1085731-01	ID 1395891-01
TNC7	NC SW 81762x-16 and later	
TNC7 basic	–	
TNC 640	NC SW 34059x-04 and later	
TNC 620	–	
TNC 320	–	
Installation by the machine manufacturer		
For more information, see the *Information for Machine Manufacturers* brochures		

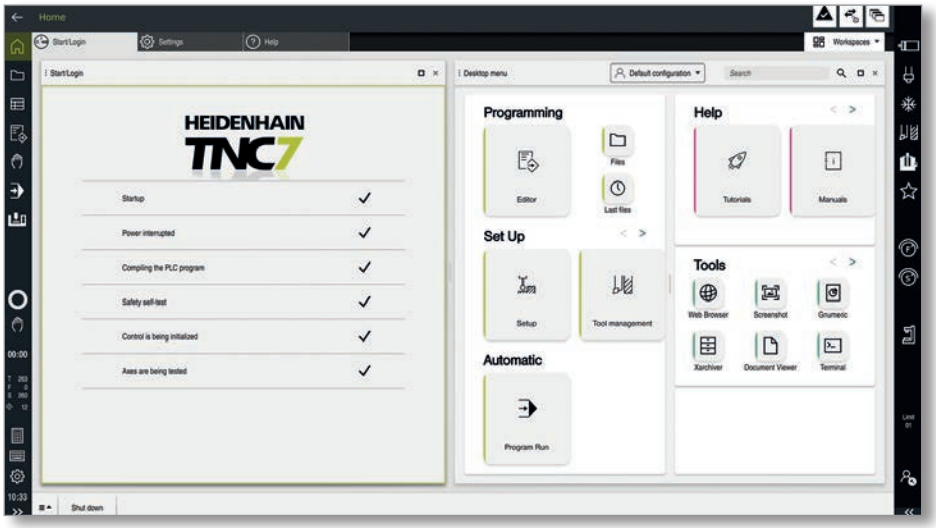
Software
TNCremo programs for data transfer

The free PC software package TNCremo assists the user 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.



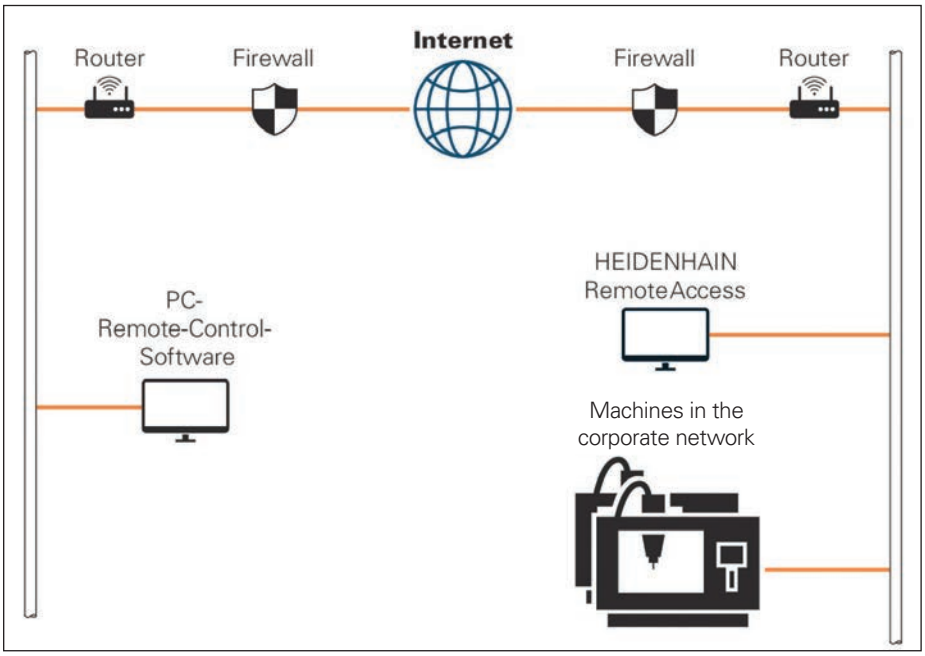


TNCremo	Free download
TNCremoPlus	Enabling module requires a fee ID 340447-xx
TNC7 **TNC7 basic** **TNC 640** **TNC 620** **TNC 320**	NC SW 81762x-16 and later NC SW 81762x-18 and later NC SW 34059x-01 and later NC SW 34056x-01/73498x-01/81760x-01 and later NC SW 34055x-01/771851-01 and later
Installation by the user	
For more information, visit www.klartext-portal.com	

Software
RemoteAccess for remote diagnostics of 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
- Automatic integration of HEIDENHAIN PC tools
- Can be enhanced with OEM-specific applications

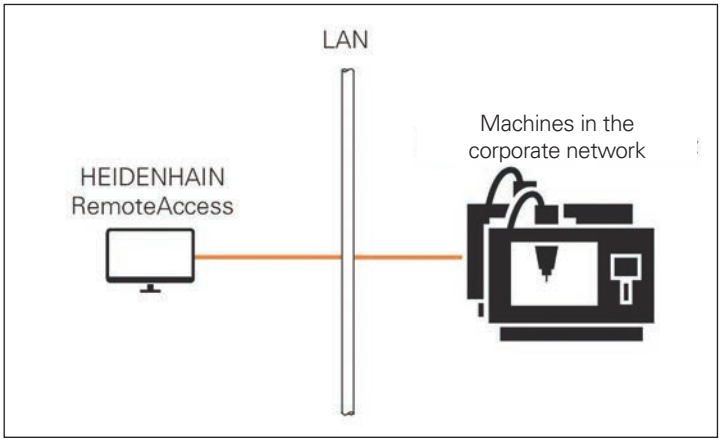
Single station license ID 1339577-01
Network license for 14 stations (ID 1339577-02)
Network license for 20 stations (ID 1339577-03)



Remote connection over PC remote control software

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 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 and later
TNC7 basic	NC SW 81762x-18 and later
TNC 640	NC SW 34059x-01 and later
TNC 620	NC SW 34056x-01/73498x-01/81760x-01 and later
TNC 320	NC SW 34055x-01/771851-01 and later
Installation by the machine manufacturer	

Software
StateMonitor: for collecting and evaluating 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 machines' utilization. This collection of data helps to uncover potential for optimization in order to lower costs and use machines more efficiently.



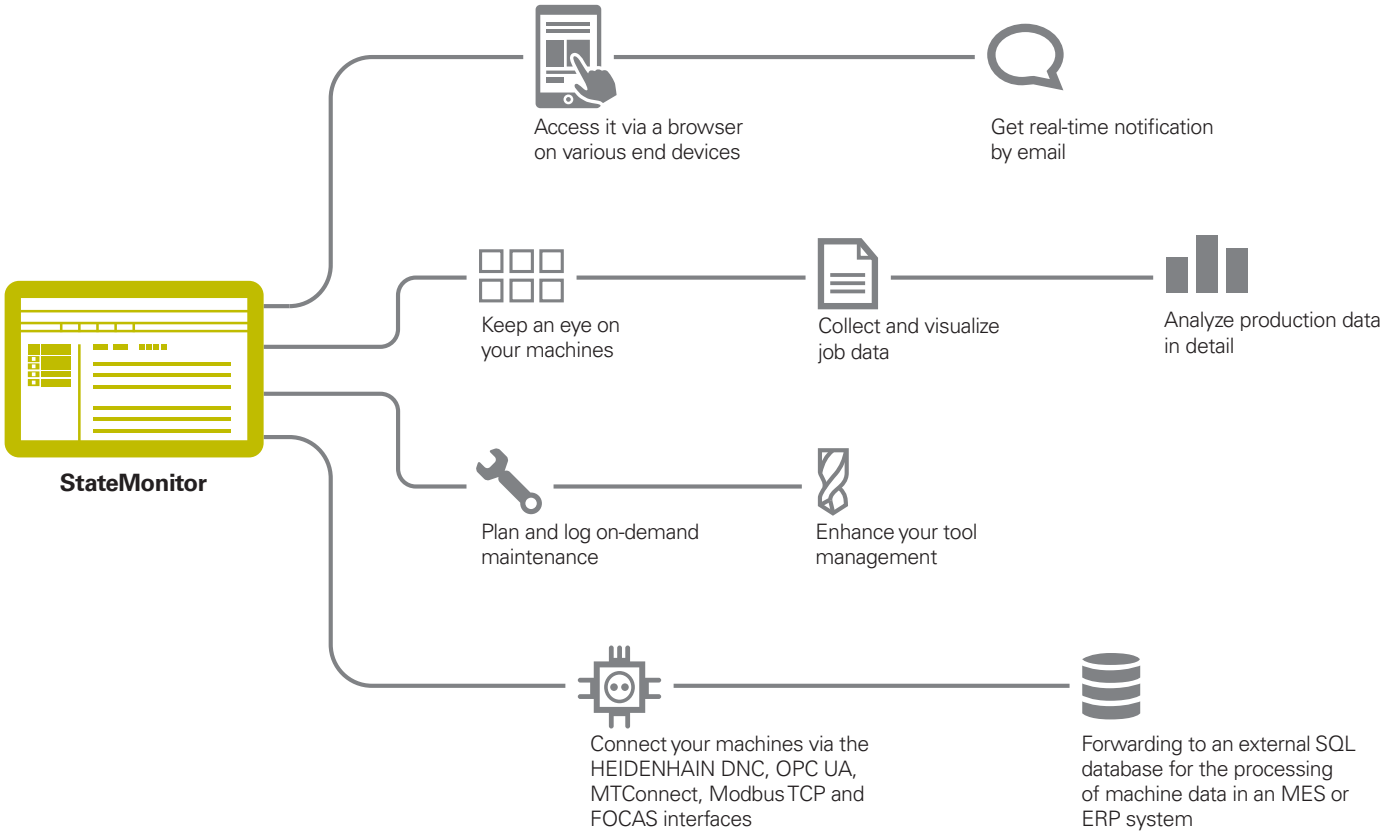
StateMonitor can record and visualize the following information:

- Machine status
- Program status and program name, as well as subprogram names if applicable
- Program run-time
- Machine messages
- Operating mode
- Override positions (spindle, feed rate, rapid traverse)
- SIK number and NC software number
- Maintenance entries and the maintenance history
- Job durations and job quantities
- Other freely definable machine-specific signals
- Machine-specific signal values



Machines can be connected to StateMonitor regardless of the controls they are operated with. The software supports the protocol types HEIDENHAIN DNC, OPC UA, MTConnect, Modbus TCP and FOCAS.

For more information, visit:
www.klartext-portal.com/software/machine-data-collection



StateMonitor	ID 1218930-xx
5 Machines	ID 1220884-xx
MaintenanceManager	ID 1308520-xx
JobTerminal	ID 1268674-xx
5 Signals	ID 1308521-xx
OPC UA Interface	ID 1268673-xx
MTConnect Interface	ID 1268675-xx
Modbus Interface	ID 1268670-xx
FOCAS Interface	ID 1385356-xx
Data Interface	ID 1367514-xx

TNC7	NC SW 81762x-16 and later
TNC7 basic	NC SW 81762x-18 and later
TNC 640	NC SW 34059x-01 and later
TNC 620	With NC-SW 34056x-01/73498x-01/81760x-01 or later
TNC 320	NC SW 34055x-01/771851-01 and later

The following software option must be enabled: **HEIDENHAIN DNC**

Installation by the machine manufacturer or end user

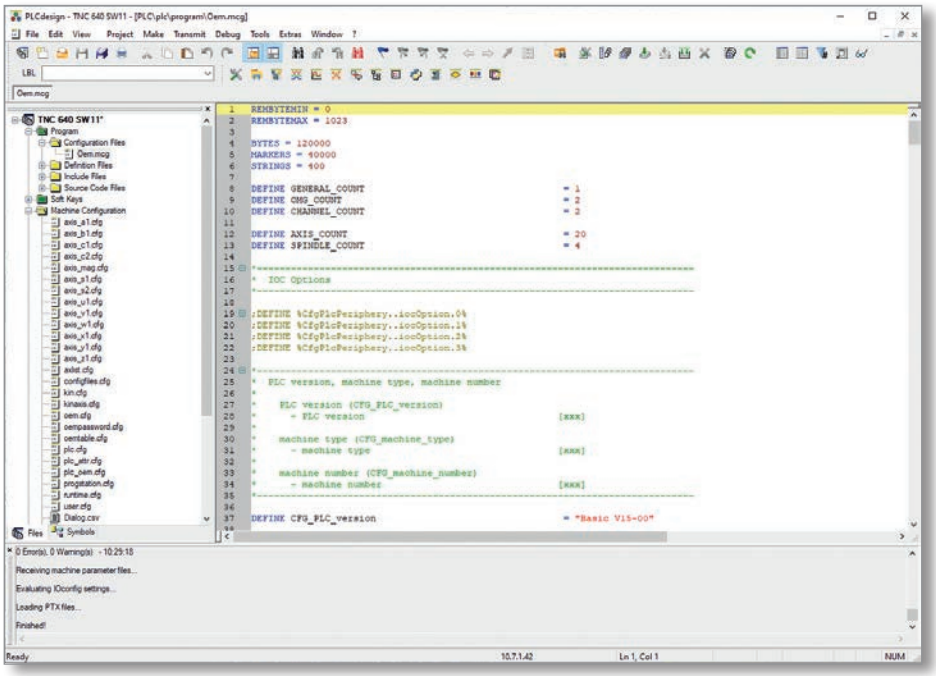
For more information, visit www.heidenhain.com/products/software and www.klartext-portal.com, and see the *Digital Shop Floor* flyer

Software
PLCdesign software for developing PLC programs

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 STL
- 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 and later
TNC7 basic	NC SW 81762x-18 and later
TNC 640	NC SW 34059x-01 and later
TNC 620	NC SW 34056x-01/73498x-01/81760x-01 and later
TNC 320	NC SW 34055x-01/771851-01 and later

Installation by machine manufacturers or servicing providers

For more information, see the *Information for Machine Manufacturers* brochures

Software
KinematicsDesign for creating machine kinematic models

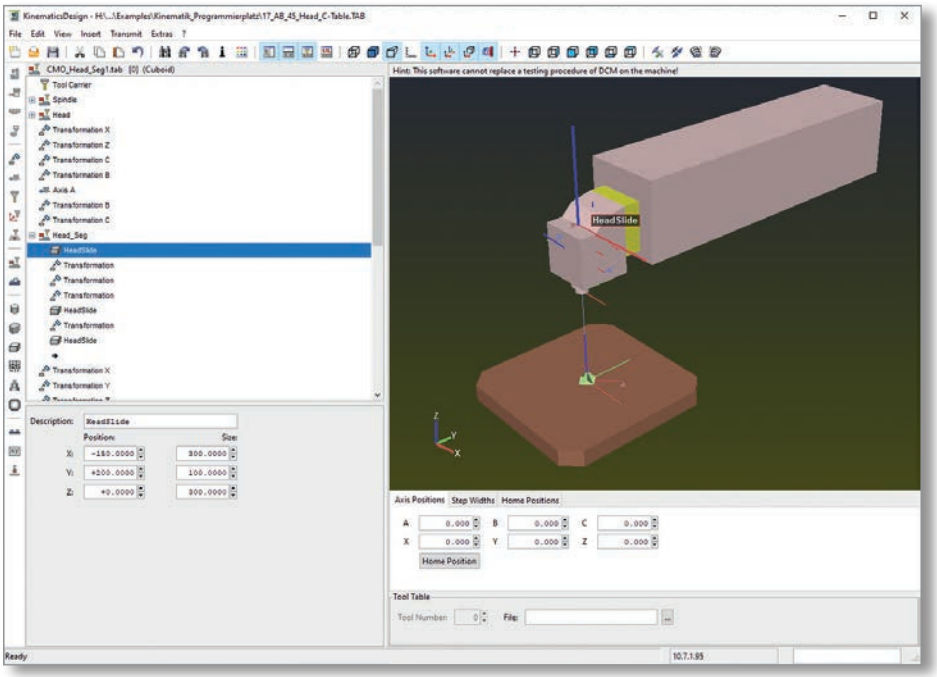
KinematicsDesign is a PC software application for the graphically supported creation of kinematic descriptions for HEIDENHAIN controls. It also offers a convenient way to configure and start using the DCM collision monitoring function.

KinematicsDesign can create all objects of the kinematic chain, transfer them to and from the control, and edit them. Extensive visualization possibilities allow the user to display the currently needed information 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 is a PC tool that 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, TNC7 basic, 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 user can always keep the difference from the original object in view. M3D Converter requires a software release module (ID 1124969-xx).



KinematicsDesign	Registration required
TNC7	NC SW 81762x-16 and later
TNC7 basic	NC SW 81762x-18 and later
TNC 640	NC SW 34059x-01 and later
TNC 620	NC SW 34056x-01/73498x-01/81760x-01 and later
TNC 320	NC SW 34055x-01/771851-01 and later
Installation by machine manufacturers or servicing providers	
For more information, see the *Information for Machine Manufacturers* brochures	

M3D Converter	Enabling module requires a fee ID 1124969-xx
TNC7	NC SW 81762x-16 and later
TNC7 basic	NC SW 81762x-18 and later
TNC 640	NC SW 34059x-05 and later
TNC 620	NC SW 81760x-02 and later
TNC 320	NC SW 77185x-02 and later
Installation by machine manufacturers or servicing providers	
For more information, see the *Information for Machine Manufacturers* brochures	

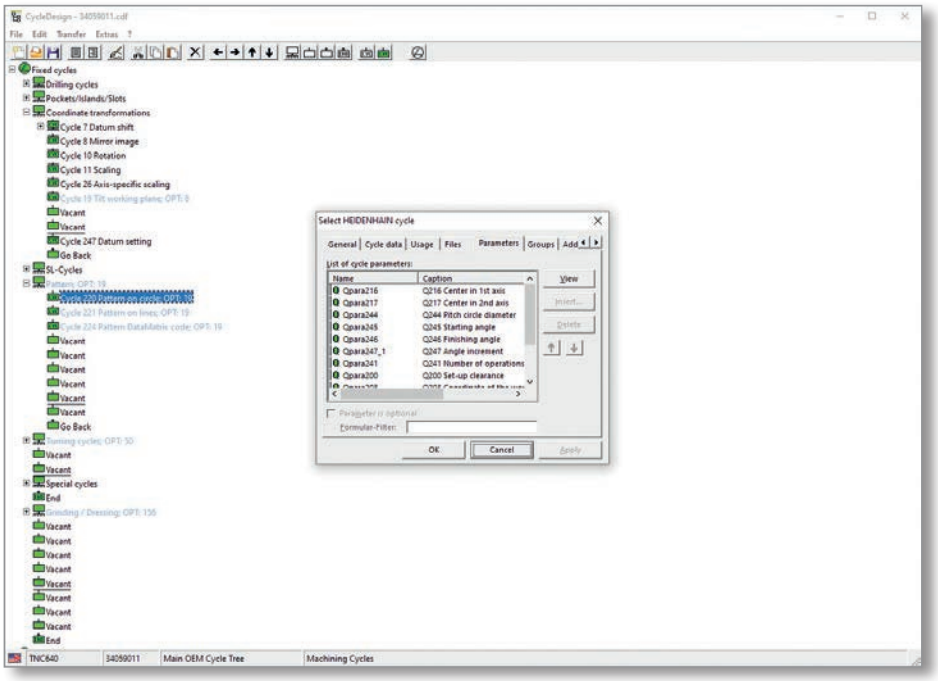
Software
CycleDesign for saving NC subprograms as cycles

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 and TNC7 basic). You can choose whether to add your cycles to the HEIDENHAIN cycle bar on the side or completely replace the HEIDENHAIN cycle bar.

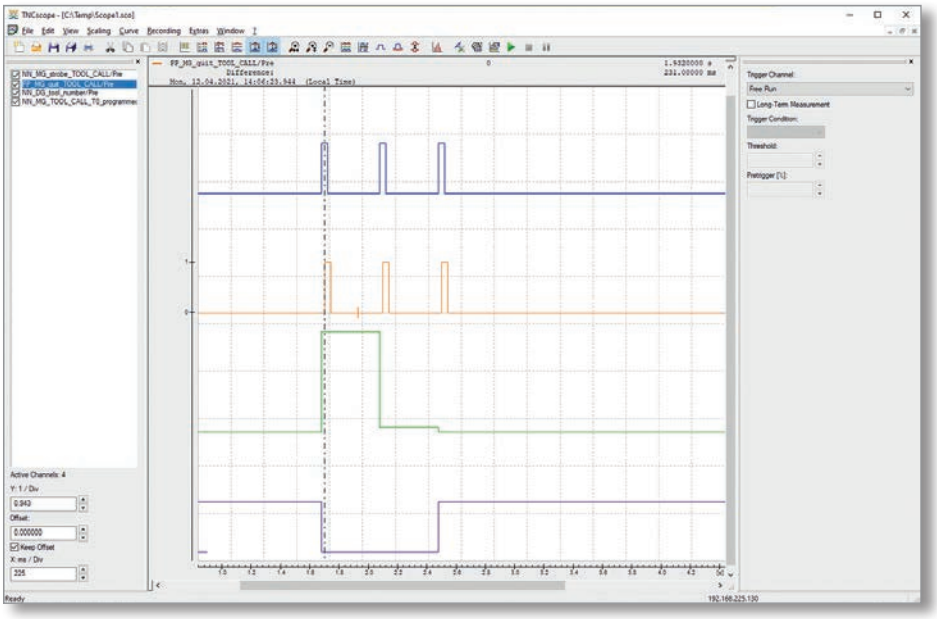
CycleDesign lets you save this cycle data in the control's memory.



CycleDesign	Registration required
TNC7	NC SW 81762x-16 and later
TNC7 basic	NC SW 81762x-18 and later
TNC 640	NC SW 34059x-01 and later
TNC 620	NC SW 34056x-01/73498x-01/81760x-01 and later
TNC 320	NC SW 34055x-01/771851-01 and later
Installation by users or machine manufacturers	
For more information, see the *Information for Machine Manufacturers* brochures	

Software
TNCscope for exporting oscilloscope files to a PC

TNCscope is a PC oscilloscope software program for machine manufacturers to record and evaluate 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)

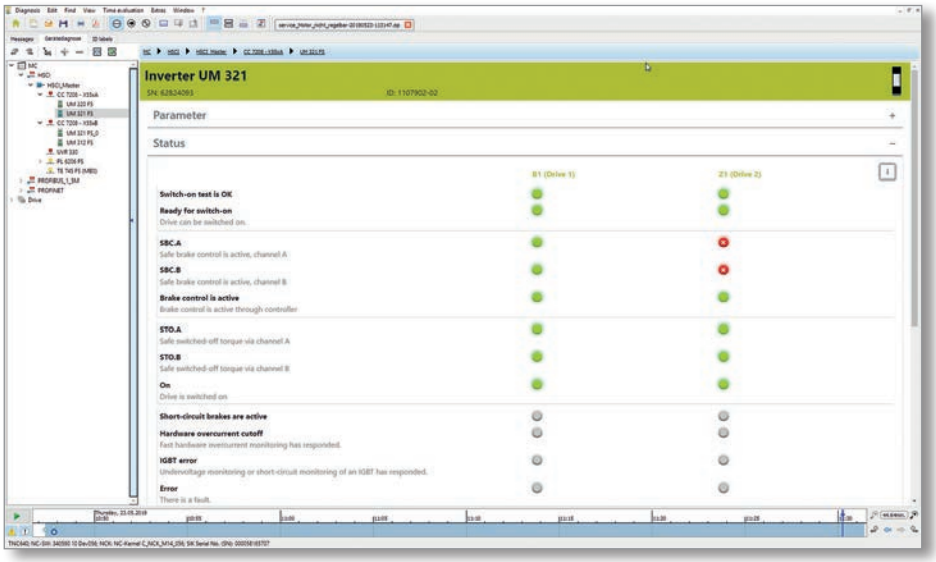
TNCscope	Registration required
TNC7	NC SW 81762x-16 and later
TNC7 basic	NC SW 81762x-18 and later
TNC 640	NC SW 34059x-01 and later
TNC 620	NC SW 34056x-01/73498x-01/81760x-01 and later
TNC 320	NC SW 34055x-01/771851-01 and later
Installation by machine manufacturers or servicing providers	
For more information, see the *Information for Machine Manufacturers* brochures	

Software

TNCdiag for evaluating status information and diagnostic information

Based on the HFL interface (HEIDENHAIN Fiber 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

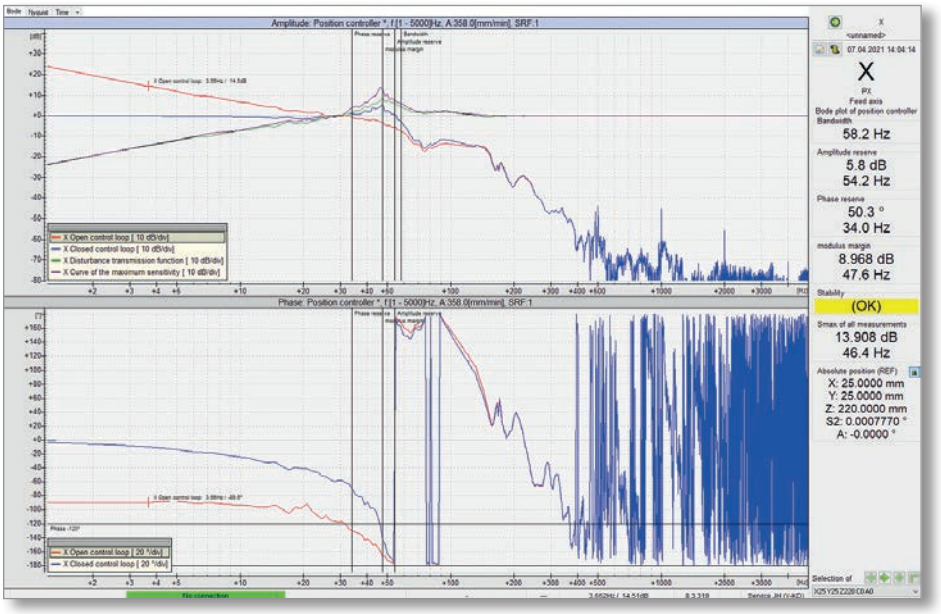
TNCdiag	Registration required
TNC7	NC SW 81762x-16 and later
TNC7 basic	NC SW 81762x-18 and later
TNC 640	NC SW 34059x-10 and later
TNC 620	NC SW 81760x-07 and later
TNC 320	–
Installation by machine manufacturers or servicing providers	
For more information, see the *Information for Machine Manufacturers* 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)



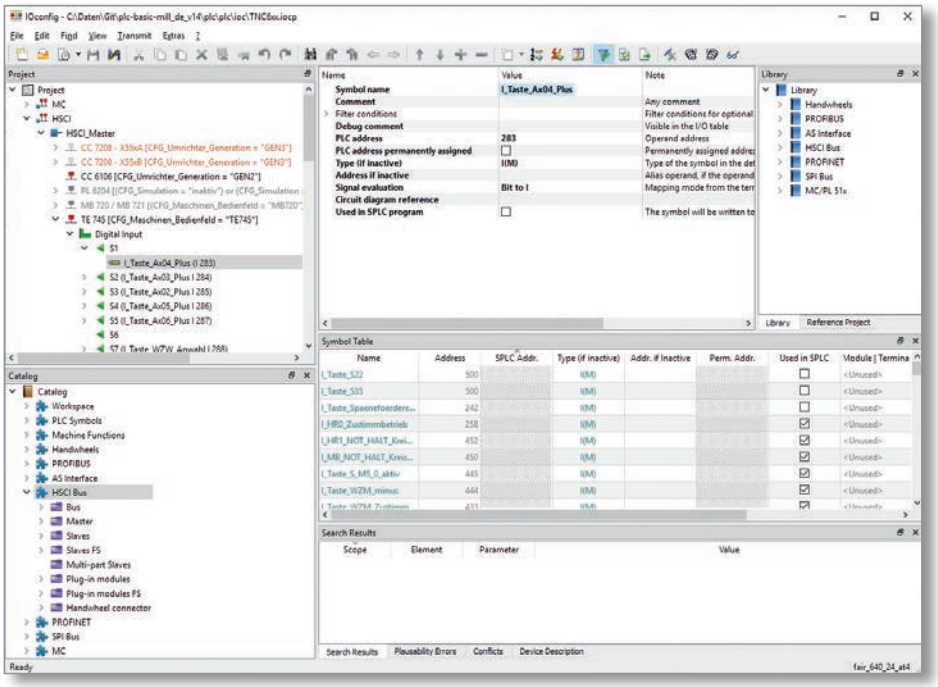
TNCopt	Registration required
TNC7	NC SW 81762x-16 and later
TNC7 basic	NC SW 81762x-18 and later
TNC 640	NC SW 34059x-04 and later
TNC 620	NC SW 81760x-01 and later
TNC 320	–
Installation by machine manufacturers or servicing providers	
For more information, see the *Information for Machine Manufacturers* brochures	

Software
IOconfig for configuring 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.



IOconfig	Registration required
TNC7	NC SW 81762x-16 and later
TNC7 basic	NC SW 81762x-18 and later
TNC 640	NC SW 34059x-01 and later
TNC 620	NC-SW 34056x-01/73495x-01/81760x-01 and later
TNC 320	NC SW 77185x-01 and later

Installation by machine manufacturers or servicing providers

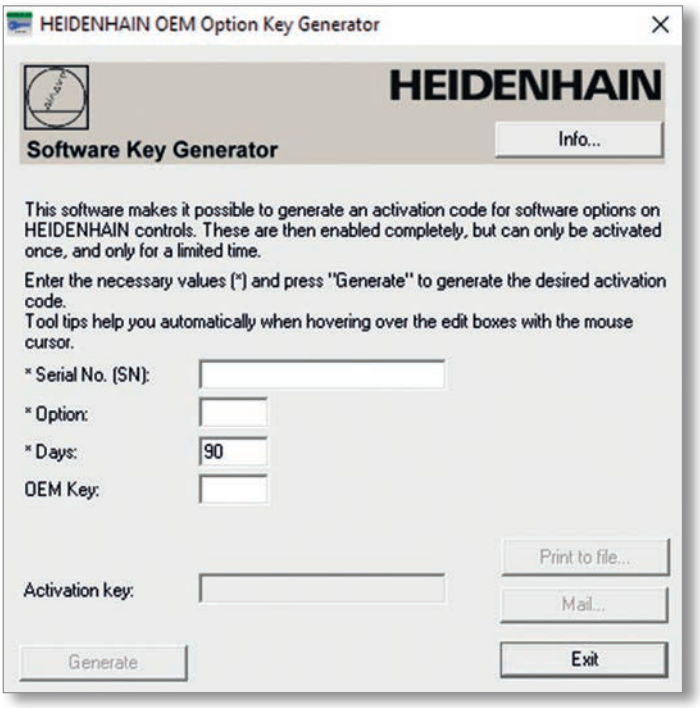
For more information, see the *Information for Machine Manufacturers* brochures

Software
Enabling of machine manufacturer software options

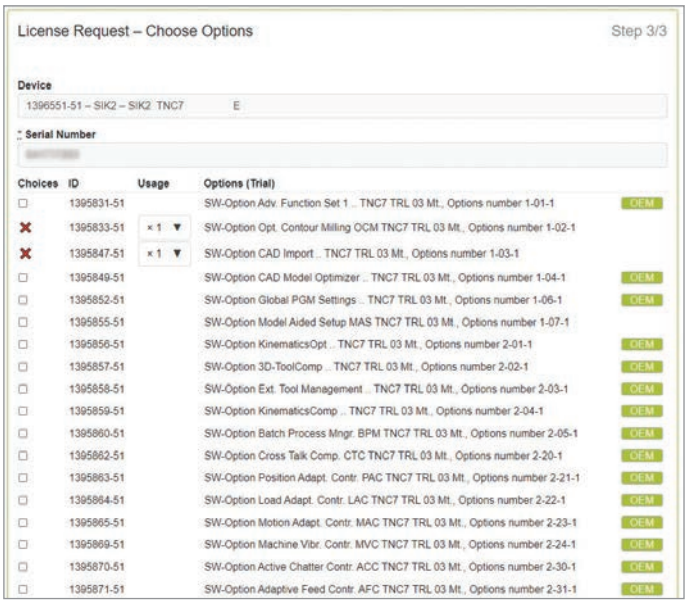
The software options are saved on the SIK (System Identification Key) plug-in board. Software options can be enabled for up to 90 days at no charge for comprehensive testing.

The plug-in board (SIK or SIK2) determines which of the following applications you use to generate the temporary enabling key for software options of HEIDENHAIN controls:

SIK
With the Software Key Generator PC tool:
Manual entry of an enabling key



SIK2
HEIDENHAIN Portal License Key web application:
A license file is transmitted to the control



Software Key Generator/ Web application	Registration required
TNC7	NC SW 81762x-16 and later
TNC7 basic	NC SW 81762x-18 and later
TNC 640	NC SW 34059x-01 and later
TNC 620	NC SW 34056x-03/73498x-01/81760x-01 and later
TNC 320	NC SW 34055x-05/771851-01 and later
Installation by machine manufacturers	
For more information, see the *Information for Machine Manufacturers* brochures	

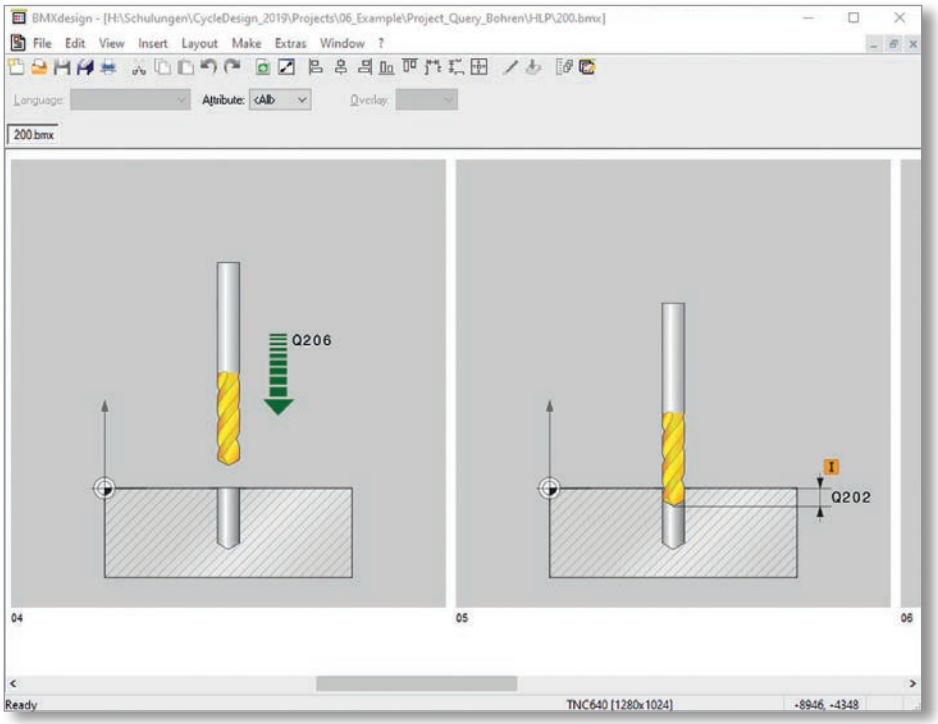
Software
BMXdesign

BMXdesign allows you to interactively create BMX project files and final 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 graphical and text 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 and later
TNC7 basic	NC SW 81762x-18 and later
TNC 640	NC SW 34059x-01 and later
TNC 620	NC SW 34056x-01/73498x-01/81760x-01 and later
TNC 320	NC SW 34055x-01/771851-01 and later

Installation by machine manufacturers

For more information, see the *Information for Machine Manufacturers* 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
The programming, testing, and optimizing of HEIDENHAIN Klartext or G-code programs with the programming station substantially reduces machine idle times. And it's easy to get used to because every keystroke feels the same: the programming station's keyboard is just like the one on the machine.

Testing programs created offline
Of course, you can also test programs that were written on a CAD/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 with G codes.

What does the programming station look like?
The programming station is an application for PCs with Windows operating systems. The software offers you the same TNC user interface and graphical support as the control on your machine. There are also different operating interfaces.

Which programming stations are available?
Programming stations are available for all current TNC controls. The programming station contains:
- Programming station software
- VirtualBox (VBox) virtualization software from ORACLE
- VBox expansion: TNC VB Base Software and data for the installation and operation of all programming stations (Control Center and virtual keyboard)



TNC7, TNC 640, TNC 620/TNC 320, iTNC 530 programming stations	
With TNC programming station keyboard	ID 1113967-03
With virtual keyboard	
Single station license	ID 1113924-04
Network license, 1 station	ID 1125955-04
Network license, 14 stations	ID 1113926-04
Network license, 20 stations	ID 1113928-04

Programming station keyboard	
Operating panel without dongle	ID 1113967-52
TNC7 programming station keyboard	ID 1380256-01

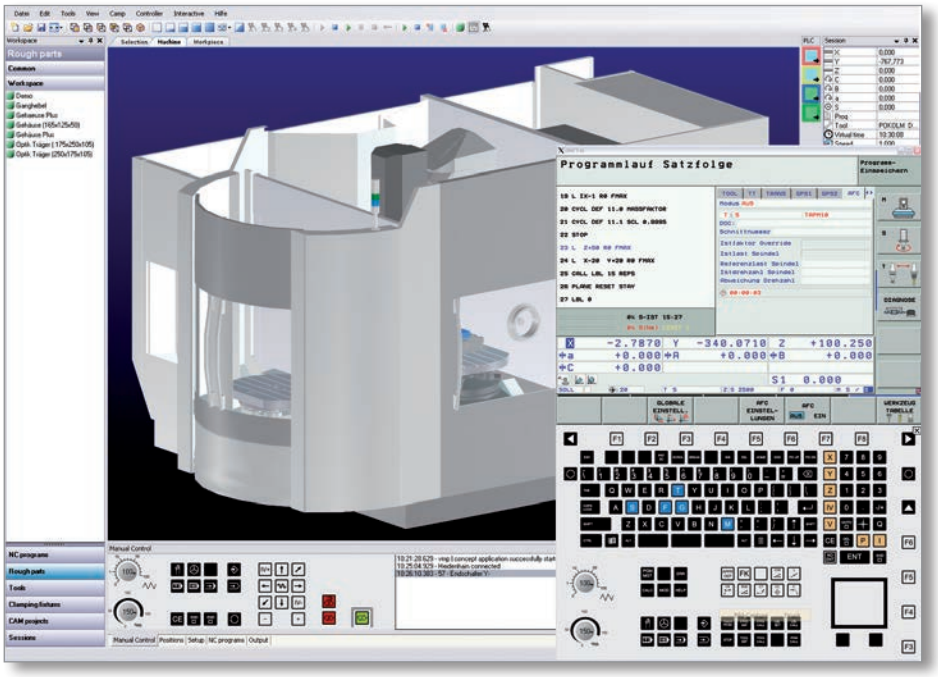
Software
virtualTNC for control of virtual machines

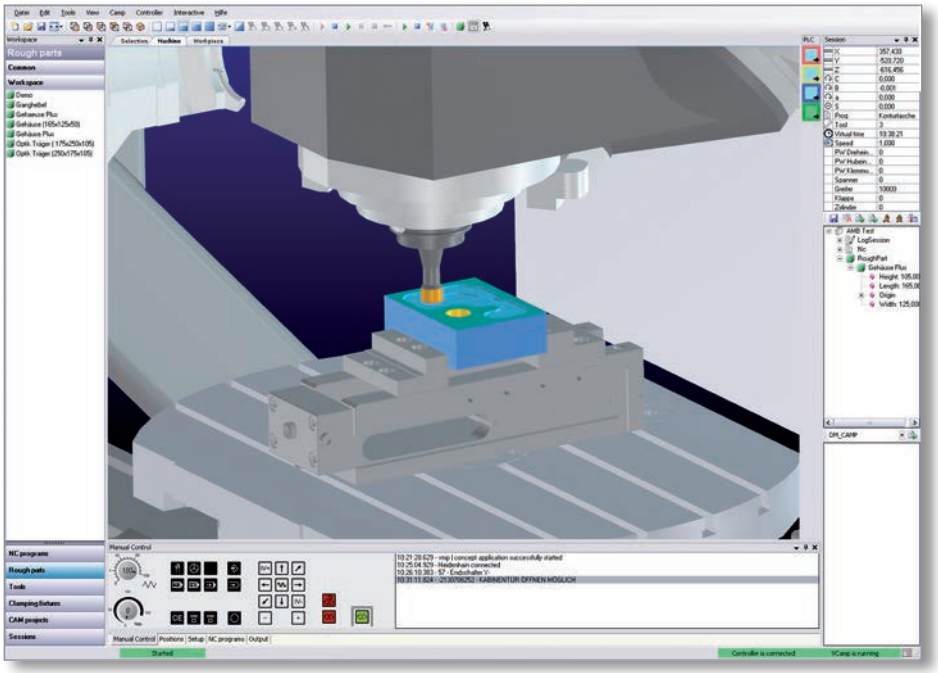
The ability to simulate NC programs on the control has long been a feature of HEIDENHAIN contouring controls. The PC software virtualTNC makes it possible to use TNC controls as a control component for machine-simulation applications on external computer systems.

How a virtual machine with virtualTNC works
Machine simulation applications 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. Users program and operate the control in the same way as they do an actual HEIDENHAIN control.

virtualTNC is the programming station software of the respective HEIDENHAIN control with a special interface that enables the machine simulation software to identify the current axis positions of the running "virtual" control. In order to simulate the interaction with a real machine, virtualTNC also makes the input and output states of the integrated PLC available.

Connecting virtualTNC over HEIDENHAIN DNC
Software manufacturers who would like to simulate a production system can connect their machine simulation to virtualTNC via HEIDENHAIN DNC. The axis positions must be coupled by the AxisPositionStreaming interface of the COM component. The JHIOsim DLL is used to read the PLC input and output states. Both are included in the RemoTools SDK development package.
A software release module is required to enable the functionality of the interface.





virtualTNC is available for a fee	**TNC7/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 *RemoTools SDK/virtualTNC* 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** Minimum NC SW	**TNC7 basic** Minimum NC SW	**TNC 640** Minimum NC SW	**TNC 620** Minimum NC SW	**TNC 320** Minimum NC SW
HR 130	ID 540940-01/540940-03	81762x-16	81762x-18	34059x-01	34056x-01/73498x-01/81760x-01	34055x-01
HR 180	ID 540940-16/540940-17	81762x-19	81762x-19	34059x-01	34056x-01/73498x-01/81760x-01	34055x-01
HR 510	ID 1120313-xx/1119971-xx	81762x-16	81762x-18	34059x-01	34056x-01/73498x-01/81760x-01	34055x-01
HR 510 FS	ID 1119974-xx/1120311-xx	81762x-16	81762x-18	34059x-02	34056x-02/73498x-02/81760x-01	34055x-06
HR 520	ID 670303-xx/670302-xx	81762x-16	81762x-18	34059x-02	34056x-04/73498x-02/81760x-01	34055x-06
HR 520 FS	ID 670305-xx/670304-xx	81762x-16	81762x-18	34059x-02	34056x-04/73498x-02/81760x-01	34055x-06
HR 550 FS	ID 1183021-xx/1200495-xx	81762x-16	81762x-18	34059x-02	34056x-04/73498x-02/81760x-01	34055x-06
HRA 551 FS for HR 550 FS	ID 1119052-xx	81762x-16	81762x-18	34059x-02	34056x-04/73498x-02/81760x-01	34055x-06
Installation by the machine manufacturer						
For more information, see the *Information for Machine Manufacturers* 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.

For more information, see the *Touch Probes* brochure



SE 660/ SE 661



TS 460


TS 260

Workpiece touch probes	**TNC7** Minimum NC SW	**TNC7 basic** Minimum NC SW	**TNC 640** Minimum NC SW	**TNC 620** Minimum NC SW	**TNC 320** Minimum NC SW
TS 260 ID 738283-xx	81762x-16	81762x-18	34059x-05	81760x-02	771851-02
TS 460 ID 1178530-xx	81762x-16	81762x-18	34059x-05	81760x-02	77185x-02
TS 740 ID 573757-xx	81762x-16	81762x-18	34059x-01	34056x-03/73498x-01/81760x-01	340551-05/771851-01
TS 760 ID 1283267-xx	81762x-16	81762x-18	34059x-11 SP5/ 34059x-12 SP5/ 34059x-16	81760x-07 SP5/81760x-08 SP5/81760x-16	–
TS 642 ID 653217-xx	81762x-16	81762x-18	34059x-09	81760x-06	77185x-06
Installation by the machine manufacturer					
For more information, see the *Touch Probes for Machine Tools* brochure					

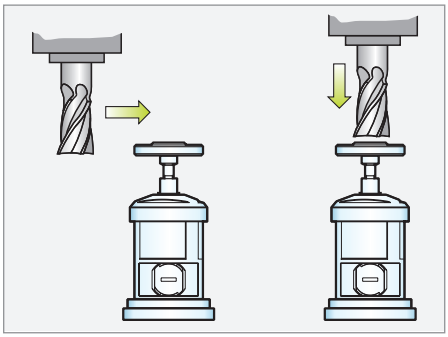
Hardware enhancements
TT tool touch probes

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.

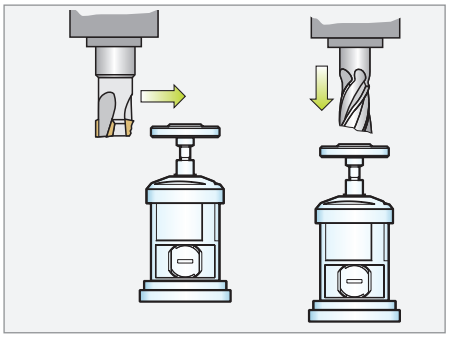
These touch probes are installed directly within the machine's workspace, where they enable tool measurement either before machining or during interruptions.

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** employs 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	**TNC7 basic** 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	81762x-18	34059x-05	81760x-02	771851-02
TT 460	ID 1192582-xx	81762x-16	81762x-18	34059x-05	81760x-02	77185x-02
Installation by the machine manufacturer						
For more information, see the *Touch Probes for Machine Tools* brochure						

Hardware enhancements
VT 121/VT 122 vision systems for tool inspection

Vision systems from HEIDENHAIN
The VT 121 and VT 122 vision systems enable tool imaging within a machine tool's work envelope. The systems consist of two components:
- A camera with two objectives (VT 121, VT 122)
- The Visual Tool Check software (VTC)

As part of the tool inspection, the non-contacting cameras take close-up images of each flute as well as detailed panoramic images of the entire tool circumference. The tools can be imaged from the side and also from below. The VT 122 camera also enables tool measurement.

Comparison of VT 121 and VT 122
The VT 121 camera scores with its particularly compact design and is ideal for visual tool inspection. If tool measurement is also required, then the VT 122 is recommended. It features an integrated LED for measuring both the length and diameter of tools. The VT 122 is available in two variants:
- Standard version with connections for cable routing through the housing end
- Space-saving version with connections through the base plate

Both camera models have defined probing surfaces for simple and automated setup using a touch probe. This way, its position in the work envelope can be ascertained precisely for optimum use in the VTC software.

Together with the VTC software and the corresponding cycles in the control software, tool images can be generated quickly and easily. The TNC7 from HEIDENHAIN offers cycles for automatic tool inspection:
- Positioning of the tool
- Activation of the compressed air supply
- Capturing the images

The images are transferred via Gigabit Ethernet to a PC or IPC, and can also be viewed on the control's screen (Remote Desktop).

Actual tool inspection and subsequent evaluations can then easily by performed on the control's touchscreen. The VTC software provides an inspection overview feature that makes it easy to inspect tools with a large number of flutes, allowing you to navigate through the image series with a zoom window for greater detail.

Advantages of the vision systems
- Compact dimensions and durable camera design
- Brief and spot-on use of compressed air is sufficient
- Tool remains clamped in the spindle
- Images generated automatically within the machine tool
- High-resolution images
- Software with simple touch operation

The compact and highly durable cameras are designed to be installed inside the machine's work envelope. They can be employed regardless of whether cooling lubricant is used or dry machining is performed.

The tools and cover glasses are cleaned using compressed air from the integrated nozzles. The compressed air is needed only shortly before the image is taken. This suffices for reliable removal of chips and liquids in the area being imaged, permitting useful and meaningful tool inspection.



VT 121



VT 122

Vision system Tool Inspection		**TNC7** Minimum NC SW	**TNC7 basic** Minimum NC SW	**TNC 640** Minimum NC SW	**TNC 620** Minimum NC SW	**TNC 320** Minimum NC SW
VT 121	1249466-01	81762x-16	81762x-18	34059x-10	–	–
VT 122	1373589-xx	81762x-18	81762x-18	34059x-10	–	–
Installation by the machine manufacturer						
For more information, see the *VT 121/VT 122 Cameras for Tool Inspection within Machine Tools* Product Information document						

Inspection within the machine tool
The HEIDENHAIN vision systems inspect tools quickly and effortlessly. Generating these tool images before critical machining steps has nearly no influence on a machine's productive time.

Thanks to these high-resolution images and the possibility of the tool remaining in the spindle, analyses can now be performed that previously required a large amount of effort:
- Optimization of cutting parameters
- Improvement of NC programs

After a tool exceeds its theoretical service life or total distance traveled, visual inspection can provide sound reasons for whether to continue using the tool—thereby effectively contributing to a reduction in tool costs. Furthermore, the vision system is also used for breakage detection, even for small flute damage.

Tool wear
Tools are subject to heavy loads while machining, meaning that wear is unavoidable. This leads to dimensional deviations, poor surface quality, increased cutting forces, more power being required, and, in extreme cases, to tool breakage.

A tool's condition is usually assessed through the volume of chips it has cut, its service life, or its distance traveled. Factors like the workpiece material, machining parameters, and lubricant are often largely ignored. Whether the standard assessments are appropriate in any specific instance can practically be determined only with great effort.

By being able to mount the VT 121 and VT 122 cameras within the work envelope, and by using the control's comprehensive package of cycles, checking the tool condition is now fast and easy: The cameras feature a resolution of 1.3 megapixels with large zoom factors, so that even very small indications of wear are visible. The VTC software doesn't just present the wear visually; it also makes it directly measurable on the control's screen.

AI-supported wear measurement
With the software option "Automated flank-wear measurement" for the VTC software, the flank wear that commonly occurs especially during machining with an end mill can now be automatically tracked. The software was trained at HEIDENHAIN using a very comprehensive image gallery, so that it can evaluate cutting edges and automatically ascertain the average and maximum values of the flank-wear width. Even irritants like chips and droplets on the flutes are recognized and factored out during the evaluation.

The system provides precise data on the wear already just a couple seconds after generating the images—completely without manual intervention. The measuring accuracy is typically ±20 µm. If adaptable limit values are exceeded, a warning notice or disabling notice can be generated. This could lead, for example, to an end mill in the TNC7 being blocked from further use. This ensures safe tool use even in highly automated operations, such as during night shifts.

The simple and rapid generation of images also enables documentation of the wear over time and clearly estimating any future possible use of the tool. All recorded data are available in CSV files for subsequent analysis and reporting.



Hardware enhancements
The ITC additional operating stations

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 860
As the
"Extended Workspace"

Additional operating station		TNC7 Minimum NC SW	TNC7 basic Minimum NC SW	TNC 640 Minimum NC SW	TNC 620 Minimum NC SW	TNC 320 Minimum NC SW
ITC 342	ID 1354570-xx	–	81762x-18	–	–	–
ITC 352	ID 1374639-xx	81762x-18	–	–	–	–
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 Machine Manufacturers* brochures						

Efficient proving out of NC programs
OC 310 override controller

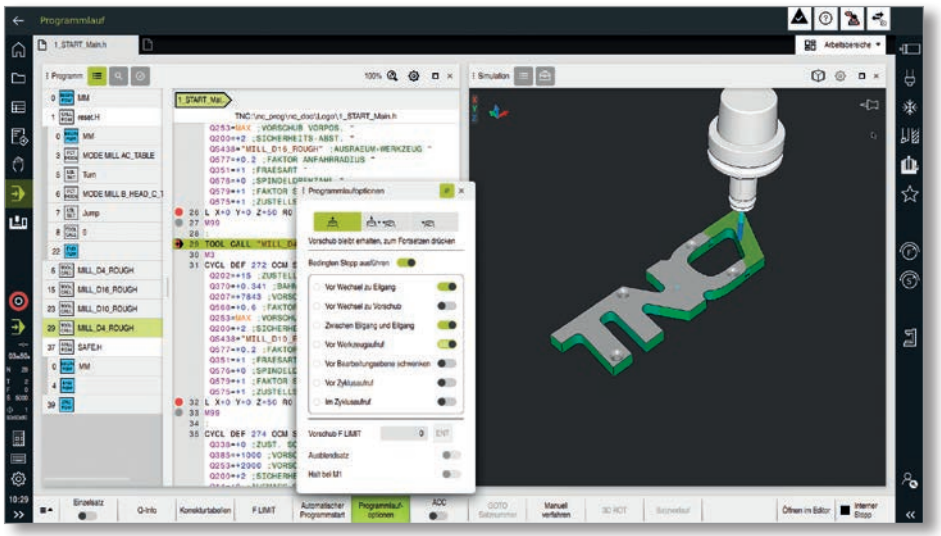
Proving out a new NC program for a workpiece requires much time and concentration. This procedure becomes much easier and more reliable with the following enhancements:
- Conditional stops using breakpoints
- OC 310 override controller



OC 310 override controller
The OC 310 is an innovative operating device. It replaces the feed-rate and rapid-traverse potentiometers on the TNC7 operating panel. The OC 310 is endlessly rotatable, and features a multicolor LED ring and an illuminated NC Start key. An integrated vibration motor gives haptic feedback to the user regarding specific events, such as when switching to the following:
- Minimum feed rate
- Maximum feed rate
- 100% feed rate

An integrated and intelligent panic function detects a rapid turning down and automatically sets the feed-rate value to 0.

Conditional stops using breakpoints
When proving out a new program, it is critical that the machine stops at the right time. The TNC7 allows you to define events triggering a conditional stop. For example, the switching from machining feed rate to rapid traverse or the calling of a new tool can be an event for a conditional stop. On the control, you select the events that will trigger a program stop. In addition you can specify how to resume program run after the conditional stop, such as by pressing the OC 310 override controller or by turning it clockwise.



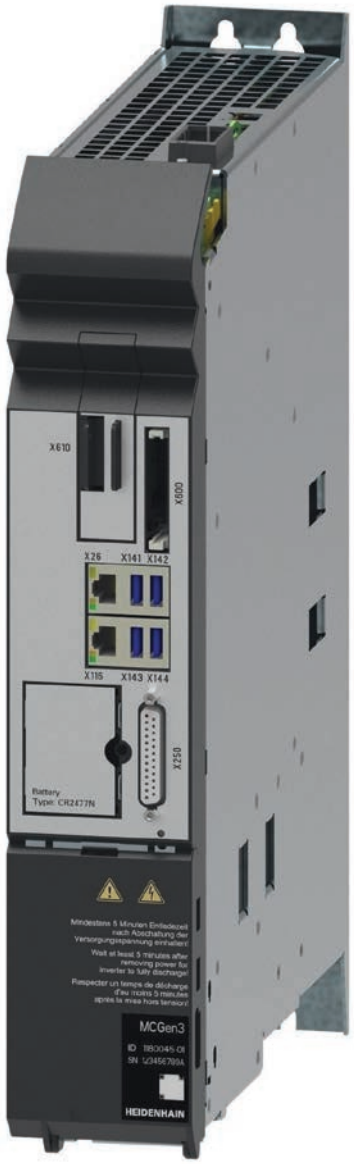
Requirements	TNC7		TNC7 basic	
NC software	NC SW 81762x-18 and higher		NC SW 817621-18 and higher	
Suitable keyboard units	TE 350 TE 350 FS TE 361 TE 361 FS	ID 1370209-02 and higher ID 1370220-02 and later ID 1313011-03 and higher ID 1326583-03 and higher	TE 340 TE 340 FS	ID 1320800-02 and higher ID 1352798-02 and higher
Suitable machine operating panels	MB 350 MB 350 FS	ID 1372719-xx ID 1374704-xx	MB 340 MB 340 FS	ID 1388531-xx ID 1388532-xx
Installation by the machine manufacturer				
For more information, see the *Information for Machine Manufacturers* brochures				

Hardware enhancements
The IPC industrial computer

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. The **Remote Desktop Manager** software option 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 Windows 8 or 10 operating systems can be installed on this empty data medium.



IPC 306

Controls		**TNC 640**	**TNC 620**
Industrial PC		Required: 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 *Information for Machine Manufacturers* brochures			

Hardware enhancements
Tool breakage detector for machine tools

This sensor enables fast and reliable non-contact tool breakage inspection at rapid traverse, thereby ensuring exceptional process reliability. It still works when exposed to coolant and other contaminants, making it ideal in challenging manufacturing environments. The tool breakage detector can be used for tools with diameters as small as 0.4 mm made of HSS or carbide and can be used on all common machine tool controls regardless of make.



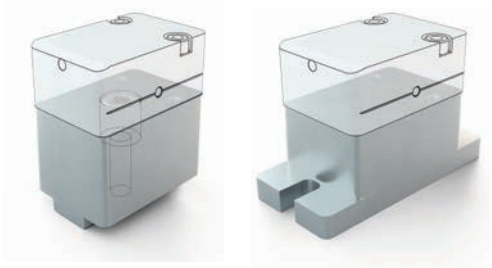
Application
The tool breakage detector can be positioned in the work envelope such that it is easily accessible during tool changes at rapid traverse. The inductive sensor can detect tools as they pass by while they are spinning at their operational speed, or at least at 1000 rpm. Thanks to its rapid scanning technology, even very small tools made of HSS steel or carbide are detected. For use on all controls, the breakage sensor generates a floating trigger signal and an HTL switching signal. The control can evaluate these signals via standardized instruction sets. In addition, an LED indicates whether a tool has been detected. As a result, machines already in use can benefit from automated breakage detection as well.

Advantages
- Non-cutting time for breakage inspection is reduced by up to six seconds.
- The wear-free TD 110 is unaffected by coolant and contamination.
- Tools do not need to be cleaned for reliable tool inspection.
- Reduction of system costs and carbon emissions due to not needed purge air (unlike laser systems).

Connecting to TNC controls
The tool breakage detector can be connected via the cables for the TT 160 touch probe to the touch probe input X112/X113 of the PLB, UEC, or UMC, and to X12/X13 of the iTNC 530.
Our Service team provides optimal support for the operation of HEIDENHAIN controls, including cycles for breakage detection that can be added via TNCremo. Please contact the HEIDENHAIN Service team:
Remote support
service.nc-pgm@heidenhain.de
ID 1369787-35

Connection to other CNC controls
The HTL switching signals and the floating outputs can be used for connection to the PLC or to other IO connections of nearly every CNC control (assistance from the machine manufacturer may be necessary). For information about programming cycles, refer to the TD 110 documentation.

Fast and versatile mounting
Installing the tool breakage detector at the location where tool changes occur enables fast changing times and a maximum reduction in non-cutting time. The different detection approaches permit various configurations for installation in the work envelope. In the simplest case, the TD 110 is mounted at the edge of the machine table.



Mounting examples using a base provided by the customer

Tool breakage detector		**TNC7** Minimum NC SW	**TNC7 basic** Minimum NC SW	**TNC 640** Minimum NC SW	**TNC 620** Minimum NC SW	**TNC 320** Minimum NC SW
TD 110	1377561-01	817620-16	817620-18	34059x-07	81760-04	771851-04
Installation by the machine manufacturer						
For more information, see the *TD 110* Product Information document, and visit www.heidenhain.com						

HEIDENHAIN

Mastering nanometer accuracy



HEIDENHAIN

DR. JOHANNES HEIDENHAIN GmbH
Dr.-Johannes-Heidenhain-Straße 5
83301 Traunreut, Germany
☏ +49 8669 31-0
FAX +49 8669 32-5061
info@heidenhain.de

www.heidenhain.com

HEIDENHAIN worldwide

827222-2C · 02/2026 · CD · Printed in Germany