HEIDENHAIN

Options and Accessories

For TNC Controls

07/2019

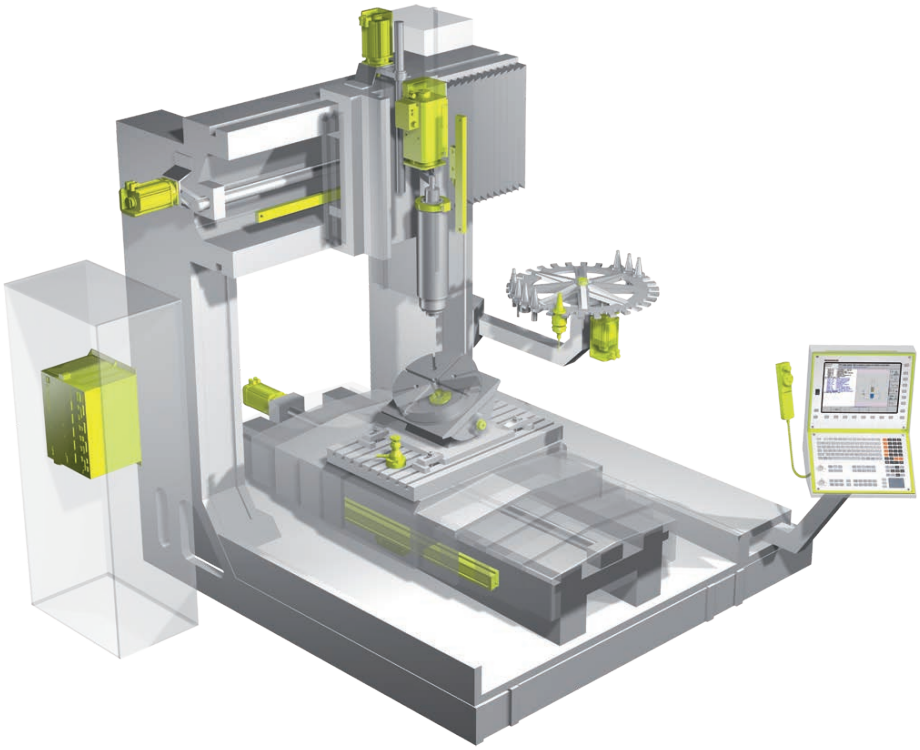
Options and accessories for TNC controls

HEIDENHAIN controls are known for their complete range of features and comprehensive equipment. In addition, they can be optimally adapted to the respective application thanks to a series of control options and accessories. This brochure provides you with a comprehensive overview that is independent of the control version.

Options are functions that are integrated in the control and that allow you to tailor the feature range of the TNC to your actual requirements, including at a later time. Some options have to be adapted by the machine tool builder. The options are enabled with a simple keyword.

HEIDENHAIN provides you with useful tools for applications outside the TNC as **PC software**, ranging from software for supporting the transfer of data or for creating a PLC program, all the way to the complete NC programming station.

With the **hardware enhancements**, your work with the machine becomes faster, safer, and simpler. An electronic handwheel, for example, allows especially delicate traverse of the machine, and a workpiece touch probe reduces the time needed for setting up the workpiece.



Contents

Please also note the page references in the overview tables.

Overview

Option	Option number	ID	TNC 320	TNC 620	TNC 640	iTNC 530	Adaptation by OEM necessary	Page
Programming and operation								
Machining with a Rotary Table • Programming of cylindrical contours as if in two axes • Feed rate in mm/min or degrees/min	**8**	617920-01 *iTNC 530:* 367591-01	•	•	•	•	Yes	13
Coordinate Transformation Tilting of the working plane, PLANE function	**8**	617920-01 *iTNC 530:* 367591-01	•	•	•	•	Yes	14
Display Step down to 0.01 µm or 0.00001°	**23**	632986-01	–	•	•	–	No	–
Touch Probe Cycles • Compensation of workpiece misalignment, setting of presets • Automatic tool and workpiece measurement • Touch probe input enabled for non-HEIDENHAIN systems	**17**	634063-01	✔	•	✔	✔	Yes	15
Extended Programming Functions • FK free contour programming • Fixed cycles • Peck drilling, reaming, boring, counterboring, centering • Milling of internal and external threads • Clearing of level and oblique surfaces • Multi-operation machining of straight and circular slots • Multi-operation machining of rectangular and circular pockets • Cartesian and polar point patterns • Contour train, contour pocket—also with contour-parallel machining • Special cycles developed by the machine tool builder can be integrated • Engraving cycle: engraving of text or numbers in a straight line or on an arc • Contour slot with trochoidal milling	**19**	628252-01	✔	•	✔	✔	No	16
Program-Verification Graphics, Program-Run Graphics • Plan view • Projection in three planes • 3-D view	**20**	628253-01	✔	•	✔	✔	No	17
Finely Detailed 3-D View	**20**	628253-01	✔	•	✔	–	No	18
Pallet Editor	**22**	628255-01	–	•	✔	✔	Yes	19
DXF Converter: importing of contours and machining positions from DXF files	**42**	526450-01	•	•	•	•	No	21
CAD Import: importing of contours from 3-D models	**42**	526450-01	•	•	•	–	No	22
Turning Functions • Tool management for turning • Tool-tip radius compensation • Switching between milling and turning modes of operation • Lathe-specific contour elements • Package of turning cycles	**50**	634608-01	–	–	•	–	Yes	23
Eccentric Turning	**50**	634608-01	–	–	•	–	Yes	24

• = Available as option
– = Not available
✔ = Standard

Option	Option number	ID	TNC 320	TNC 620	TNC 640	iTNC 530	Adaptation by OEM necessary	Page
Programming and operation								
Synchronization of two or more spindles • Hobbing cycle	**131** **50**	806270-01	–	–	•	–	Yes	28
Expanded Tool Management	**93**	676938-01	•	•	•	•	Yes	26
Interpolating Spindle: interpolation turning	**96**	751653-01	–	–	•	•	Yes	27
CAD Viewer: opening of 3-D CAD data directly on the TNC	**98**	800553-01	✔	✔	✔	•	No	29
Batch Process Manager: easy-to-understand depiction of the pallet management	**154**	1219521-01	–	•	•	–	Yes	20
Gear Cutting: functions for the production of gears	**157**	1237235-01	–	–	•	–	Yes	30
Advanced Function Set Turning: extended turning cycles and turning functions	**158**	1237237-01	–	–	•	–	Yes	25
Machine accuracy								
KinematicsOpt: touch probe cycles for automatic measurement of rotary axes	**48**	630916-01	–	•	•	•	No	31
KinematicsComp: 3-D spatial compensation	**52**	661879-01	–	–	•	•	Yes	32
Cross Talk Compensation (CTC): compensation of position error through axis coupling	**141**	800542-01	–	•	•	•	Yes	33
Position Adaptive Control (PAC): position-dependent adaptation of control parameters	**142**	800544-01	–	•	•	•	Yes	34
Load Adaptive Control (LAC): load-dependent adaptation of control parameters	**143**	800545-01	–	•	•	•	Yes	35
Motion Adaptive Control (MAC): motion-dependent adaptation of control parameters	**144**	800546-01	–	•	•	•	Yes	36
Active Vibration Damping (AVD): active vibration amplitude reduction	**146**	800548-01	–	•	•	•	Yes	37

• = Available as option
– = Not available
✔ = Standard

Overview

Option	Option number	ID	TNC 320	TNC 620	TNC 640	iTNC 530	Adaptation by OEM necessary	Page
Machining functions								
Interpolation: circular in 3 axes with tilted working plane	**8**	617920-01 *iTNC 530:* 367591-01	•	•	•	•	No	14
Interpolation: linear in 5 axes	**9**	617921-01 *iTNC 530:* 367590-01	–	•	•	•	No	–
Spline Interpolation: processing of third-degree polynomials	**9**	367590-01	–	–	–	•	No	–
5-Axis Simultaneous Machining • 3-D tool compensation through surface normal vectors • Use of the electronic handwheel to change the angle of the swivel head during program run without affecting the position of the tool point (TCPM = Tool Center Point Management) • Keeping the tool normal to the contour • Tool radius compensation perpendicular to the tool direction • Manual traverse in the active tool-axis system	**9**	617921-01 *iTNC 530:* 367590-01	–	•	•	•	Yes	38
Handwheel Superimpositioning: superimposing handwheel positioning during program run	**21**	628254-01	–	•	✔	✔	Yes	39
Tool Compensation: radius-compensated contour precalculation (LOOK-AHEAD)	**21**	628254-01	✔	•	✔	✔	No	40
Dynamic Collision Monitoring (DCM)	**40**	526452-01	–	–	•	•	Yes	41
Global Program Settings	**44**	576057-01	–	–	•	•	Yes	42
Adaptive Feed Control (AFC)	**45**	579648-01	–	–	•	•	Yes	43
3D-ToolComp: 3-D radius compensation depending on the tool's inclination angle (only with option 9)	**92**	679678-01	–	–	•	•	No	45
Active Chatter Control (ACC): active suppression of tool chatter	**145**	800547-01	–	•	•	•	Yes	46
Visual Setup Control (VSC): camera-based monitoring of the setup situation	**136**	1099457-01	–	–	•	–	Yes	47
Component Monitoring: monitoring for component overloading and wear	**155**	1226833-01	–	•	•	–	Yes	44
Communication								
HEIDENHAIN-DNC: communication with external Windows applications over COM component	**18**	526451-01	•	•	•	•	Yes	48
Remote Desktop Manager: display and remote operation of external computer units (e.g., Windows PC)	**133**	894423-01	•	•	•	•	Yes	49
State Reporting: provision of operating conditions over the State Reporting Interface (SRI)	**137**	1232242-01	•	•	•	–	No	50

• = Available as option
– = Not available
✔ = Standard

Option		Option number	ID	TNC 320	TNC 620	TNC 640	iTNC 530	Adaptation by OEM necessary	Page
Interfacing to the machine									
Additional Control Loops	Additional axis 1	**0**	354540-01	•	•	•	•	Yes	51
	Additional axis 2	**1**	353904-01	•	•	•	•	Yes	
	Additional axis 3	**2**	353905-01	–	–	•	•	Yes	
	Additional axis 4	**3**	367867-01	–	–	•	•	Yes	
	Additional axis 5	**4**	367868-01	–	–	•	•	Yes	
	Additional axis 6	**5**	370291-01	–	–	•	•	Yes	
	Additional axis 7	**6**	370292-01	–	–	•	•	Yes	
	Additional axis 8	**7**	370293-01	–	–	•	•	Yes	
	4 Additional Control Loops	**77**	634613-01	–	–	•	•	Yes	
	8 Additional Control Loops	**78**	634614-01	–	–	•	•	Yes	
Synchronized Axes: gantry axes, tandem tables		**24**	634621-01	•	•	✔	✔	Yes	52
Python OEM Process: realization of special functions		**46**	579650-01	•	•	•	•	Yes	53
Double Speed: short control-loop cycle times for direct drives		**49**	632223-01	–	•	•	•	Yes	54
OEM Option		**101 – 130**	579651-01 – 579651-30	–	–	•	•	Yes	55
Real-Time Coupling (RTC): function for synchronizing axes and spindles		**135**	1085731-01	–	–	•	–	Yes	56

• = Available as option
– = Not available
✔ = Standard

Overview

PC software		TNC 320	TNC 620	TNC 640	iTNC 530	Adaptation by OEM necessary	Page
StateMonitor		•	•	•	•	No	59
TNCremo		•	•	•	•	No	57
TNCremoPlus		•	•	•	•	No	57
TeleService		•	•	•	•	Yes	58
RemoTools SDK		•	•	•	•	Yes	48
virtualTNC		–	–	•	•	Yes	60
PLCdesign		•	•	•	•	Yes	61
KinematicsDesign		•	•	•	•	Yes	62
CycleDesign		•	•	•	•	Yes	63
TNCscope		•	•	•	•	Yes	64
DriveDiag		–	•	•	•	Yes	65
TNCopt		–	•	•	•	Yes	66
IOconfig		–	•	•	•	Yes	67
TNCkeygen		•	•	•	•	Yes	68
BMXdesign		•	•	•	•	Yes	69
FixtureWizard		–	–	–	•	Yes	70
Programming station	**Single-user license for demo version**	•	•	•	•	No	71
	Single-user license with TNC keyboard	•	•	•	•	No	
	Single-user license with virtual keyboard	•	•	•	•	No	
	Network license with virtual keyboard For 1 workstation	•	•	•	•	No	
	Network license with virtual keyboard For 14 workstations	•	•	•	•	No	
	Network license with virtual keyboard For 20 workstations	•	•	•	•	No	

• = Available as option
– = Not available
✔ = Standard

Hardware enhancements			ID	TNC 320	TNC 620	TNC 640	iTNC 530	Adaptation by OEM necessary	Page
Handwheel	**HR 130** TTL; panel mounted	With detent W/o detent	540940-01 540940-03	•	•	•	•	No	72
	HR 510 portable handwheel	With detent W/o detent	1120313-xx 1119971-xx	•	•	•	•	No	
	HR 510 FS portable handwheel	With detent W/o detent	1119974-xx 1120311-xx	•	•	•	•	No	
	HR 520 portable handwheel with display	With detent W/o detent	670303-xx 670302-xx	•	•	•	•	No	
	HR 520 FS portable handwheel with display	With detent W/o detent	670305-xx 670304-xx	•	•	•	•	No	
	HR 550 FS portable wireless handwheel with display	With detent W/o detent	1183021-xx 1200495-xx	•	•	•	•	No	
	HRA 551 FS handwheel holder for HR 550 FS		1119052-xx	•	•	•	•	No	
Workpiece touch probe	**TS 260** with cable		738283-xx	•	•	•	•	Yes	73
	TS 460 with radio or infrared transmission		1178530-xx	•	•	•	•	Yes	
	TS 642 with infrared transmission		653217-xx	•	•	•	•	Yes	
	TS 740 with infrared transmission		573757-xx	•	•	•	•	Yes	
Tool touch probe	**TT 160** with cable		729763-xx	•	•	•	•	Yes	74
	TT 460 with radio or infrared transmission		1192582-xx	•	•	•	•	Yes	
Additional operating station	**ITC 755** with touchscreen and ASCII keyboard		1039527-xx	–	–	•	•	Yes	75
	ITC 750 15-inch screen; separate TNC keyboard required		1039544-xx	–	•	•	•	Yes	
	ITC 860 19-inch touchscreen; separate TNC keyboard required		1174935-xx	–	–	•	–	Yes	
Industrial PC	**IPC 6641** for Windows interface on TNC with 8 GB of RAM with 16 GB of RAM		 1039543-01 1039543-02	–	•	•	•	Yes	76
Camera system	**VS 101** for workspace monitoring		1137063-01	–	–	•	–	Yes	77

• = Available as option
– = Not available
✔ = Standard

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. They improve the machine's dynamic performance, attain higher accuracy at the TCP, and therefore permit milling at the limit of the technologically possible, regardless of the age of the machine, its load, and the machining position. And all this is done without any modification of the machine's mechanics.

It's no longer necessary to work slowly in order to achieve accurate parts with high surface definition. With Dynamic Precision, machine tools work **with speed and precision** at the same time.

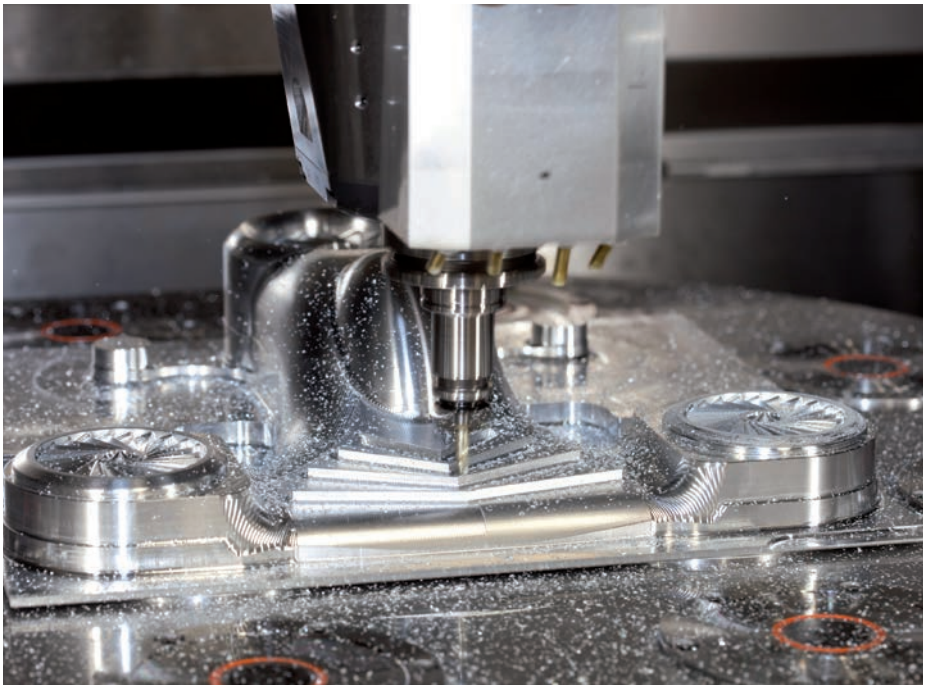
High precision together with fast machining is the basis for an increase in productivity. Unit costs are reduced without compromises in accuracy and surface quality. Dynamic Precision also ensures that accuracy is retained regardless of operating time and weight. It isn't necessary to reduce feed rates due to age or load.

The functions of Dynamic Precision are available as options for controls from HEIDENHAIN. The machine tool builder can apply them individually as well as in combination.

- CTC: compensation of acceleration-dependent position errors at the tool center point, thereby increasing accuracy in acceleration phases
- AVD: active vibration damping for better surfaces
- PAC: position-dependent adaptation of control parameters
- LAC: load-dependent adaptation of control parameters enhances accuracy regardless of load and age
- MAC: motion-dependent adaptation of control parameters

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

Because Dynamic Precision consists of software functions, it requires no intervention in the mechanics of the machine or in its power train. However, the machine manufacturer must enable the individual functions, enter their parameters, and adapt them to the machine.



	TNC 640	iTNC 530	TNC 620	Page
Dynamic Precision	✔	✔	✔	
CTC: compensation of position errors through mechanical compliance	Option	Option	Option	33
AVD: active vibration damping	Option	Option	Option	37
PAC: position-dependent adaptation of control parameters	Option	Option	Option	34
LAC: load-dependent adaptation of control parameters	Option	Option	Option	35
MAC: motion-dependent adaptation of control parameters	Option	Option	Option	36
Installation by the machine tool builder				

For more information, see the Technical Information on *Dynamic Precision* 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. The software functions support the machine operator but also make the manufacturing process itself faster, more stable, and more predictable—in short, more efficient. Dynamic Efficiency helps to increase the metal removal rate and reduce machining time.

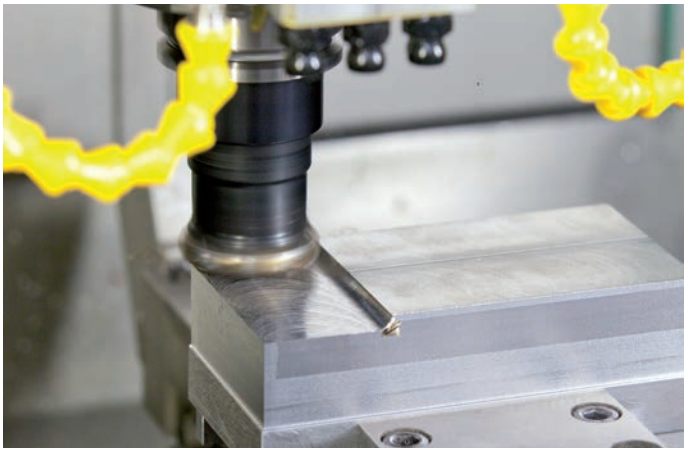
Dynamic Efficiency comprises three software functions:

- ACC: reduces chatter tendencies and permits higher feed rates and greater infeeds
- AFC: controls the feed rate depending on the machining situation
- Trochoidal milling: a function for the roughing of slots and pockets that eases the load on the tool

The AFC and ACC functions are later described in detail. They are identified with the Dynamic Efficiency logo.

Each solution by itself offers decisive advantages in the machining process. But it is particularly the combination of these TNC features that exploits the potential of the machine and the tool while simultaneously reducing the mechanical load. Changing machining conditions, such as interrupted cuts, various material plunging procedures, and simple clear-out operations are also examples of why these features are worthwhile. In practice, removal rate increases of 20 to 25 percent are possible.

Dynamic Efficiency permits higher removal rates and therefore increases productivity without making the user resort to special tools. The avoidance of tool overload and premature tool wear, as well as the additional gain in process reliability, contribute considerably to improved economic efficiency..



	TNC 640	iTNC 530	TNC 620	Page
Dynamic Efficiency	✔	✔	–	
Active Chatter Control (ACC)	Option	Option	Option	46
Adaptive Feed Control (AFC)	Option	Option	–	43
Trochoidal milling	Standard	Standard	Standard	–

Installation by the machine tool builder/user

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

Connected Machining
Solutions for networked manufacturing

connected + machining

With its Connected Machining package of functions, HEIDENHAIN offers solutions for the customized networking of production processes. These solutions place the user at the center of digital job management through the control on his milling machine or lathe. The HEIDENHAIN control is networked with all of the production-related areas of the company on a highly individual basis, with tailoring to already existing structures, and is open for future developments.

You benefit from workload reduction thanks to simple data usage, time-saving workflows, and transparent processes in all areas of your company—not only in the shop but also in the areas of design and production planning, as well as in management, logistics, and servicing, etc. The manufacturing strengths of modern machines and plants are thus supplemented by the uniformly digital job management of Connected Machining.

Even a simple TNC control system integrated into the company network over Ethernet offers, with its standard functions, a wide range of possibilities for receiving and using digital job data directly on the control:

- PDF viewer (e.g., for displaying fixture diagrams and design drawings)
- CAD Viewer (e.g., for displaying 3-D models)
- Image display (e.g., for displaying fixture setups and production notes)
- Web browser for accessing web-based applications, such as ERP and MES clients, e-mail clients, and HTML5 clients

Beyond this, the following solutions and options are offered:

- The **StateMonitor** software makes processes transparent through data acquisition and data analysis
- **Remote Desktop Manager** (option 133) allows access to PCs and the installed software right at the control
- The **HEIDENHAIN DNC interface** (option 18) connects the control to enterprise-resource planning and production-activity control systems
- The **Extended Workspace** display provides clear representation and processing of the job data on the machine

	TNC 640	iTNC 530	TNC 620	Page
Connected Machining	✔	✔	✔	
StateMonitor	✔	✔	✔	59
Remote Desktop Manager	Option	Option	Option	49
HEIDENHAIN DNC	Option	Option	Option	48
Extended Workspace	✔	–	–	75

Installation by the machine tool builder

For more information, see the *Connected Machining* brochure

Programming and operation
Machining with a Rotary Table

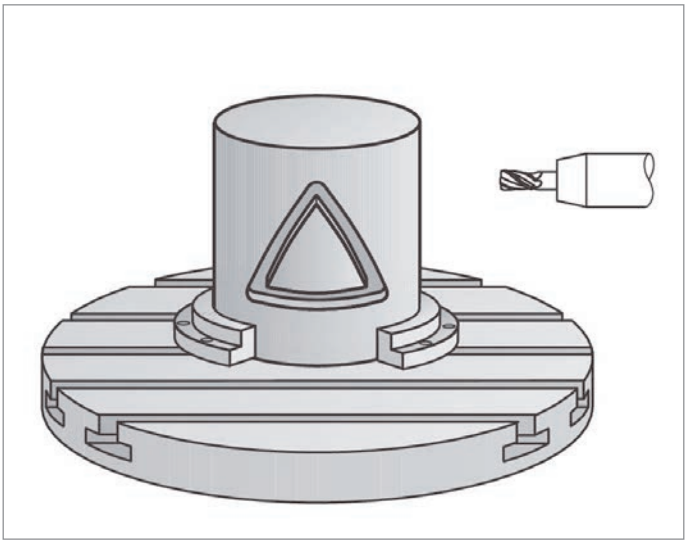
Many 5-axis operations that, at first glance, may seem very complex can be reduced to conventional 2-D movements that are merely wrapped around a cylindrical surface. The TNC supports you with application-oriented functions to help you write and edit such programs quickly and simply without a CAM system.

Cylinder surface machining
With the TNC, it is easy to program contours (consisting of straight lines and arcs) on cylindrical surfaces using rotary and tilting tables: you simply program the contour in a plane as if the cylinder surface were unrolled (regardless of the axis on the TNC 640, TNC 620, and TNC 320). The TNC then calculates and machines the corresponding cylindrical contour.

The TNC features four cycles for cylindrical surface machining:
- Slot milling (the slot width is the same as the tool diameter)
- Guide-groove milling (the slot width is greater than the tool diameter)
- Ridge milling
- Outside contour milling

Feed rate for rotary axes and tables in mm/min
By default, the feed rate of rotary axes is programmed in degrees/min. However, the TNC can also interpret this feed rate in mm/min. The feed rate at the contour is then independent of the distance of the tool center from the center of the rotary axis.

User administration
The user administration of the TNC 640 lets you specify various roles and access rights for users. Each user can then operate only within his assigned rights. This can prevent an unintentional or unauthorized deleting of files or contents of system files. Additionally, many functions can be accessed only if the user has the corresponding rights. The user administration of the TNC 640 thus not only provides increased data protection, but also increases machine operating safety.



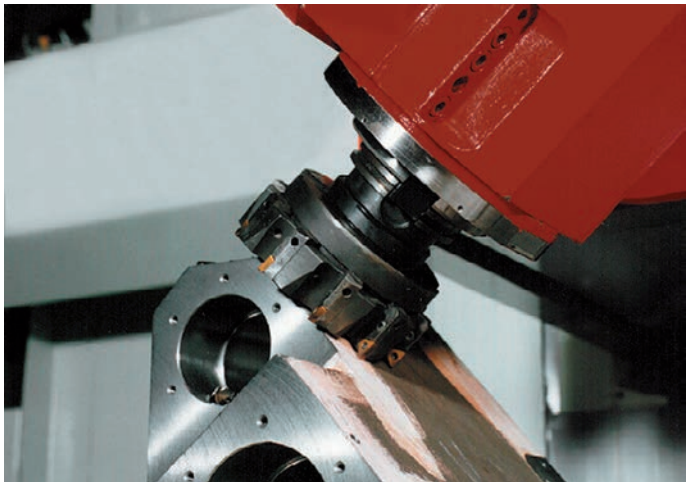
Machining with a Rotary Table TNC 640 HSCI/TNC 620 HSCI/TNC 320 iTNC 530 HSCI/iTNC 530	Option 8	 ID 617920-01 ID 367591-01
TNC 640 HSCI	As of NC SW 34059x-01	
TNC 620 HSCI	As of NC SW 34056x-01/73498x-01/81760x-01	
TNC 320	As of NC SW 34055x-01/771851-01	
iTNC 530 HSCI	As of NC SW 60642x-01	
iTNC 530	As of NC SW 34049x-01	
Installation by the machine tool builder		
For more information, see the TNC brochures and visit *www.klartext-portal.com*		

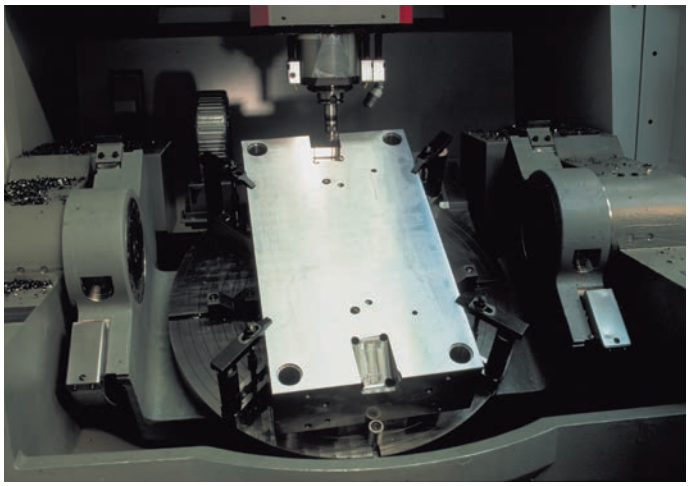
Programming and operation
Coordinate Transformation: tilting of the working plane, PLANE function

Programs for contours and holes on inclined surfaces are often very complex and require time-consuming computing and programming work. Here the TNC helps you to save a great deal of programming time. You program the machining operation as usual in the main plane—for example, in X/Y. The machine then runs the program in a plane that has been tilted by one or more rotary axes with respect to the main plane.

The PLANE feature makes it easy to define a tilted working plane; you can specify tilted working planes in seven different ways, depending on the information in the workpiece drawing. In order to keep the use of these complex functions as simple as possible, a separate animation is available for each possible plane definition, so that you can view them before selecting the function. Clearly arranged support graphics assist you during input.

You can also use the PLANE function to define the positioning behavior for tilting so that there are no unpleasant surprises when the program is run. The settings for defining the positioning behavior are identical for all PLANE functions, making everything that much easier.





Coordinate Transformation TNC 640 HSCI/TNC 620 HSCI/TNC 320 iTNC 530 HSCI/iTNC 530	Option 8	 ID 617920-01 ID 367591-01
TNC 640 HSCI	As of NC SW 34059x-01	
TNC 620 HSCI	As of NC SW 34056x-01/73498x-01/81760x-01	
TNC 320	As of NC SW 34055x-01/771851-01	
iTNC 530 HSCI	As of NC SW 60642x-01	
iTNC 530	As of NC SW 34049x-01	
Installation by the machine tool builder		
For more information, see the TNC brochures and visit *www.klartext-portal.com*		

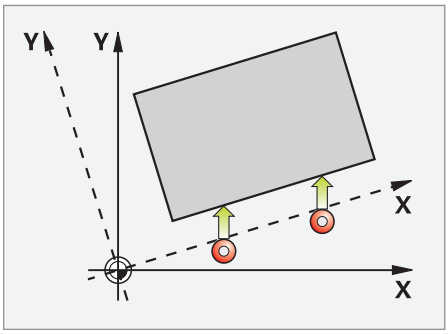
Programming and operation
Touch Probe Cycles

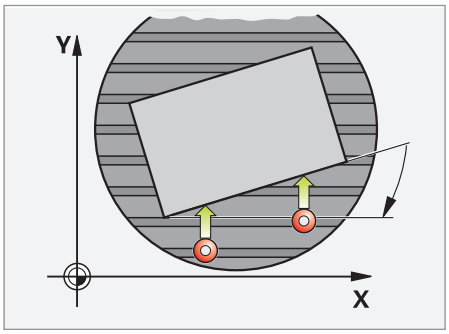
Workpiece alignment
With HEIDENHAIN touch probes and the probing functions of the TNC, you can forgo any tedious manual alignment of the workpiece:
- Clamp the workpiece in any position.
- The touch probe ascertains the workpiece misalignment by probing a surface.
- The TNC compensates the misalignment with a "basic rotation," meaning that the NC program runs as if the part has been rotated by the measured misalignment, or the rotary table itself is turned to correct the misalignment.
- The TNC offers manual, automatic, and semiautomatic cycles for correcting misalignments in two or three dimensions.

Preset setting
Fast and reliable definition of the preset reduces nonproductive time and increases machining accuracy. The TNC features a large number of probing cycles for the automatic setting of presets.

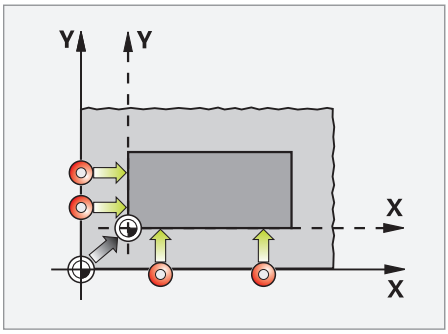
Workpiece inspection
The TNC features a number of measuring cycles for checking the geometry of the machined workpieces. This enables you to do the following:
- Recognize a workpiece and call an appropriate part program
- Check whether all machining operations were conducted correctly
- Detect and compensate for tool wear, etc.

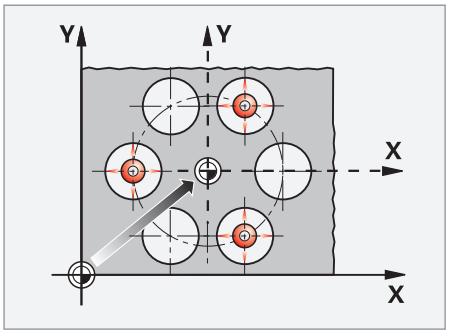
Tool measurement
Together with the TT touch probes for tool measurement, the TNC makes it possible to measure tools automatically while they are in the machine spindle. The TNC saves the ascertained values for tool length and radius in the central tool file. By inspecting the tool during machining, you can quickly and directly measure wear or breakage to prevent scrap or rework.



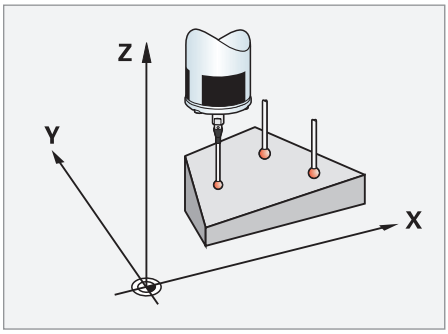


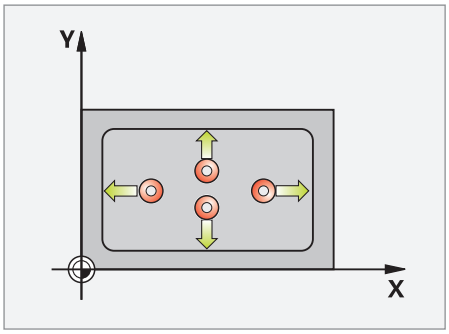
Compensating for workpiece misalignment by rotating the coordinate system or turning the table



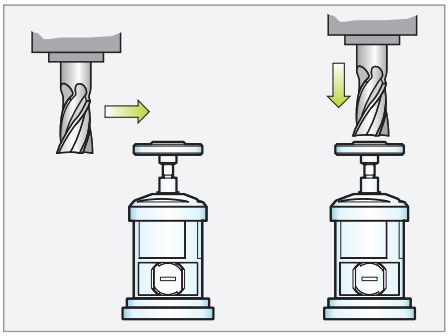


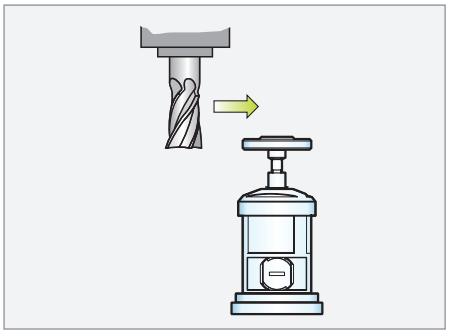
Setting a preset at a corner, for example, or in the center of a circular hole pattern





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





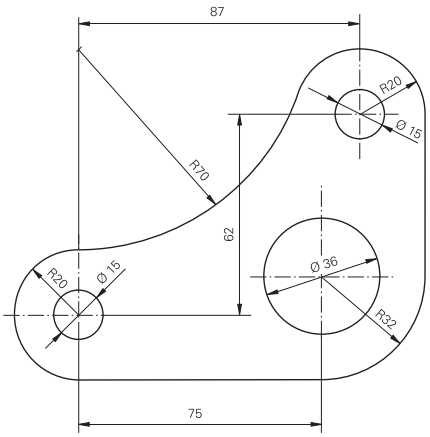
Tool measurement (e.g., tool length and radius or tool wear)

Touch Probe Cycles	Option 17	ID 634063-01
TNC 640 HSCI	Standard	
TNC 620 HSCI	As of NC SW 34056x-01/73498x-01/81760x-01	
TNC 320	Standard	
iTNC 530 HSCI/iTNC 530	Standard	
Installation by the machine tool builder		
For more information, see the TNC brochures and the *Touch Probes* brochure		

Programming and operation

Advanced Programming Functions: FK free contour programming, fixed cycles

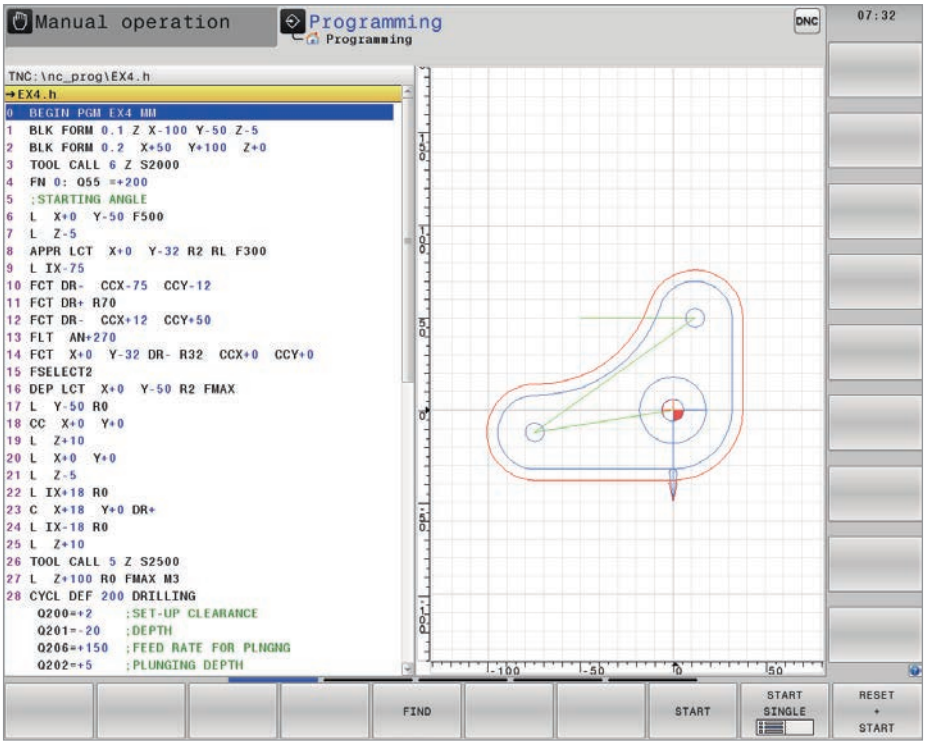
FK free contour programming
Not all workpieces are dimensioned for conventional NC programming. Thanks to FK, the control's free contour programming feature, in such cases you simply type in the data from the drawing—without first having to convert or calculate your data! It does not matter if individual contour elements are not completely defined as long as the complete contour has been. If the given data result in more than one mathematical solution, then the helpful TNC programming graphics will offer the possible variants for selection.



Standard cycles
Besides the fixed cycles for drilling and tapping (with or without floating tap holder), with option 19 there are cycles for thread milling, reaming, boring, and for hole patterns, as well as milling cycles for clearing plane surfaces and for roughing and finishing pockets, slots, and studs.

Cycles for complex contours
The Subcontour List (SL) cycles are particularly helpful for clearing pockets with combined contours. This term is used to identify machining cycles for pilot drilling, roughing, and finishing when the contour or subcontours are specified in subprograms. In this way, one contour description can be used for more than one operation using different tools.

OEM cycles
As original equipment manufacturers, machine tool builders can contribute their special manufacturing know-how by designing additional fixed cycles and saving them in the TNC. However, the end user can write his own cycles as well. HEIDENHAIN makes this possible with its PC program CycleDesign. CycleDesign enables you to organize the input parameters and soft-key structure of the TNC to suit your own needs.



Advanced Programming Functions	Option 19	ID 628252-01
TNC 640 HSCI	Standard	
TNC 620 HSCI	As of NC SW 34056x-01/73498x-01/81760x-01	
TNC 320	Standard	
iTNC 530 HSCI	Standard	
iTNC 530	Standard	

Installation by the user

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

Programming and operation
Program-Verification Graphics, Program-Run Graphics

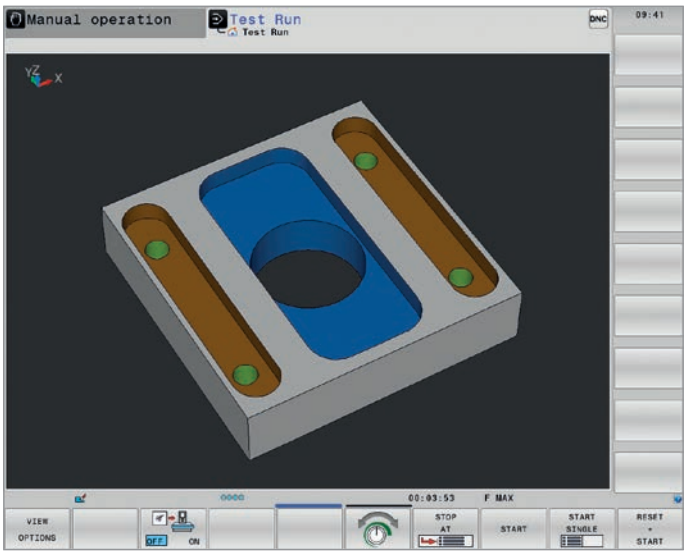
Programming graphics
HEIDENHAIN controls support you with detailed programming graphics. The programming graphics function is available as a standard feature on all controls and is described in the respective product brochures.

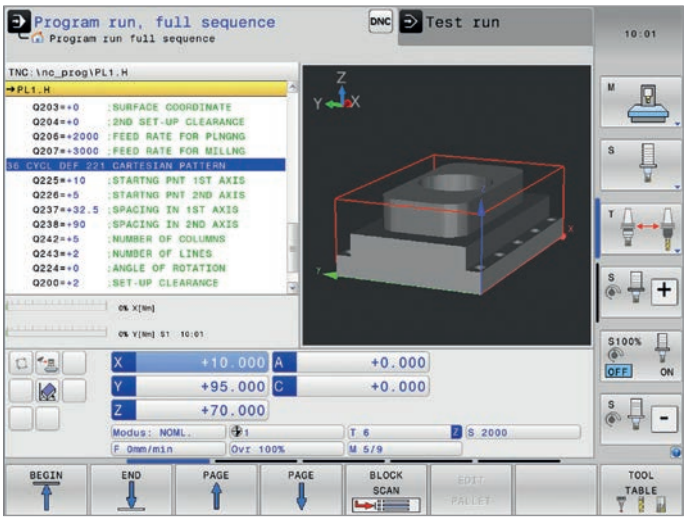
Various other graphic views are optional:

Test graphics
To be on the safe side before running a program, the TNC can graphically simulate the machining of the workpiece.
The TNC can display the simulation in the following ways:
- In plan view with different shades of depth
- In three projections (as in the workpiece drawing)
- In 3-D view

Details can be displayed with magnification. In addition, the TNC indicates the calculated machining time in hours, minutes, and seconds.

Program-run graphics
The TNC shows a real-time graphic of the machining progress. Coolant spray and protective enclosures usually obstruct any direct view of the actual workpiece. You can get around this with a simple keystroke to see the simulated progress of workpiece machining.





Program-Verification Graphics, Program-Run Graphics	Option 20	ID 628253-01
TNC 640 HSCI	Standard	
TNC 620 HSCI	As of NC SW 34056x-01/73498x-01/81760x-01	
TNC 320	Standard	
iTNC 530 HSCI	Standard	
iTNC 530	Standard	
Installation by the user		
For more information, see the TNC brochures and visit *www.klartext-portal.com*		

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

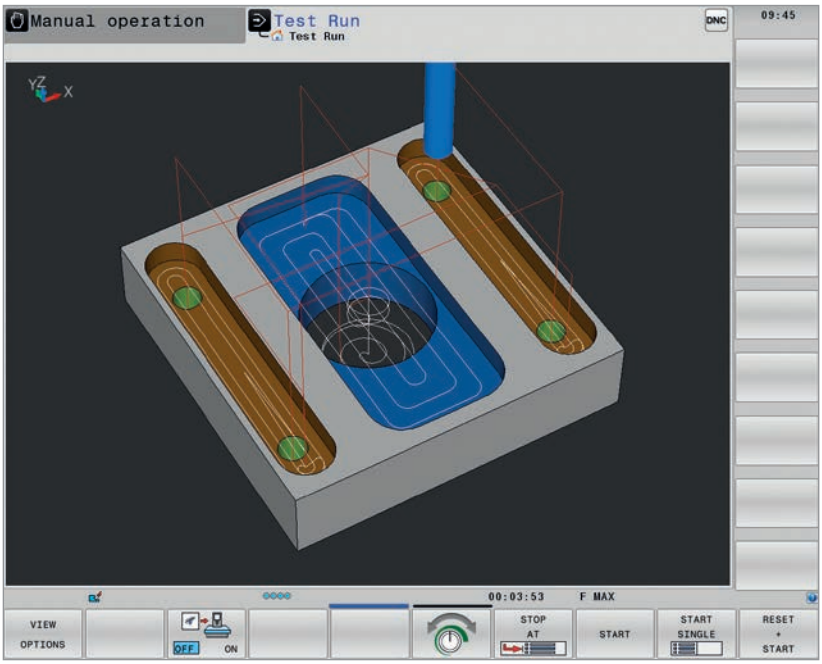
Finely detailed 3-D view
The program-verification and the program-run graphics of the TNC 640, TNC 620, and TNC 320 include further view options with detailed 3-D view. They help you to evaluate the workpiece quality in the simulation even before the actual machining operation begins, as well as during real-time machining.

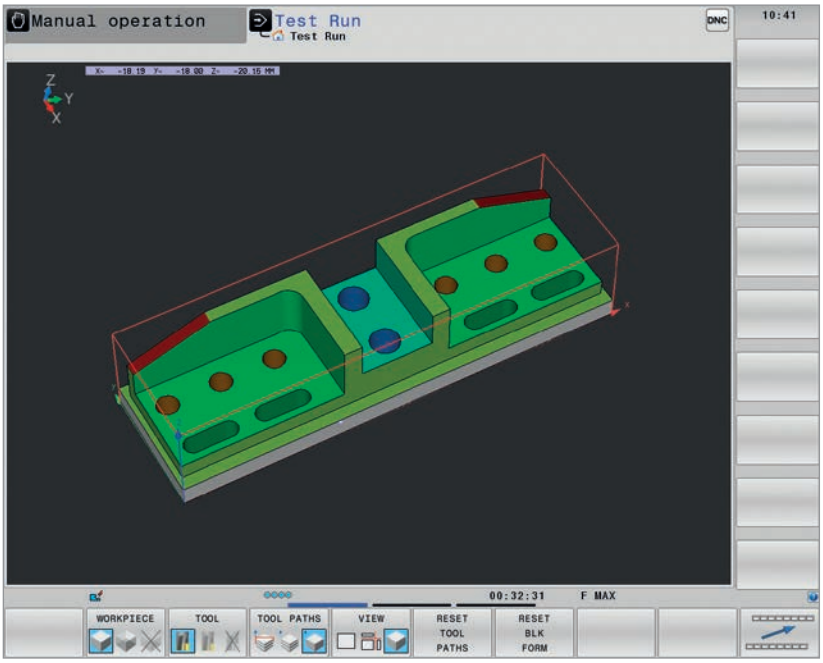
The high-resolution 3-D view with high zoom factors reveals the tiniest program errors on the workpiece surface. It also provides the possibility of showing the tool paths, including the associated block numbers, in order to permit an even closer analysis of the NC data. The selectable workpiece transparency feature that makes it possible to detect hidden cavities and undercuts is also very useful.

The enhanced graphics also improve the visibility of tool-specific machining operations: all machining operations that are performed with one and the same tool are depicted in their own color. The TNC also features a measuring function in the 3-D view. You can position the mouse pointer anywhere in the graphic to see the coordinates.

In addition to the material removal simulation, the TNC 640 also shows the entire 3-D machine model. The user can thus precisely evaluate actions in the work envelope ahead of the actual machining operation. The machine tool builder configures and enables the collision objects of the machine.

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





Program-Verification Graphics, Program-Run Graphics	Option 20	ID 628253-01
TNC 640 HSCI	Standard as of NC SW 34059x-04	
TNC 620 HSCI	As of NC SW 81760x-01	
TNC 320	Standard as of NC SW 771851-01	
iTNC 530 HSCI	–	
iTNC 530	–	
Installation by the machine tool builder		
For more information, see the TNC brochures and visit *www.klartext-portal.com*		

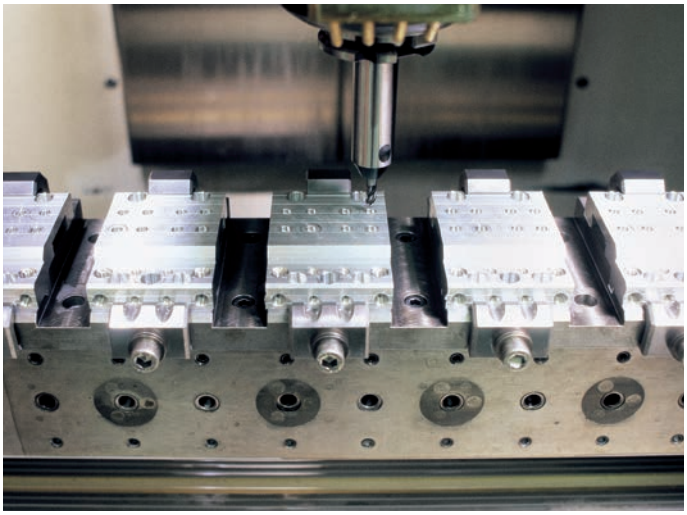
Programming and operation
Pallet Editor

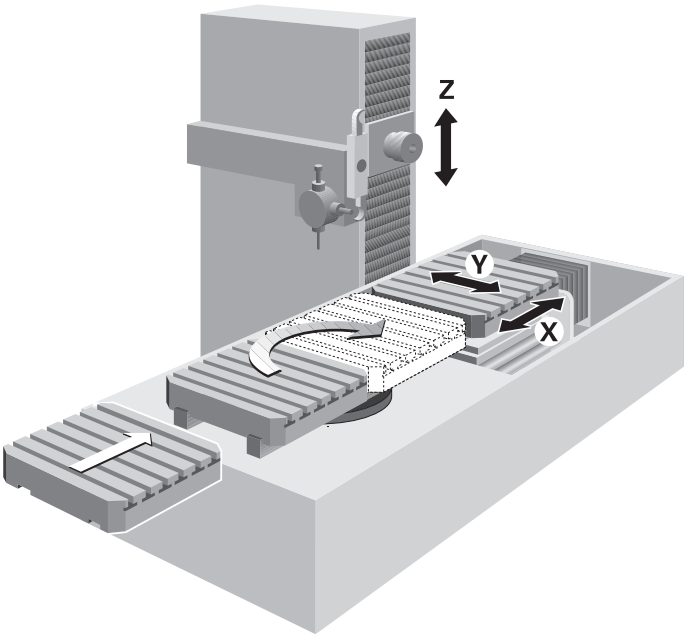
The TNC can assign the appropriate program and corresponding datum shift to workpieces that are mounted on pallets and brought to the machine in a random sequence.

If a pallet is inserted for machining, the TNC automatically calls the correct program. This permits automatic machining of a variety of parts in any sequence.

Pallet movement can be controlled via PLC axes. The order of movement, as well as the pallet and workpiece presets, must be defined in the pallet table by the user. The pallet table is freely configurable by the machine tool builder, which means that any information can be stored in the tables and called up later by the PLC.

The execution of pallet tables can be workpiece-oriented or tool-oriented (on the iTNC 530, and with the TNC 640 as of NC SW 34059x-08, and with the TNC 620 as of NC SW 81760x-05).





Pallet Editor	Option 22	ID 628255-01
TNC 640 HSCI	Standard	
TNC 620 HSCI	As of NC SW 34056x-01/73498x-01/81760x-01	
TNC 320	–	
iTNC 530 HSCI	Standard	
iTNC 530	Standard	
Installation by the machine tool builder		
For more information, see the TNC brochures		

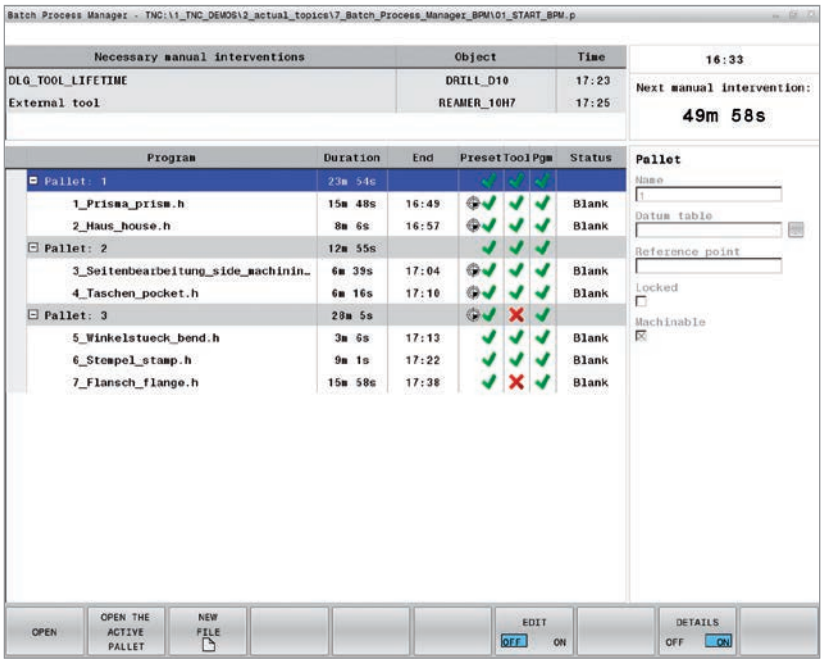
Programming and operation

Batch Process Manager: easy-to-understand depiction of the pallet management

Batch Process Manager* is a powerful function for pallet machining and series production. With the clear-cut user interface, you plan your production process and receive important information about the upcoming operations.

Batch Process Manager automatically checks for missing tools, insufficient tool life, and required manual tool changes. The result of the check is displayed in the status overview.

In Batch Process Manager, the following information is already displayed in advance:
- Sequence of operations
- Time of next manual intervention
- Program duration and run time
- Status information regarding the preset, tool, and program



* On controls with NC software versions up to 34059x-08 or 81760x-05, Extended Tool Management (option 93) must be enabled.

Batch Process Manager*	Option 154	ID 1219521-01
TNC 640 HSCI	As of NC SW 34059x-08	
TNC 620 HSCI	As of NC SW 81760x-05	
TNC 320	–	
iTNC 530 HSCI	–	
iTNC 530	–	
Installation by the machine tool builder		
For more information, see the *TNC 620* and *TNC 640* brochures		

Programming and operation

DXF Converter: importing of contours and machining positions from DXF files

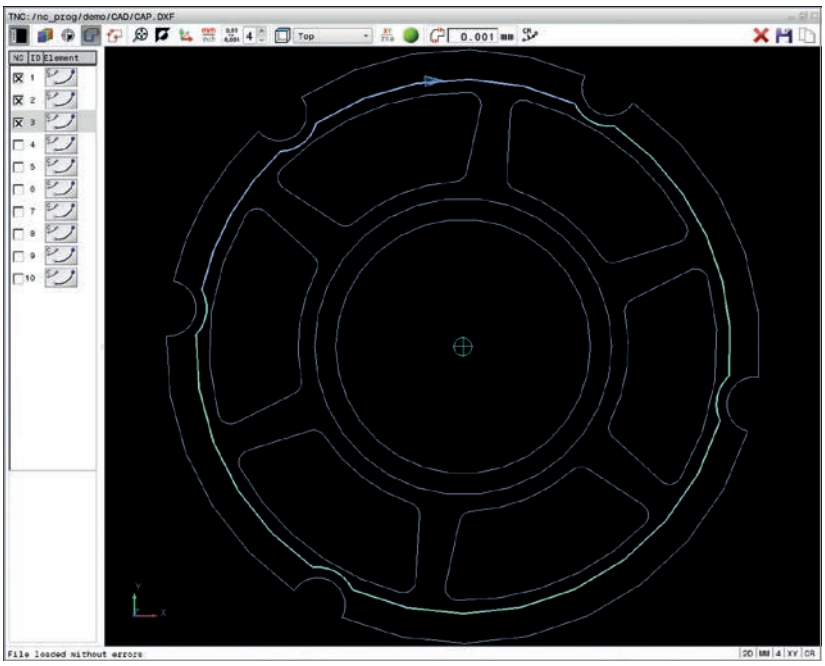
Why program contours when your drawing is already in DXF format anyway? You can open DXF files directly on the TNC in order to extract contours or machining positions from them. This not only saves time otherwise spent on programming and testing but also provides assurance that the finished contour is exactly in accordance with the design engineer's specifications.

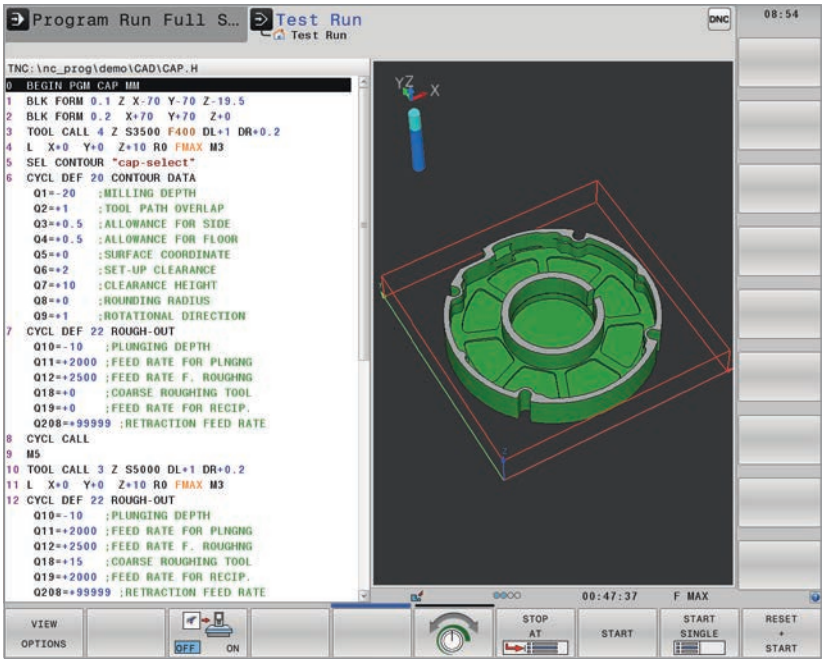
As a rule, DXF files contain multiple layers with which the design engineer organizes a drawing. In order to have as little unnecessary information as possible on the screen when selecting the contours, with a click of the mouse you can hide all **excessive layers** contained in the DXF file. This requires an operating panel with touchpad or an external pointing device. The TNC can select a contour train even if it has been saved over **different layers**.

The TNC also supports you when **defining the workpiece preset**. The TNC has a function for this, with which you can shift the datum in the drawing to a suitable location, simply by clicking on an element. Contour selection is exceptionally user friendly. You select any element by clicking it with the mouse. 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. There you click the immediately following contour element. In this way, you can define even complex contours with just a few mouse clicks. If desired, you can also shorten, lengthen, or interrupt the contour elements.

But you can also easily select **machining positions** and save them as point files, particularly in order to transfer drilling positions or starting points for pocket machining. Naturally, the TNC saves the machining positions such that they can be reached by the shortest path.

With the CAD Import option, you can also import contours and machining positions from 3-D models (see *CAD Import*, p. 22)





DXF Converter	Option 42	ID 526450-01
TNC 640 HSCI	As of NC SW 34059x-02	
TNC 620 HSCI	As of NC SW 73498x-02/81760x-01	
TNC 320	As of NC SW 771851-01	
iTNC 530 HSCI	As of NC SW 60642x-01	
iTNC 530	As of NC SW 34049x-02	
Installation by the user		
For more information, see the TNC brochures and visit *www.klartext-portal.com*		

Programming and operation
CAD Import: importing of contours from 3-D models

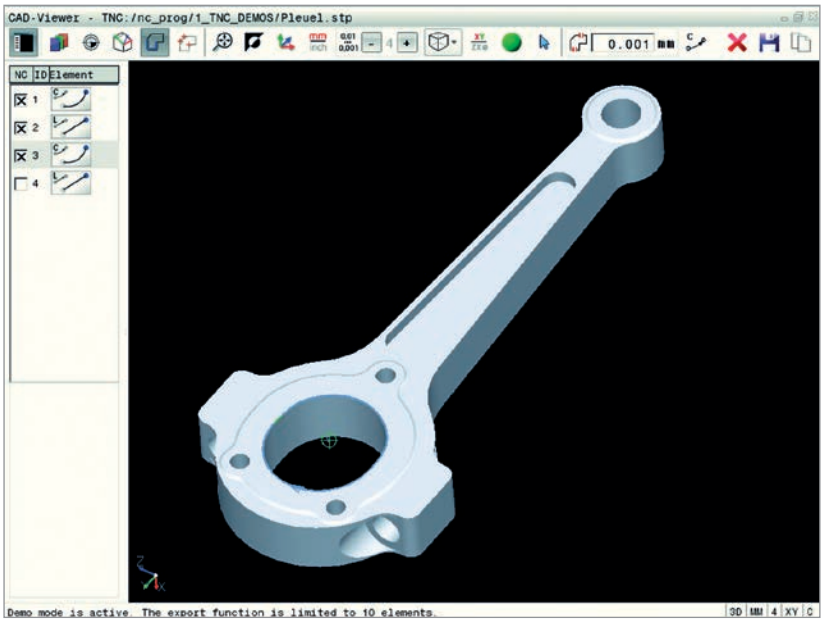
CAD Import (option 42)
The CAD Viewer enables the display of **2-D and 3-D models** (e.g., in DXF, STEP, or IGES file formats) right on the TNC. Using the CAD Import option, you can easily and directly incorporate contours and machining positions from these CAD files into your Klartext program. You can thereby reduce the amount of programming work and avoid input errors resulting from transposed digits or incorrectly placed decimal points, for example.

Extracting machining information directly from CAD data particularly provides additional possibilities for creating NC programs with a tilted machining plane. You can define the preset with a **3-D basic rotation** of the 3-D model. Plus, you can place a datum with the appropriate 3-D rotation on 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.** On the defined working plane you can extract contours and machining positions and apply them to the NC program.

Selecting the contour is particularly convenient. You select any element by clicking it with the mouse. 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. By this means, you can define extensive contours with just a few mouse clicks. Then you can simply copy the selected contour to an existing Klartext program via the clipboard.

The CAD Import option is an expansion of the DXF Converter. All previous functions have been adopted and expanded to include 3-D import functions. In addition, nearly all of the common DXF, STEP, and IGES file formats can be used. The CAD Import option is available for the TNC 640, TNC 620, and TNC 320 controls.



CAD Import	Option 42	ID 526450-01
TNC 640 HSCI	As of NC SW 34059x-08	
TNC 620 HSCI	As of NC SW 81760x-05	
TNC 320	As of NC SW 771851-05	
iTNC 530 HSCI	–	
iTNC 530	–	
Installation by the user		
For more information, see the TNC brochures and visit *www.klartext-portal.com*		

Programming and operation

Turning Functions: milling and turning on the same machine with the TNC 640

The TNC 640 offers you powerful functions that enable you to switch between turning mode and milling mode as desired in the NC program under program control. This enables you to decide with complete freedom how and when you want to combine the two machining methods.

Programming as accustomed
You can program turning operations—as always—with convenient dialog guidance in HEIDENHAIN Klartext format. Besides the standard path functions, you can also use FK free contour programming to easily create contour elements not otherwise dimensioned for NC. In addition, turning-specific recessing and undercutting contour elements are available and can be defined with the help of informative help illustrations.

Cycles for milling and turning
HEIDENHAIN controls have always been known for their comprehensive and technologically sophisticated package of cycles. Frequently recurring operations that comprise several steps are also stored in the TNC 640 as cycles. You program them under conversational guidance and are supported by enlightening help graphics that clearly illustrate the required input parameters. Besides the well-known TNC milling and drilling cycles, the TNC 640 also offers a wide variety of turning cycles, for example for roughing, finishing, recessing, and thread turning. The field-proven HEIDENHAIN lathe controls provide the software basis for the turning functions. They enable you to very easily program even complex turning operations at the machine.

Turning with a facing slide
With a facing slide, you can perform turning operations on a stationary workpiece. It makes turning operations outside the center of rotation or in a tilted plane possible. With a facing slide, the cutting edge rotates in the spindle, while an axis integrated in the facing head controls the turning tool (plan stroke). With the TNC 640, you do not have to worry about these complex motion sequences. You simply use a program command to select facing-slide operation and then program the standard turning cycles as usual. The TNC 640 handles all conversions and performs all the motion sequences on its own.

Turning Functions	Option 50	ID 634608-01
TNC 640 HSCI	As of NC SW 34059x-01	
TNC 620 HSCI	–	
TNC 320	–	
iTNC 530 HSCI	–	
iTNC 530	–	

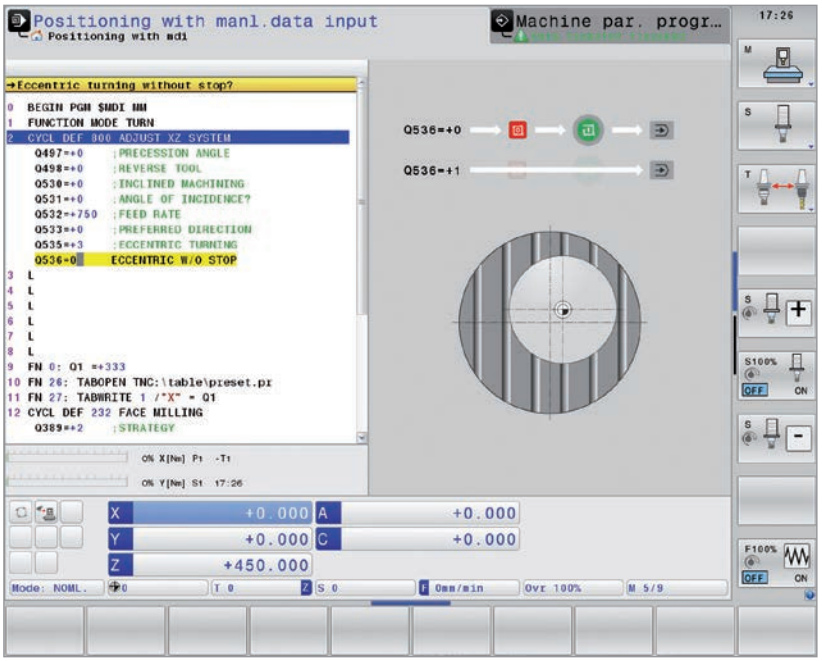
Installation by the machine tool builder

For more information, see the *TNC 640* brochure and visit *www.klartext-portal.com*

Programming and operation
Turning Functions: eccentric turning

With the Eccentric Turning function, you can perform turning operations even when the tool axis, as a result of the setup situation, is not aligned with the axis of rotation. During machining, the TNC 640 compensates any eccentricity with corrective movements of the linear axis coupled with the rotating spindle. This can significantly reduce the time required for setup.

Eccentric turning* requires software option 50 (Turning Functions).



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

Eccentric Turning	Option 50	ID 634608-01
TNC 640 HSCI	As of NC SW 34059x-04	
TNC 620 HSCI	–	
TNC 320	–	
iTNC 530 HSCI	–	
iTNC 530	–	
Installation by the machine tool builder		
More information	–	

Programming and operation
Turning Functions: simultaneous finishing

You can use this option to machine complex contours that are only possible with different inclinations. For simple contours you can use a large area of the indexable insert to achieve longer tool life. This results in machining operations with at least three axes (two linear axes and one rotary axis).

The simultaneous finishing cycle monitors the workpiece contour with respect to the tool and the tool carrier. The cycle avoids unnecessary tilting movements in order to machine optimum surfaces. However, if you want to force tilting movements, you can define beginning and end angles.

Simultaneous finishing for turning requires options 50 and 158 (Turning Functions and Advanced Function Set Turning).

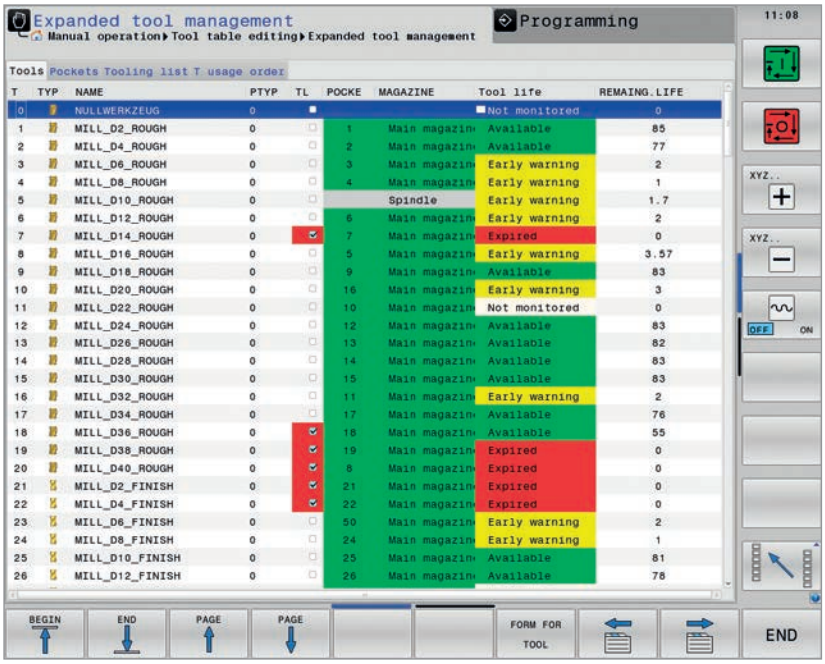
Advanced Function Set Turning	Option 50 Option 158	ID 634608-01 ID 1237237-01
TNC 640 HSCI	As of NC SW 34059x-09	
TNC 620 HSCI	–	
TNC 320	–	
iTNC 530 HSCI	–	
iTNC 530	–	
Installation by the machine tool builder		
More information	–	

Programming and operation
Expanded Tool Management

The Expanded Tool Management option contains many new features that make the management of tools and magazines considerably more transparent. Loading and unloading processes can be managed by dragging and dropping with the mouse. A tool usage list states how long which tools have been in contact with workpieces, and clearly structured tables use color coding to indicate various tool statuses. In addition, all tools used in the selected program can be displayed in a tooling list.

There is now an import function for reading and exporting CSV files. CSV (Comma Separated Values) is a text file format for the exchange of simply structured data. This function is especially useful for data exchange if you measure and calibrate your tools with external presetters. Excel can also open and save this file format.

There is now also a simple interface for deleting tool data quickly and confidently. The TNC shows the tool data to be deleted in a pop-up window, giving you the opportunity to make sure that no important data is deleted by accident.



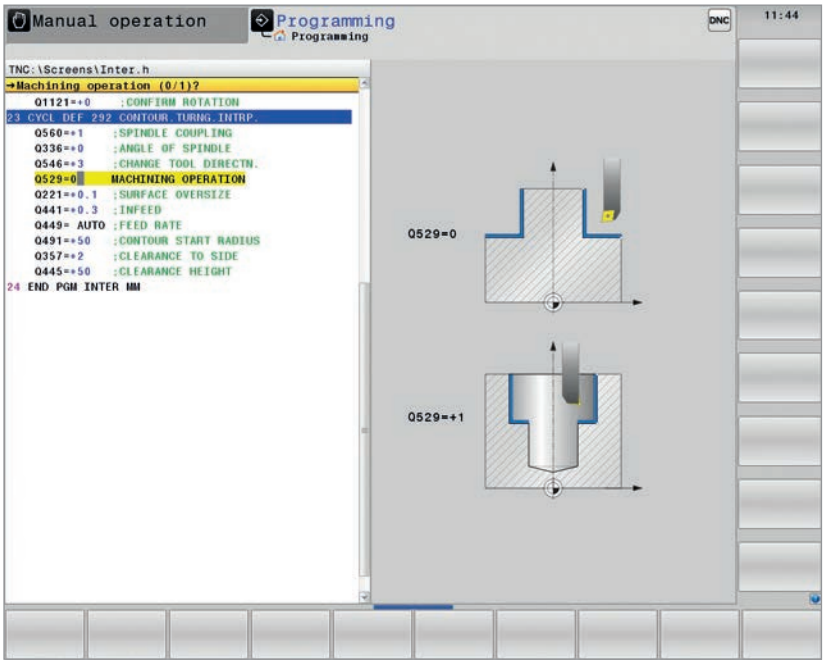
Expanded Tool Management	Option 93	ID 676938-01
TNC 640 HSCI	As of NC SW 34059x-01	
TNC 620 HSCI	As of NC SW 81760x-03	
TNC 320	As of NC SW 771851-02	
iTNC 530 HSCI	As of NC SW 60642x-01	
iTNC 530	As of NC SW 34049x-05	
Installation by the machine tool builder		
More information	–	

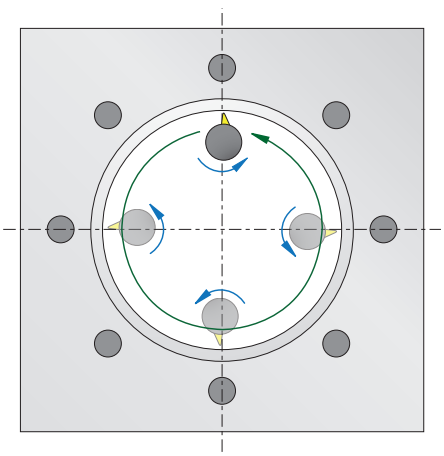
Programming and operation
Interpolating Spindle: interpolation turning

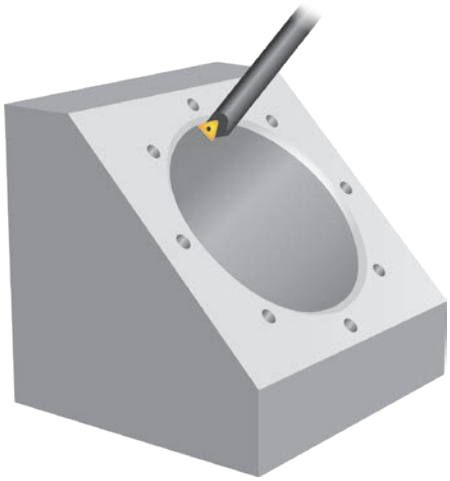
During interpolation turning, the movement of the tool's cutting edge describes a circle, with the cutting edge always oriented toward the center of the circle (outside machining) or away from the center (inside machining). By varying the circle radius and the axial position, any rotationally symmetric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

With the interpolation turning cycle, the TNC can create a rotationally symmetric shoulder in the active machining plane that is defined by its starting and ending point. The center of rotation is the starting point in the working plane at the time the cycle is called. The rotational surfaces can be inclined or rounded relative to each other.

This cycle can only be used for finishing. Roughing operations with multiple passes are not possible. The machining strategy can be chosen flexibly; both inside machining and outside machining are possible. With the TNC 640, you can also machine any rotationally symmetrical contours (without undercuts).





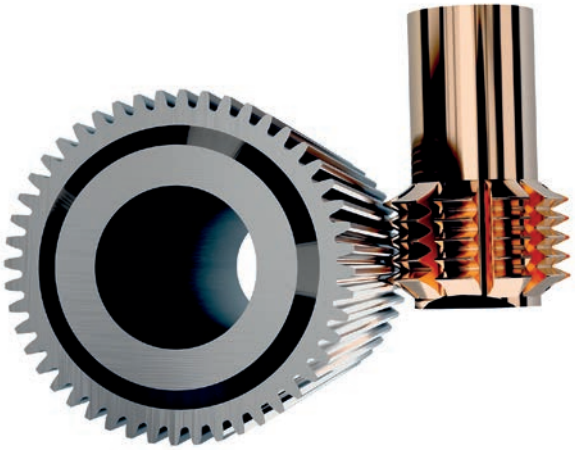


Interpolating Spindle	Option 96	ID 751653-01
TNC 640 HSCI	As of NC SW 34059x-05	
TNC 620 HSCI	–	
TNC 320	–	
iTNC 530 HSCI	As of NC SW 60642x-02	
iTNC 530	As of NC SW 34049x-07	
Installation by the machine tool builder		
For more information, see the *iTNC 530* and *TNC 640* brochures		

Programming and operation
Spindle Synchronization

Specific machining operations require that the rotation of the tool spindle be synchronized with the movement of other axes. This is the case, for example, when manufacturing external gears using the hobbing process.

In conjunction with option 50 (Turning Functions) and option 131 (Spindle Synchronization), the TNC 640 offers you Cycle 880 GEAR HOBBING which enables you to machine external cylindrical gears or helical gears with any angles. During hobbing, the rotation of the tool spindle and that of the rotary table are synchronized. In addition, the gear hob moves along the workpiece in axial direction. Cycle 880 automatically controls these complex movements and enables you to enter all relevant values easily and practically. You can use the tooth parameters directly from your drawing, from which the cycle calculates the 5-axis sequence of movements.



Spindle Synchronization	Option 131	ID 806270-01
TNC 640 HSCI	As of NC SW 34059x-05	
TNC 620 HSCI	–	
TNC 320	–	
iTNC 530 HSCI	–	
iTNC 530	–	
Installation by the machine tool builder		
For more information, see the *TNC 640* brochure		

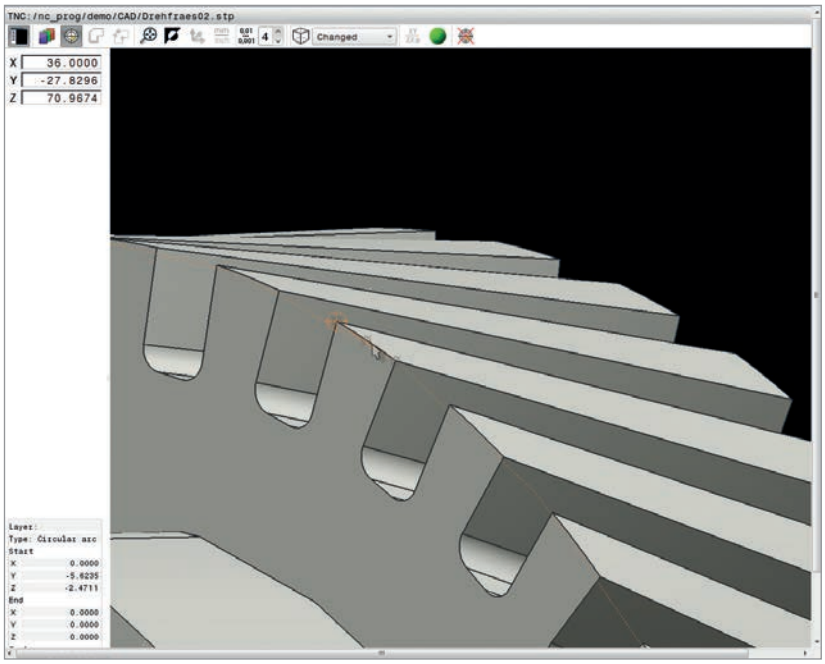
Programming and operation
CAD Viewer: display of standardized CAD formats

The **CAD Viewer** function allows you to open standardized 2-D and 3-D CAD data formats directly on the TNC. It is irrelevant whether the file is made available on the hard disk of the TNC or a connected drive.

The file can simply be selected via the file manager of the TNC, just like NC programs or other files. The user can check 3-D models for errors or problems quickly and without delays.

The 3-D CAD Viewer opens automatically when you select a CAD data format (e.g., IGS, IGES, or STEP) in the file manager of the TNC. Naturally, the 3-D CAD Viewer includes functions for shifting, rotating, and zooming so that any problematic locations can be appropriately displayed.

Moreover, you can also use the viewer to find position values and dimensions from the 3-D model. And you can set the preset as desired and select elements in the model. The CAD Viewer shows the coordinates of the selected elements in a window.



CAD Viewer	Option 98	ID 800553-01
TNC 640 HSCI	Standard as of 34059x-05	
TNC 620 HSCI	Standard as of 81760x-02	
TNC 320	Standard as of 771851-01	
iTNC 530 HSCI	As of NC SW 60642x-02	
iTNC 530	As of NC SW 34049x-07	
Installation by the user		
More information	–	

Programming and operation
Gear Cutting: manufacturing gears in one setup

Gears are usually manufactured on special machine tools, meaning that workpieces need to be laboriously rechucked.

You use the Gear Cutting function to easily, economically, and completely manufacture spur or helical gearing systems in one setup with skiving or hobbing operations. The option allows both types of operation in milling mode as well as in turning mode.

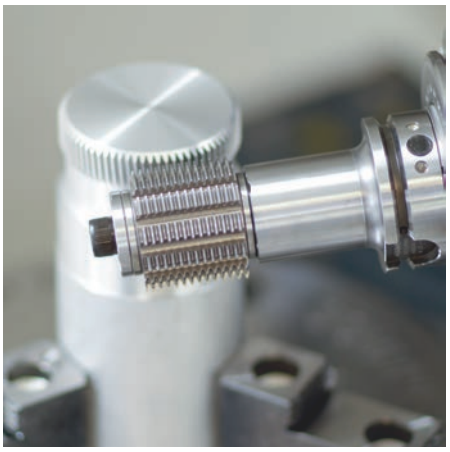
The function assists you when programming the complex sequences. You need only define the data for the gear geometry and the tools to be used. The TNC 640 performs all other calculations, especially complex synchronization of the movements. As a result, the machining of internal gear teeth is transformed into a simply-mastered standard.

Skiving
The current success of skiving is based on the significantly higher efficiency and levels of productivity compared with traditional heading.

This procedure makes the manufacture of external and internal gears possible on machines with synchronized spindles.

Hobbing
Hobbing is especially well suited for the machining of external gears. The advantages are found in the high levels of productivity and diverse tooth shapes that can be machined with relatively easily produced tools.

Lift-off
An important advantage of this option, along with its simple programming, is the safety aspect. In the event of unforeseen program interruptions (such as a power outage), these cycles support optimized lift-off in order to prevent damage. These cycles automatically define both the direction and path for retraction of the tool from the workpiece.



Gear Cutting	Option 157 (in Turning mode: option 50)	ID 1237235-01
TNC 640 HSCI **TNC 620 HSCI** **TNC 320** **iTNC 530 HSCI** **iTNC 530**	As of NC SW 34059x-09 – – – –	
Installation by the machine tool builder		
More information	–	

Machine accuracy

KinematicsOpt: easy calibration of rotary axes

Accuracy requirements are becoming ever more stringent, particularly in the realm of 5-axis machining. Complex parts must be manufactured with both precision and reproducible accuracy, including over extended periods of time.

The TNC function **KinematicsOpt** is an important component to help you meet these high requirements. With a HEIDENHAIN touch probe inserted, a touch probe cycle measures your machine's rotary axes fully automatically. Measurement is the same regardless of whether the axis is a rotary table, a tilting table, or a swivel head.

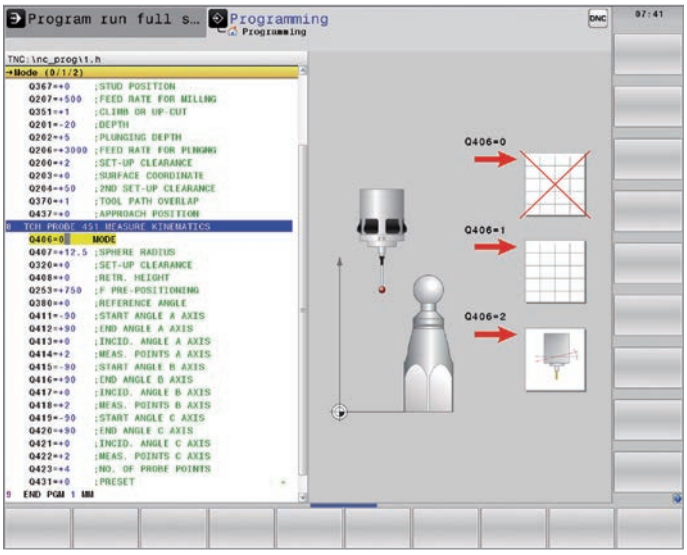
To measure the rotary axes, a calibration sphere is fixed at any position on the machine table and probed with the HEIDENHAIN touch probe. But first you define the resolution of the measurement and define the range that you want to measure for each rotary axis.

The TNC uses the measured values to determine the spatial errors resulting from the tilting of the axes. The cycle then calculates an optimized kinematic machine description, in which these errors are minimized, and saves it as the machine kinematics.

Of course, a comprehensive log file is also saved containing the actual measured values and the measured and optimized dispersion (measure of the static tilting accuracy), as well as the actual compensation values.

An especially rigid calibration sphere is necessary for optimum use of KinematicsOpt. This helps to reduce deformations that occur as the result of probing forces. For this reason, HEIDENHAIN offers calibration spheres with highly rigid holders that are available in various lengths.

Calibration spheres are available as accessories:
KKH 100 Height: 100 mm ID 655475-02
KKH 250 Height: 250 mm ID 655475-01



KinematicsOpt	Option 48	ID 630916-01
TNC 640 HSCI	As of NC SW 34059x-01	
TNC 620 HSCI	As of NC SW 34056x-03/73498x-01/81760x-01	
TNC 320	–	
iTNC 530 HSCI	As of NC SW 60642x-01	
iTNC 530	As of NC SW 34049x-04	
Installation by the machine tool builder		
For more information, see the *KinematicsOpt* brochure		

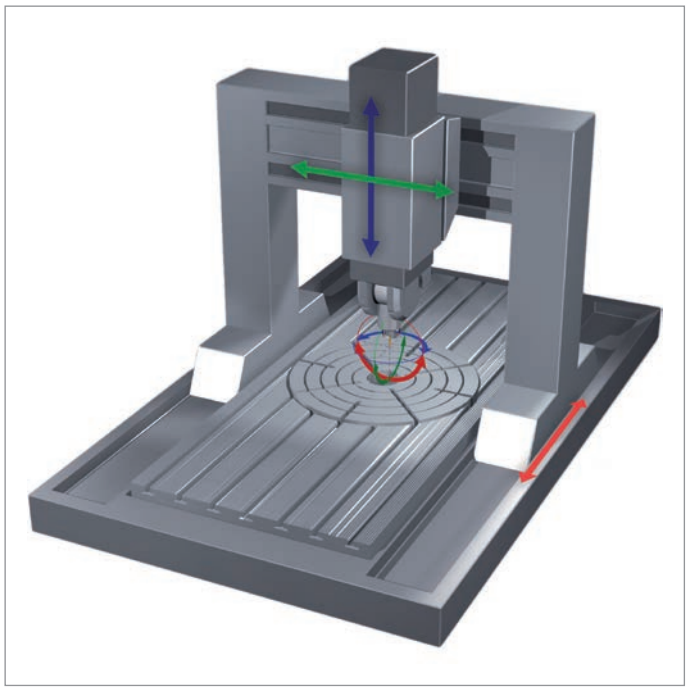
Machine accuracy
KinematicsComp: 3-D spatial compensation

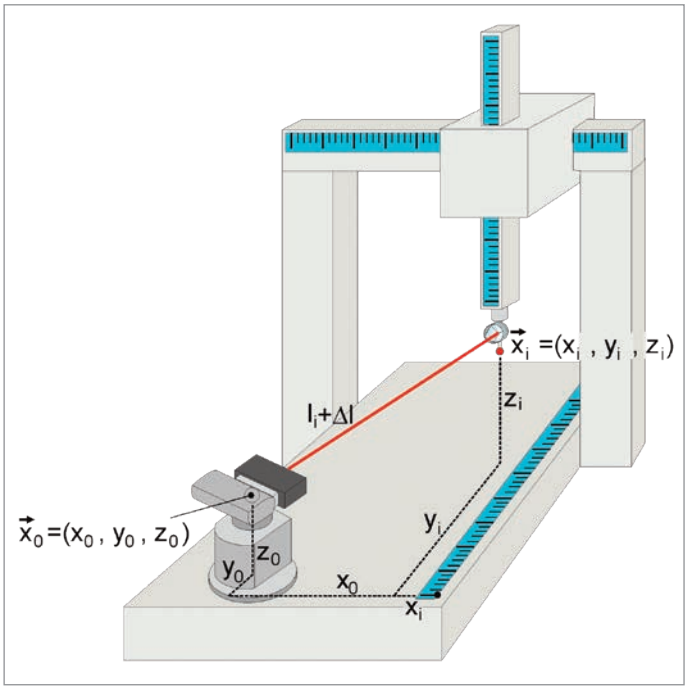
Narrow workpiece tolerances require high machine accuracy. However, machine tools inevitably have errors resulting from their assembly or production.

The more axes a machine has, the more sources of errors there are. For example, according to ISO 230-1, a linear axis can have eight relevant types of error (six component errors and two relevant position errors), and a rotary axis can have eleven (six component errors and five relevant position errors). The use of mechanical means to cope with these errors requires considerable effort. These errors become particularly noticeable on 5-axis machines or very large machines. Thermal expansion, which can cause very complex geometric changes to machine components, must also be considered.

The **KinematicsComp** function enables the machine tool builder to improve machine accuracy considerably. The machine's degrees of freedom and the positions of the centers of rotation of the rotary axes are described in the standard kinematics of the TNC. The expanded kinematics description of KinematicsComp permits the import of compensation-value tables. Most of the geometry errors of a machine can be described in compensation-value tables. They are compensated for so that the tool center point (TCP) can follow the ideal nominal contour exactly. Thermally induced errors can also be measured and compensated for via sensors and the PLC. For example, the spatial errors of the tool center point can be measured with a laser tracer or laser interferometer and can be converted in compensation-value tables.

The KinematicsComp option is not available in the export versions.





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

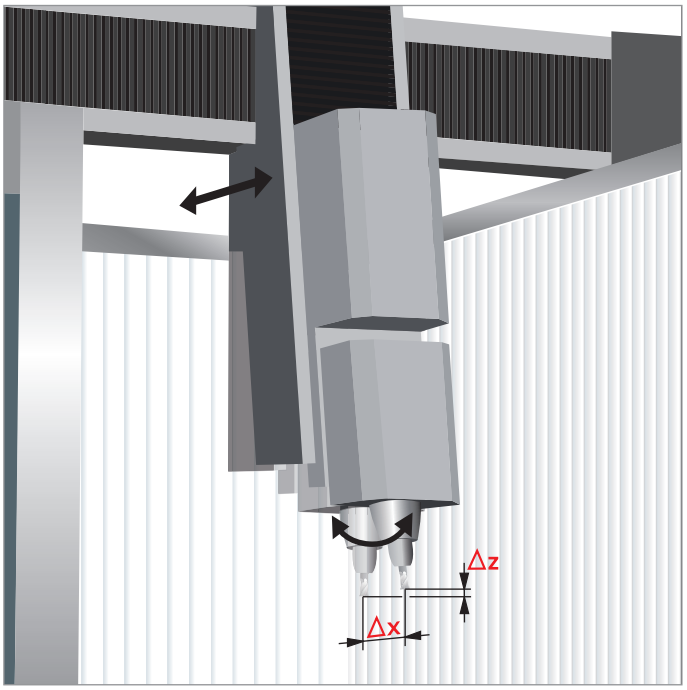
KinematicsComp	Option 52	ID 661879-01
TNC 640 HSCI	As of NC SW 34059x-05	
TNC 620 HSCI	–	
TNC 320	–	
iTNC 530 HSCI	As of NC SW 60642x-01	
iTNC 530	As of NC SW 34049x-06	
Installation by the machine tool builder		
For more information, see the *KinematicsComp* brochure		

Machine accuracy
CTC: compensation of position errors through axis coupling

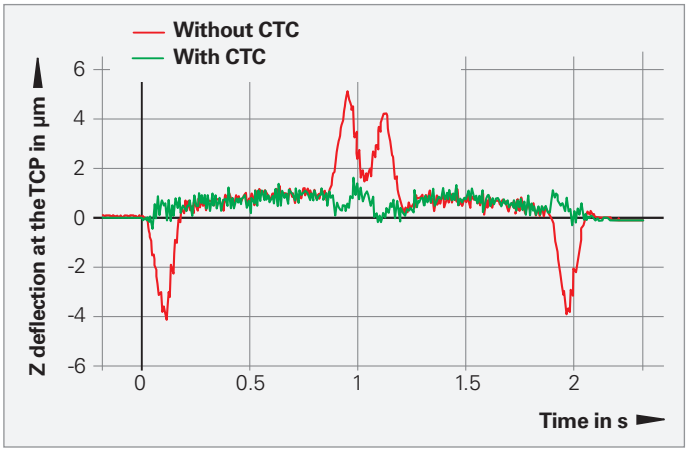
Dynamic acceleration processes introduce forces to the structure of a machine tool. They can deform parts of the machine for a short time and thereby lead to deviations at the tool center point (TCP). In addition to causing deformation in the axis direction, the dynamic acceleration of an axis on account of mechanical axis coupling can also result in axis deformation that is oblique to the direction of acceleration. This is especially the case if the point of force application on an axis does not coincide with its center of gravity, potentially resulting in pitching during the 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, this acceleration-dependent error can be compensated for with the servo-control option **CTC** (Cross Talk Compensation) 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 mutually mechanically coupled axes can be used to measure the acceleration-dependent position error of these axes. Often, the resulting error at the TCP depends not only on the acceleration but also on the position of the axes in the working space. This can also be compensated for by the CTC servo control option.



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



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

dynamic + precision

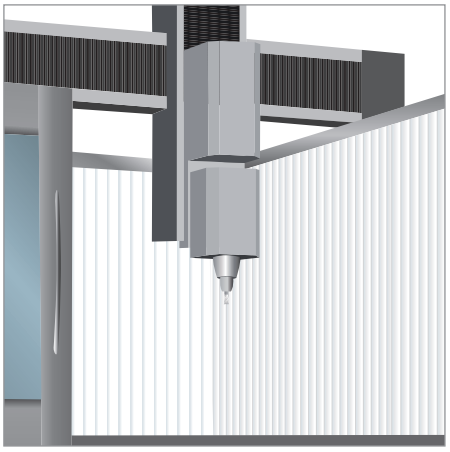
CTC	Option 141	ID 800542-01
TNC 640 HSCI	As of NC SW 34059x-02	
TNC 620 HSCI	As of NC SW 34056x-04/73498x-02/81760x-01	
TNC 320	–	
iTNC 530 HSCI	As of NC SW 60642x-02	
iTNC 530	–	
Installation by the machine tool builder		
For more information, see the *Dynamic Precision* Technical Information document		

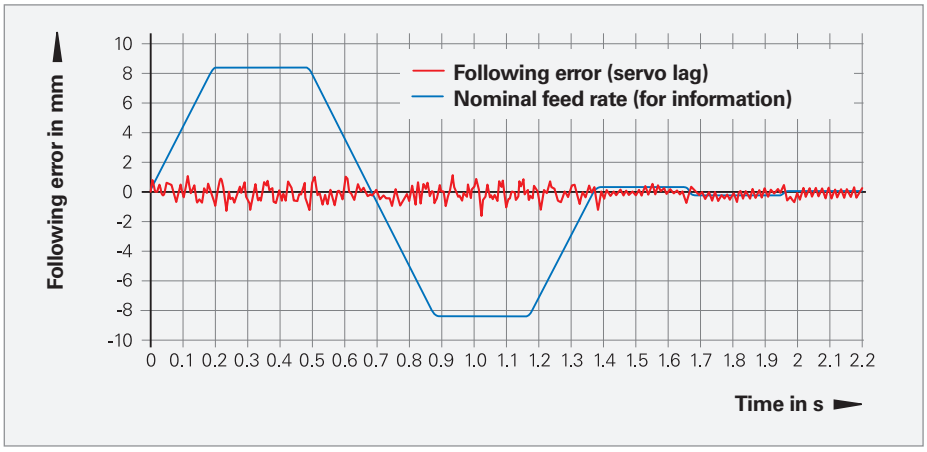
Machine accuracy
PAC: position-dependent adaptation of control parameters

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

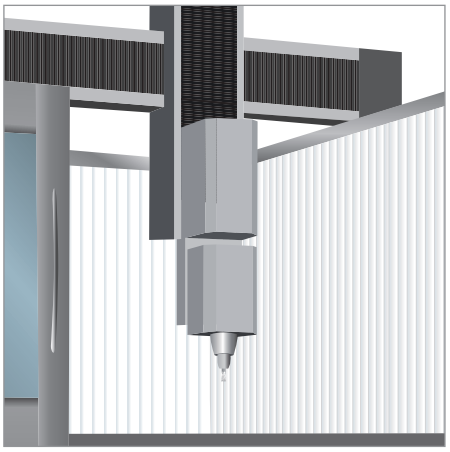
To exploit the machine's dynamic possibilities, you can use the **PAC** option (Position Adaptive Control) to change machine parameters based on position.

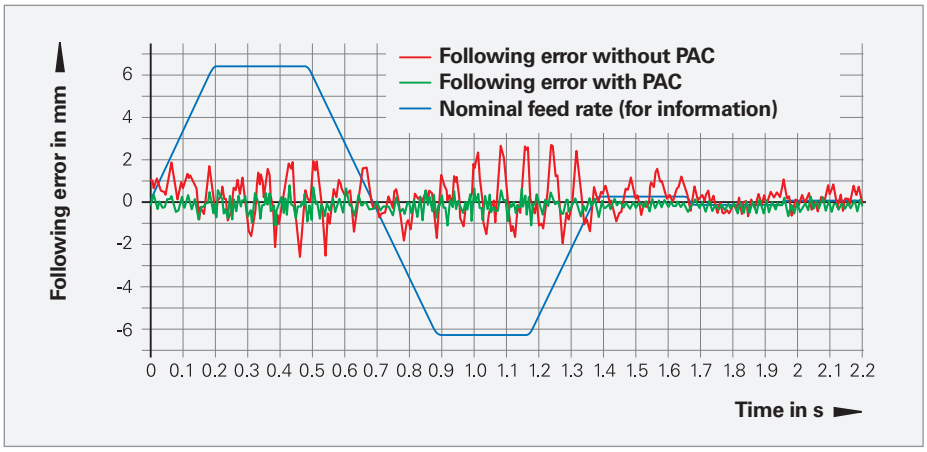
This makes it possible to assign the respective optimal loop gain to defined interpolation points. Additional position-dependent filter parameters can be defined in order to further increase control loop stability.





Servo control optimized for Z = 0, following error within the tolerance band (±1 µm)





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

dynamic + precision

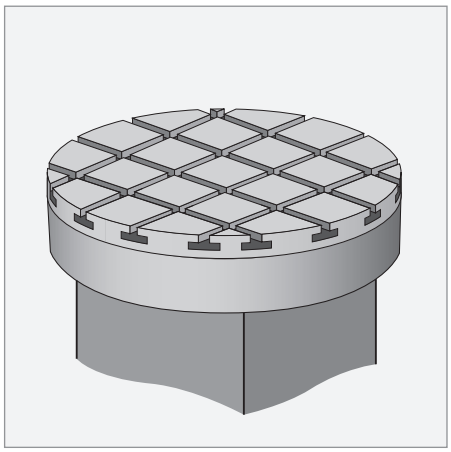
PAC	Option 142	ID 800544-01
TNC 640 HSCI	As of NC SW 34059x-02	
TNC 620 HSCI	As of NC SW 34056x-04/73498x-02/81760x-01	
TNC 320	–	
iTNC 530 HSCI	As of NC SW 60642x-02	
iTNC 530	–	
Installation by the machine tool builder		
For more information, see the *Dynamic Precision* Technical Information document		

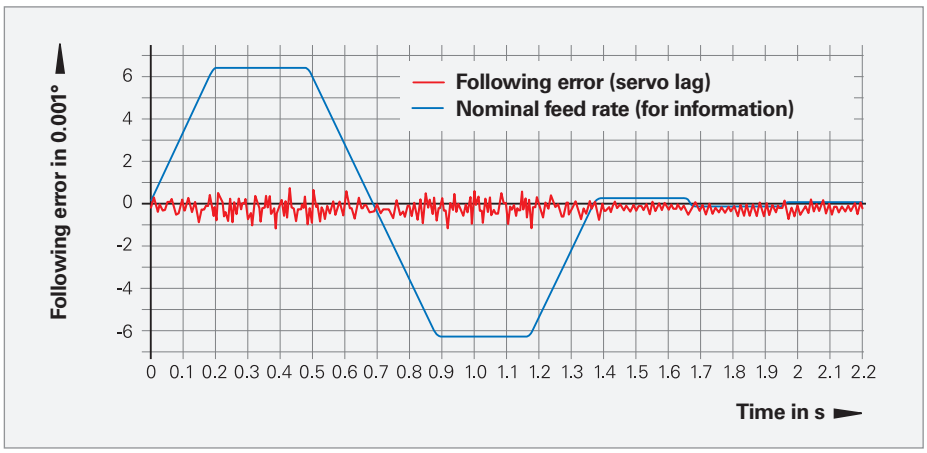
Machine accuracy
LAC: load-dependent adaptation of control parameters

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

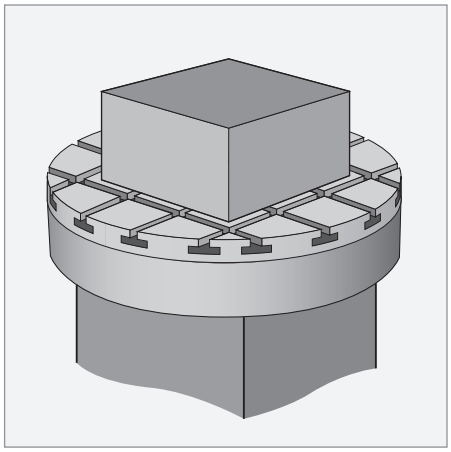
The **LAC** option (Load Adaptive Control), enables the control to automatically ascertain the current workpiece mass or mass moment of inertia as well as current frictional forces. In order to ensure an optimum response to changes in the behavior of the machine at differing loads, adaptive feedforward controls can be activated and the control loop gain can be changed depending on the load. During workpiece machining, the control can also continuously adjust the parameters of the feedforward control to the current mass of the workpiece.

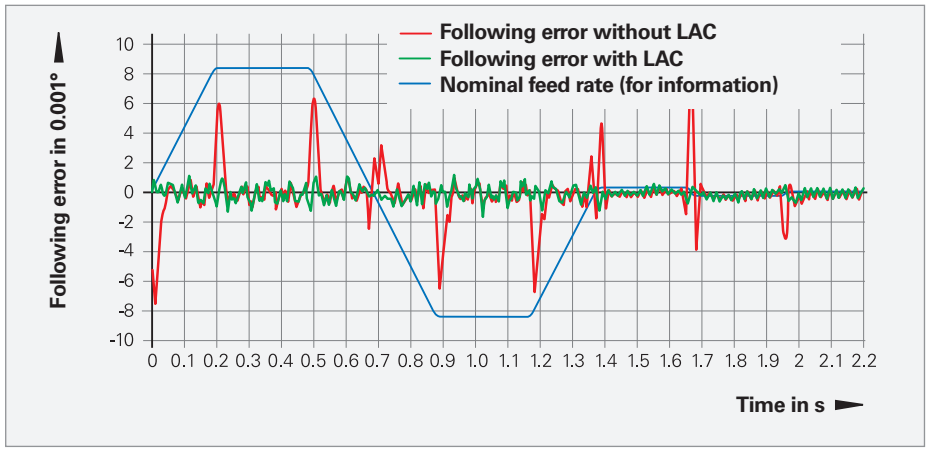
In order to allow rapid adjustment due to sudden changes in the load (e.g., from the loading and unloading of the workpiece), the TNC 620 and the TNC 640 provide Cycle 239 for ascertaining the current load status.





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





With change in load
- Without LAC: when the feedforward control is unchanged, the following error is outside of the tolerance band (±0.008°)
- With LAC: when LAC is active in the feedforward control, the following error is within the tolerance band (±0.001°)

dynamic + precision

LAC	Option 143	ID 800545-01
TNC 640 HSCI	As of NC SW 34059x-02	
TNC 620 HSCI	As of NC SW 34056x-04/73498x-02/81760x-01	
TNC 320	–	
iTNC 530 HSCI	As of NC SW 60642x-02	
iTNC 530	–	
Installation by the machine tool builder		
For more information, see the *Dynamic Precision* Technical Information document		

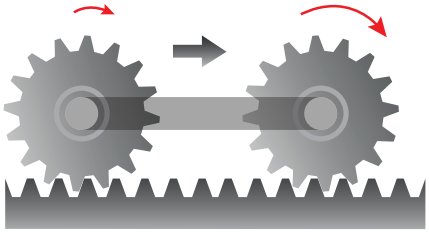
Machine accuracy

MAC: motion-dependent adaptation of control parameters

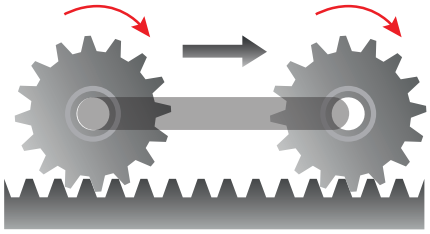
In addition to the position-dependent adaptation of control parameters by the PAC option, the **MAC** option (Motion Adaptive Control) also provides a means of changing machine parameters based on other input quantities such as velocity, following error, or drive acceleration. This motion-dependent adaptation of the control parameters makes it possible, for example, to realize a velocity-dependent adaptation of the k_V factor on motors that exhibit stability changes at different traversing velocities.

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

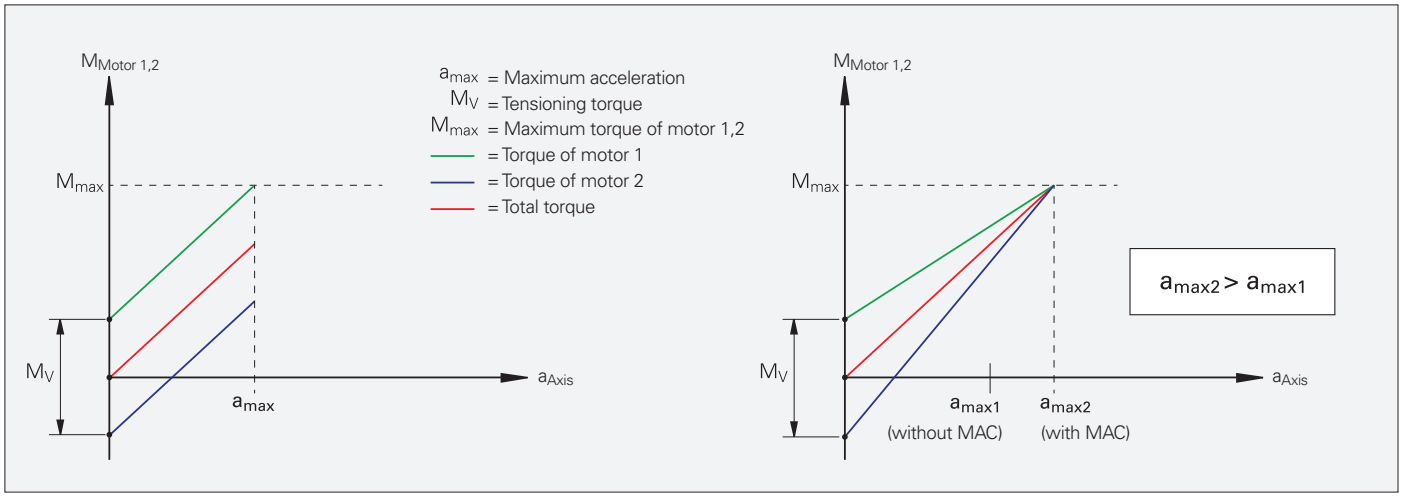
With the MAC option, this arrangement makes it possible to attain a significantly higher maximum acceleration at rapid traverse—for example, through parameter-based reduction of the tensioning torque with increasing acceleration.



Without MAC



With MAC



MAC	Option 144	ID 800546-01
TNC 640 HSCI	As of NC SW 34059x-02	
TNC 620 HSCI	As of NC SW 34056x-04/73498x-02/81760x-01	
TNC 320	–	
iTNC 530 HSCI	As of NC SW 60642x-03	
iTNC 530	–	
Installation by the machine tool builder		
For more information, see the *Dynamic Precision* Technical Information document		

dynamic + precision

Machine accuracy
AVD: active vibration damping

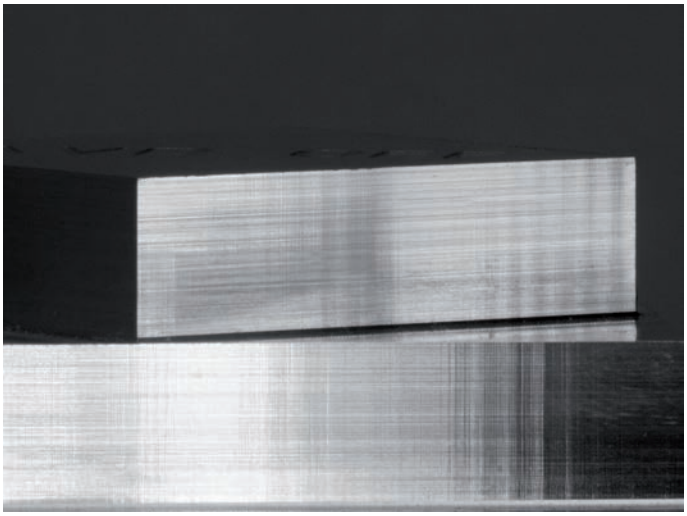
If there are low-frequency vibrations on a machine tool, inclined or curved surfaces can often have finish problems in the form of visible shadows or fluctuations in contrast. Peaks as small as 1 µm or even smaller can be visible on the workpiece surface. These defects often necessitate reworking of the surface, which results in additional costs.

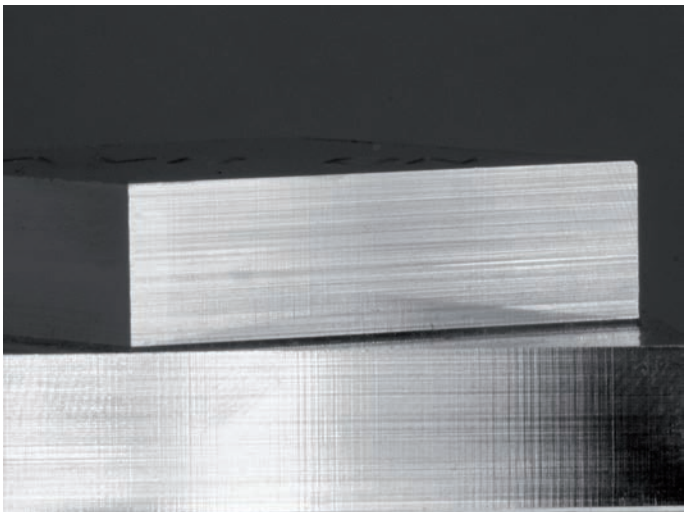
Common causes of low-frequency disturbances are elasticity in the drive train, such as when there are vibrations between the drive side (motor) and the friction side (slide), and vibrations from the machine setup, where high accelerations of the machine tool axes result in disturbances through the fastening elements of the machine setup or through the machine-tool base.

Excitations through high accelerations can be reduced by lowering the jerk, but this causes longer machining times.

The **Active Vibration Damping** (AVD) function uses the control loop of the TNC to precisely suppress a dominant low-frequency vibration. The AVD option has two effects: on the one hand, it leads to a clean workpiece surface since vibrations that become apparent there are suppressed, and on the other hand, AVD enables fast and low-vibration milling operations.

AVD thus increases the productivity of a machine tool and improves the surface quality of the workpieces.





dynamic + precision

AVD	Option 146	ID 800548-01
TNC 640 HSCI	As of NC SW 34059x-04	
TNC 620 HSCI	As of NC SW 34056x-04/73498x-02/81760x-01	
TNC 320	–	
iTNC 530 HSCI	As of NC SW 60642x-03	
iTNC 530	–	
Installation by the machine tool builder		
For more information, see the *Dynamic Precision* Technical Information document		

Machining functions
5-Axis Simultaneous Machining

The TNC provides numerous powerful functions specifically developed for 5-axis simultaneous machining.

The NC programs for 5-axis simultaneous machining are produced with CAM systems in conjunction with postprocessors. In principle, such programs contain either all coordinates of the machine's existing NC axes or NC blocks with surface normal vectors. During 5-axis machining with three linear axes and two additional tilting axes, the tool is always normal to the workpiece surface or is kept at a specific angle to it (inclined tool machining).

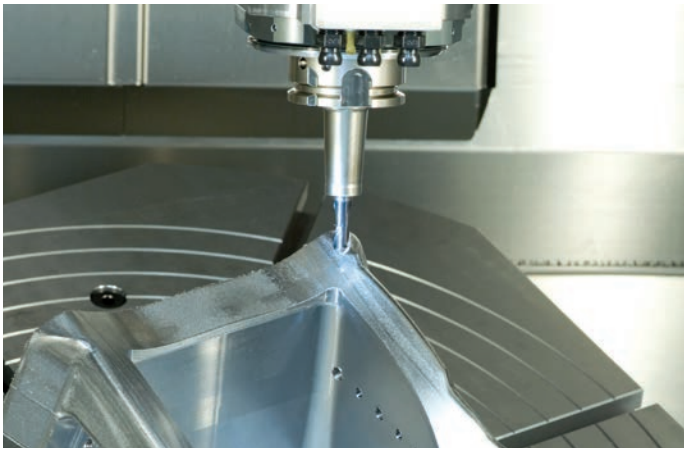
Regardless of what type of 5-axis programs you wish to run, the TNC performs all of the necessary compensating movements in the linear axes that result from movements in the tilting axes. The TNC's tool center point management feature (TCPM)—an improvement on the proven TNC function M128—provides optimal tool guidance and prevents contour gouging.

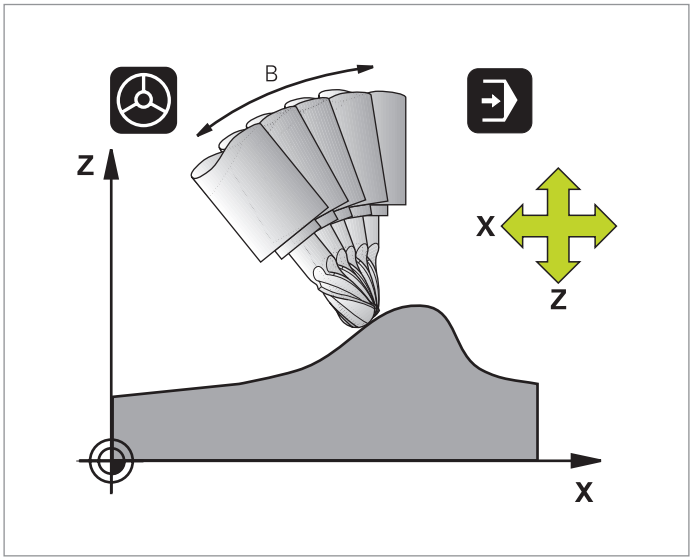
5-Axis Simultaneous Machining	Option 9	
TNC 640 HSCI/TNC 620 HSCI		ID 617921-01
iTNC 530 HSCI/iTNC 530		ID 367590-01
TNC 640 HSCI	As of NC SW 34059x-01	
TNC 620 HSCI	As of NC SW 34056x-01/73498x-01/81760x-01	
TNC 320	–	
iTNC 530 HSCI	As of NC SW 60642x-01	
iTNC 530	As of NC SW 34049x-01	
Installation by the machine tool builder		
For more information, see the TNC brochures		

Machining functions

Handwheel Superimpositioning: superimposing of 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 if you want to change the inclination angles of rotary axes in externally written NC programs, since these often result in 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.



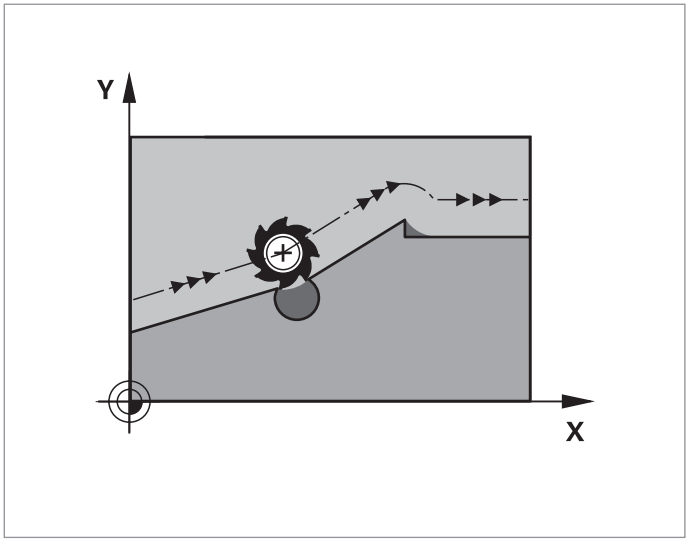


Handwheel Superimpositioning	Option 21	ID 628254-01
TNC 640 HSCI	Standard	
TNC 620 HSCI	As of NC SW 34056x-01/73498x-01/81760x-01	
TNC 320	–	
iTNC 530 HSCI	Standard	
iTNC 530	Standard	
Installation by the machine tool builder		
For more information, see the TNC brochures		

Machining functions

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

The LOOK AHEAD function in the geometry processing system of the TNC checks radius-compensated paths for contour undercuts and tool path intersections, and it calculates the tool path ahead of time starting from the current block. Areas of the contour that might be damaged by the tool are not machined (dark areas in the figure) and can be reworked later with a smaller tool. You can also use this function to enter tool radius compensation in NC programs that were created with an external programming system and that were output with uncompensated contour. This makes it possible to compensate for inaccuracies in the NC programs resulting from calculations in the CAM system.



Tool Compensation	Option 21	ID 628254-01
TNC 640 HSCI	Standard	
TNC 620 HSCI	As of NC SW 34056x-01/73498x-01/81760x-01	
TNC 320	Standard	
iTNC 530 HSCI	Standard	
iTNC 530	Standard	
Installation by the machine tool builder		
For more information, see the TNC brochures		

Machining functions
DCM: dynamic collision monitoring

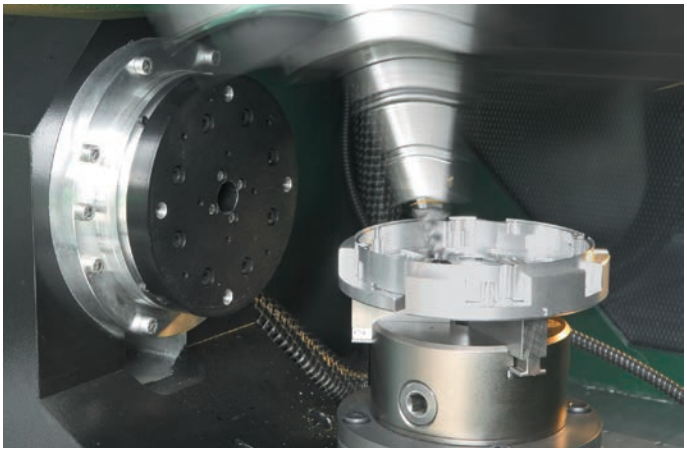
The complex motions and the normally high traversing speeds of 5-axis machining make axis movements difficult to foresee. This makes collision monitoring a valuable function that relieves the machine operator and protects the machine from damage.

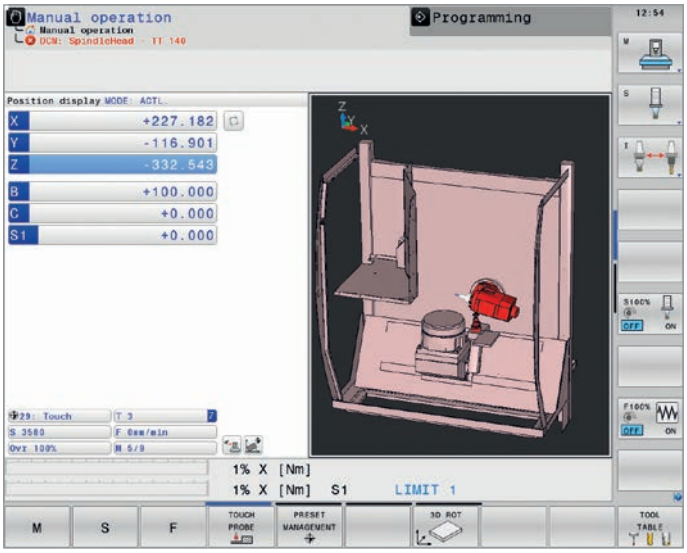
In cases such as these, the machine operator is supported by the **DCM** option (Dynamic Collision Monitoring) of the TNC. The control interrupts machining whenever a collision is imminent, thereby increasing the safety for the machine and its operator. This helps to prevent machine damage, which can result in costly downtimes. Unattended shifts become safer and more reliable.

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

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

As of NC software 34059x-05, the TNC 640 also offers a convenient way to import collision objects from standard CAD models (e.g., STL) into the control as M3D data. This provides a finely detailed image of the machine components and allows for a better utilization of the machine's work envelope.





DCM	Option 40	ID 526452-01
TNC 640 HSCI	As of NC SW 34059x-02	
TNC 620 HSCI	–	
TNC 320	–	
iTNC 530 HSCI	As of NC SW 60642x-01	
iTNC 530	As of NC SW 34049x-02	
Installation by the machine tool builder		
For more information, see the *iTNC 530* or *TNC 640* brochures		

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 feature allows you to define various coordinate transformations and settings with global and superimposed effect for the selected NC program without having to actually edit the program.

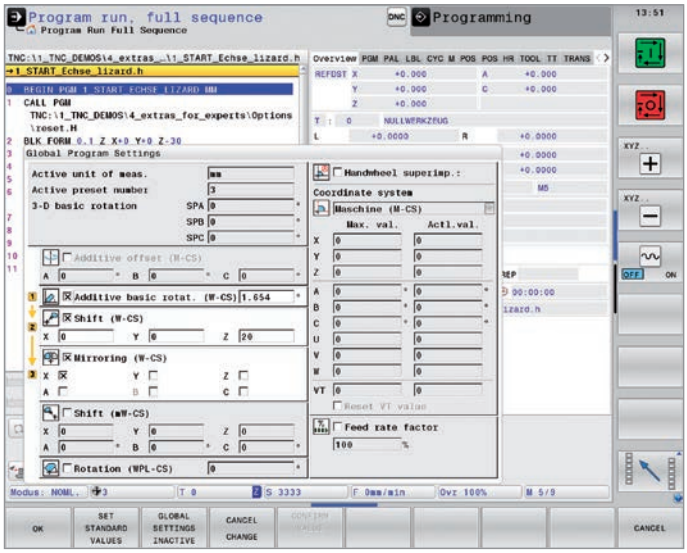
You can change the global program settings during a program stop, even in mid-program. A clearly structured form is provided for this. After program start the TNC then moves, if necessary, to a new position with a positioning logic influenced by you.

The following functions are available:
- Additional, additive datum shift
- Superimposed mirroring
- Handwheel superimposition with axis-specific memory of paths covered per handwheel, also in virtual axis direction
- Superimposed basic rotation
- Superimposed rotation
- Globally valid feed-rate factor
- Mirroring axes

Handwheel superimpositioning is possible in various coordinate systems:

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

You can select the desired coordinate system in an easy-to-read fillable form.



Global Program Settings	Option 44	ID 576057-01
TNC 640 HSCI	As of NC SW 34059x-08	
TNC 620 HSCI	–	
TNC 320	–	
iTNC 530 HSCI	As of NC SW 60642x-01	
iTNC 530	As of NC SW 34049x-03	
Installation by the machine tool builder		
More information	–	

Machining functions
AFC: adaptive feed control

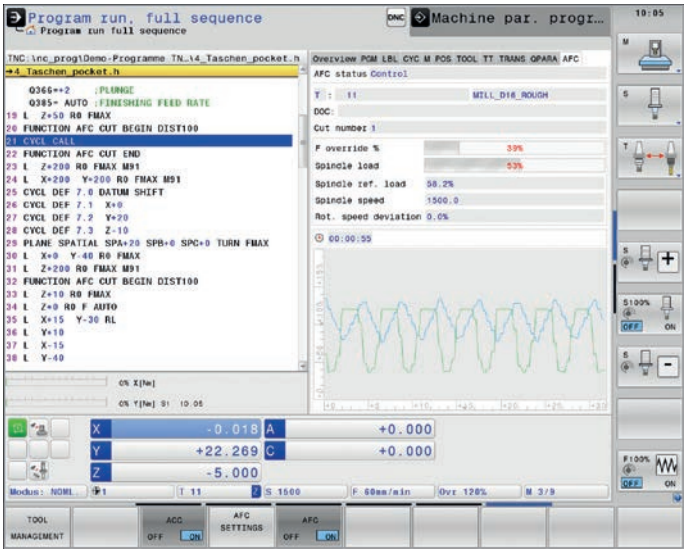
Adaptive Feed Control (AFC) automatically regulates the feed rate of the TNC, taking into consideration the respective spindle power and further process data. In a teach-in cut, the TNC records the maximum spindle power. Then, before actual machining, you define in a table the respective limit values between which the TNC will be permitted to influence the feed rate in "control" mode. Of course, various overload reactions can be specified and can also be defined by your machine tool builder.

The TNC's adaptive feed control offers various advantages:

Optimizing the machining time
More or less large fluctuations in the dimensions or the material (blowholes) can occur, especially on cast parts. Through corresponding control of the feed rate, the control tries to adhere to the previously "learned" maximum spindle power during the entire machining time. As a result of the increased feed rate in zones with less stock removal, the total machining time is shortened.

Tool monitoring
The adaptive feed control feature continuously compares the spindle power with the feed rate. As a tool becomes blunt, the spindle power increases. As a result, the TNC reduces the feed rate. As soon as the feed rate falls below a definable minimum, the TNC reacts with an NC stop or a warning, or it automatically performs a tool change with a replacement tool. This helps to prevent further damage after a tool breaks or is worn out.

Protection of the machine mechanics
Reducing the feed rate whenever the learned maximum permissible spindle power is exceeded also reduces strain and wear on the machine. The main spindle is protected effectively against overload.



dynamic efficiency

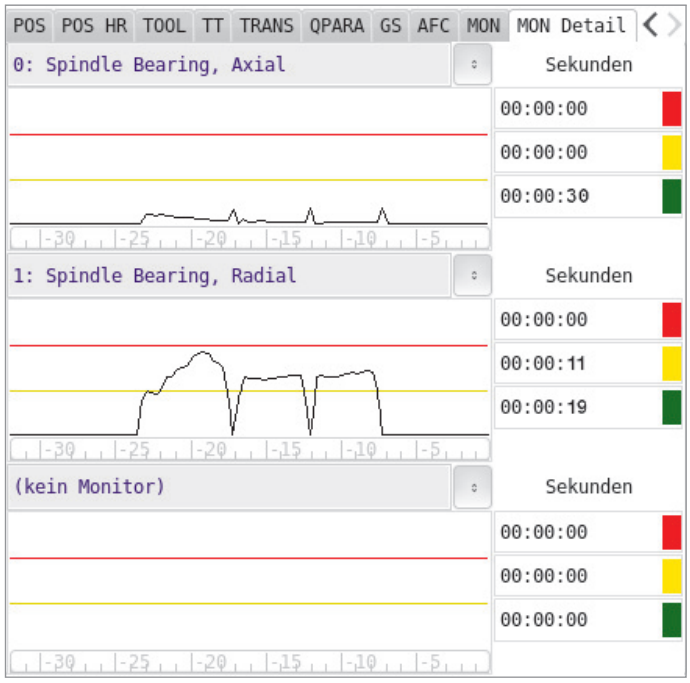
AFC: adaptive feed control	Option 45	ID 579648-01
TNC 640 HSCI	As of NC SW 34059x-02	
TNC 620 HSCI	–	
TNC 320	–	
iTNC 530 HSCI	As of NC SW 60642x-01	
iTNC 530	As of NC SW 34049x-03	
Installation by the machine tool builder		
For more information, see the *Dynamic Efficiency* Technical Information document		

Machining functions
Component Monitoring

Defective bearings on the main spindle are one of the most common causes of machine standstills. The bearings of the main spindle are often subjected to very high loads during the milling process. The spindle bearing may be damaged due to excessive continuous loading or short-term overloading. Whether the current machining process leads to damage of the spindle bearing is not easy for the machine operator to detect.

Component Monitoring (software option 155) allows you to use internal control signals to determine overloading and wear of machine components, and thus respond in time to avoid machine standstills. The machine tool builder provides formulas for the evaluation of the control signals in order to determine the load on the components.

In addition, the machine tool builder can also define warning and error thresholds as well as corresponding reactions when these thresholds are exceeded. This makes it possible to avoid greater damage.



dynamic + precision

Component Monitoring	Option 155	ID 1226833-01
TNC 640 HSCI	As of NC SW 34059x-09	
TNC 620 HSCI	As of NC SW 81760x-06	
TNC 320	–	
iTNC 530 HSCI	–	
iTNC 530	–	
Installation by the machine tool builder		
More information	–	

Machining functions

3D-ToolComp: 3-D radius compensation depending on the tool inclination angle

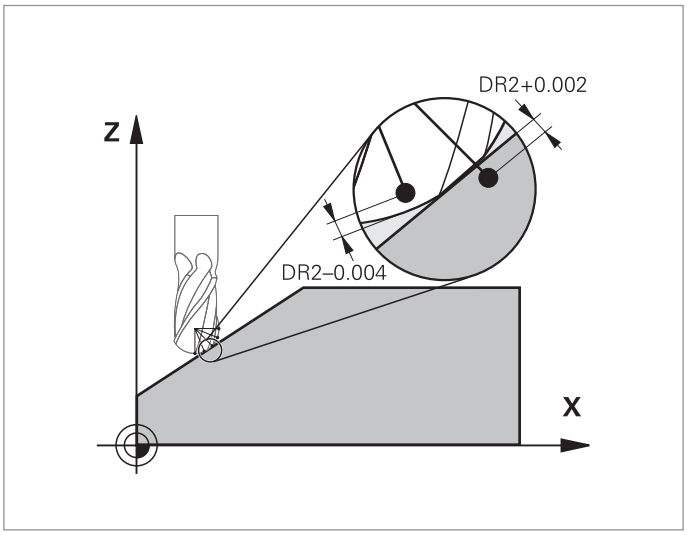
3D-ToolComp is a powerful option for three-dimensional tool radius compensation. A compensation-value table is used to define angle-dependent delta values that describe the tool deviation from an ideal circular form (see figure).

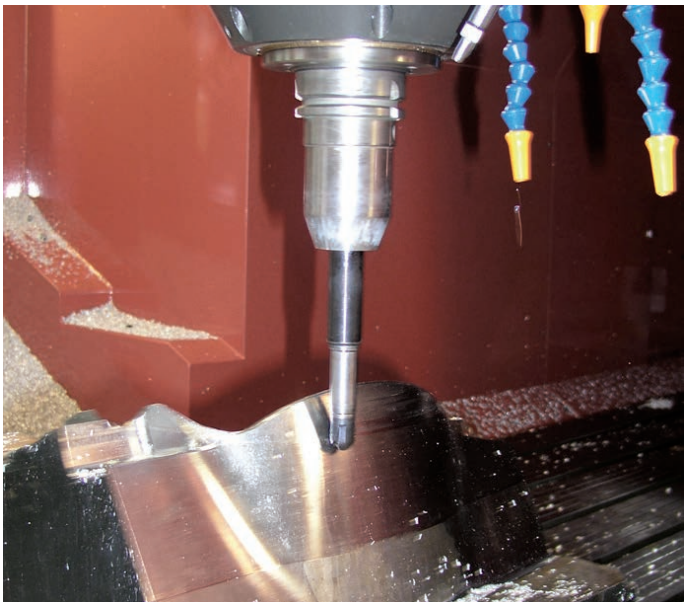
The TNC then corrects the radius value defined for the tool's current point of contact with the workpiece. In order to be able to determine the exact point of contact, the NC program needs to have been created with surface-normal blocks (LN blocks) by a CAM system. The surface-normal blocks specify the theoretical center point of the radius cutter, and in some cases also the tool orientation relative to the workpiece surface.

Ideally, you generate the compensation-value table fully automatically by measuring the tool with a laser system and a special cycle such that the TNC can use this table directly. If the form errors of the tool to be used are available to you in a calibration chart made by the tool manufacturer, then you can also create a compensation value table manually.

Measuring 3-D geometries
The TNC 640 also features a cycle with which you can measure points on 3-D geometries. In Cycle 444, 3D PROBING, you enter the respective measured point with its coordinates and the associated normal vectors. After probing, the TNC automatically calculates whether the measured point is within a preset tolerance.

You can interrogate the result through the system parameters in order, for example, to initiate program-controlled reworking. Moreover, you can trigger a program stop and a message. After measurement, the cycle automatically generates an easy-to-read measurement report in HTML format. To obtain even more accurate results, you can perform a 3-D calibration of the touch probe before running Cycle 444. Then the cycle compensates the touch probe's individual triggering behavior in any direction. Software option 92 (3D-ToolComp) is required for 3-D calibration.



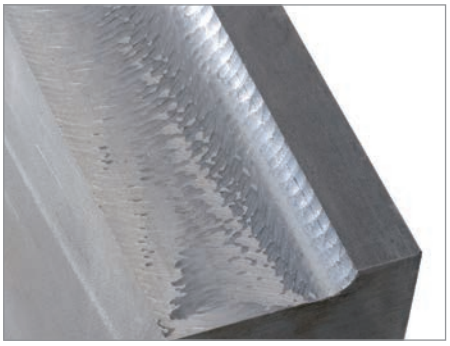


3D-ToolComp	Option 92	ID 679678-01
TNC 640 HSCI	As of NC SW 34059x-07	
TNC 620 HSCI	–	
TNC 320	–	
iTNC 530 HSCI	As of NC SW 60642x-01	
iTNC 530	As of NC SW 34049x-06	
Installation by the machine tool builder		
More information	–	

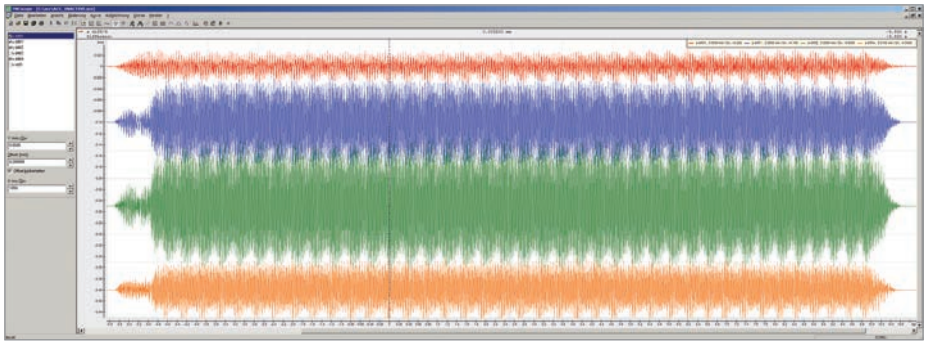
Machining functions
ACC: active chatter control

Strong forces come into play during roughing (power milling). Depending on the tool spindle speed, the resonances in the machine tool, and the chip volume (metal-removal rate during milling), the tool can sometimes begin to "chatter." Chatter subjects the machine to heavy strain and causes ugly marks on the workpiece surface. The tool, too, undergoes greater and irregular wear due to chatter. In extreme cases it can even break.

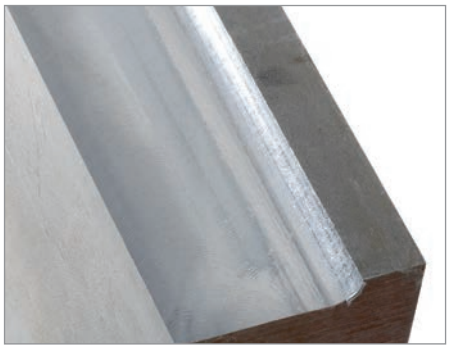
With **Active Chatter Control** (ACC), HEIDENHAIN now provides an effective control function for reducing a machine's tendency to chatter. The use of this control function is particularly advantageous during heavy machining. ACC makes substantially higher metal removal rates possible. This enables you to increase your metal removal rate by up to 25 % and more, depending on the type of machine. Thus, you can reduce the load on your machine while simultaneously increasing the life of your tools.



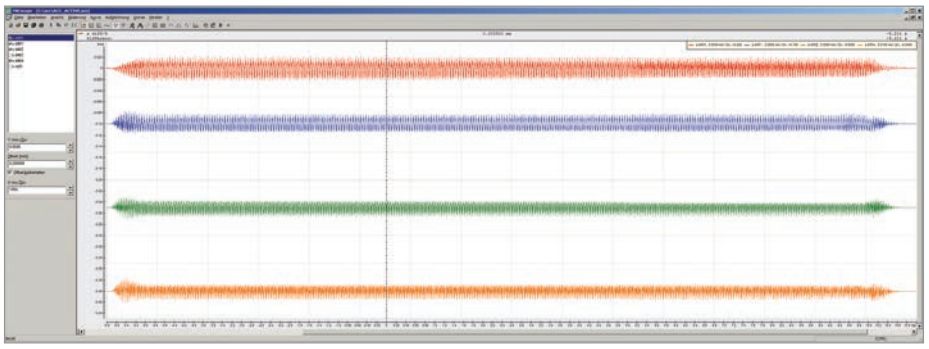
Heavy machining without ACC



Following error without ACC



Heavy machining with ACC



Reduced following error with ACC

dynamic efficiency

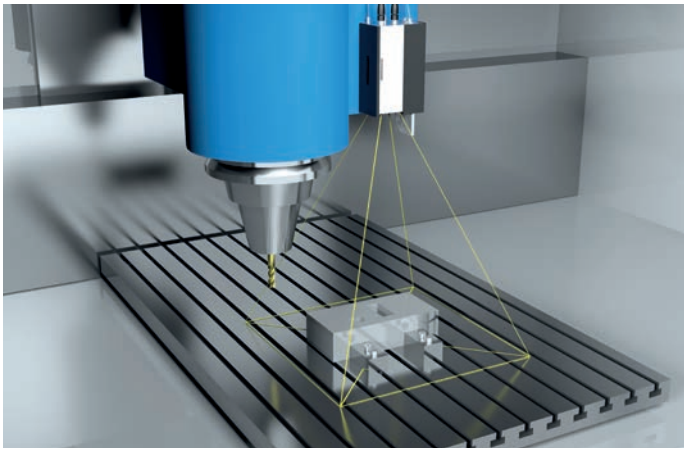
ACC	Option 145	ID 800547-01
TNC 640 HSCI	As of NC SW 34059x-02	
TNC 620 HSCI	As of NC SW 34056x-04/73498x-02/81760x-01	
TNC 320	–	
iTNC 530 HSCI	As of NC SW 60642x-03	
iTNC 530	–	
Installation by the machine tool builder		
For more information, see the *Dynamic Efficiency* Technical Information document		

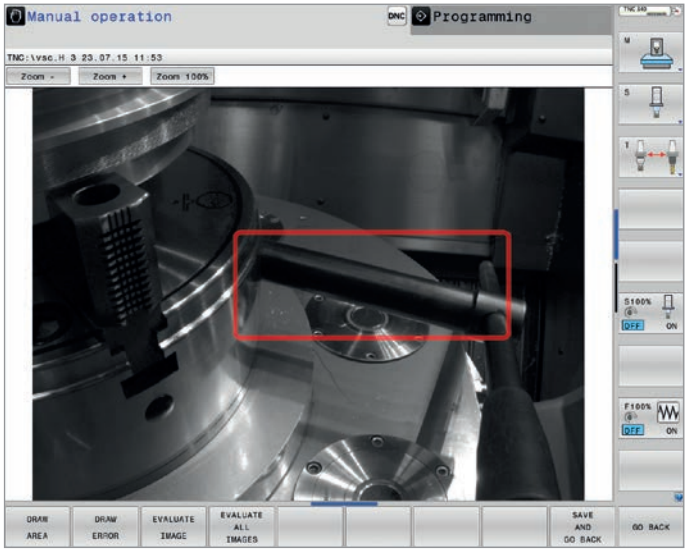
Machining functions
VSC: camera-based monitoring of the setup situation

With the **Visual Setup Control** option (VSC), the TNC can automatically monitor the current setup or machining situation during program run. With this option, reference photos are taken by the VS 101 camera system for the first parts of a series, which are then compared with the photos of the subsequent parts. User-friendly cycles enable you to specify multiple places in the NC program at which the control will perform an optical comparison of the nominal condition with the actual one. If an error is detected, the TNC reacts in a manner selected by the user.

VSC is capable of detecting the following situations:
- Missing machining operations or faulty workpieces
- Incorrectly positioned workpieces
- Incorrectly mounted or missing fixtures
- Residual chips (prior to measurements, for example)

The VSC option not only helps you to avoid expensive damage to the tool, workpiece, and machine. By taking and saving pictures, you can also use VSC to document the setup situation. By using VSC, you improve safety in the daily production process, as well as during unattended operation.



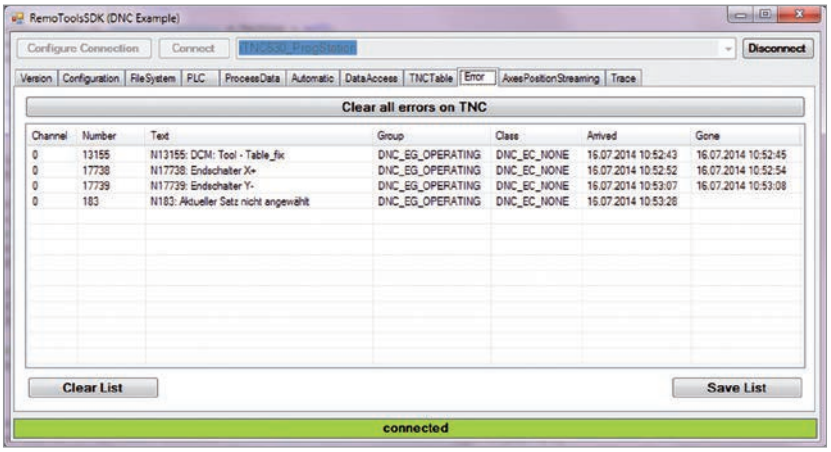


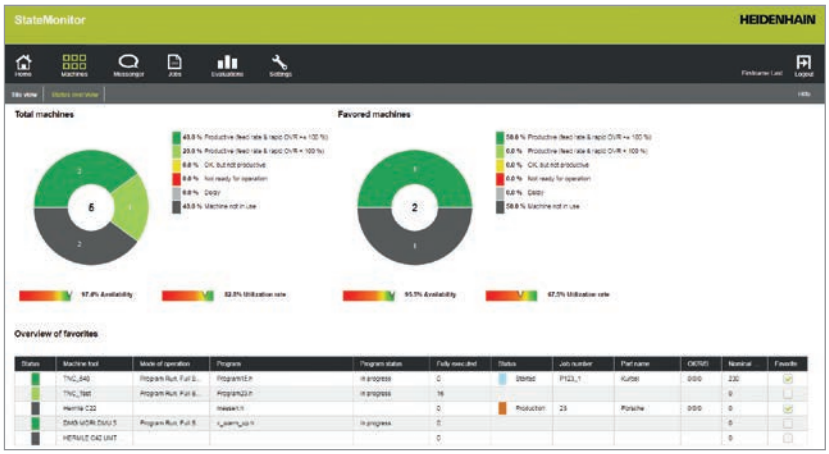
VSC	Option 136	ID 1099457-01
TNC 640 HSCI	As of NC SW 34059x-06	
TNC 620 HSCI	–	
TNC 320	–	
iTNC 530 HSCI	–	
iTNC 530	–	
Installation by the machine tool builder		
For more information, see the *TNC 640* brochure		

Communication
HEIDENHAIN DNC: communication over COM component

For meeting the requirements of the machine environment, the development environments on Windows operating systems are particularly well suited as a flexible platform for application development. The flexibility of the PC software and the large selection of ready-to-use software components and standard tools in the development environment enable you to develop PC applications of great use to your customers in very short time. The **HEIDENHAIN DNC** option enables a Windows application to access and edit data from the TNC as needed. Possible fields of application include, for example:

- Software solutions that control the manufacturing process
  - Machine and production data acquisition systems (MDA/PDA)
  - Connection to higher-level ERP/MES systems
  - Planning of preventive maintenance based on the actual condition of the machine
- Standard or customized PC software
  - Increase in process reliability and system availability
  - Error reporting systems that, for example, send the customer a message to his smartphone reporting problems with the currently running machining process
  - Overview plans that inform you about the current condition of all the machines used in production
  - Creation of a database for comprehensive data mining





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

connected + machining

HEIDENHAIN DNC	Option 18	ID 526451-01
RemoTools SDK	Accessory	ID 340442-xx
TNC 640 HSCI	As of NC SW 34059x-01	
TNC 620 HSCI	As of NC SW 34056x-01/73498x-01/81760x-01	
TNC 320	As of NC SW 34055x-01/771851-01	
iTNC 530 HSCI	As of NC SW 60642x-01	
iTNC 530	As of NC SW 34049x-01	
Installation by the machine tool builder		
For more information, see the *HEIDENHAIN DNC* brochure		

Communication

Remote Desktop Manager: display and remote operation of external computer units

In daily operations it can often be necessary to make entries in planning and control systems or to perform diagnostics using Windows-based software. The **Remote Desktop Manager** option provides the user with the opportunity to operate one or more Windows PCs directly from the TNC. It offers complete integration of Windows PC operation in the user interface of the TNC control's screen.

With a simple keystroke on the machine operating panel, you can switch between the control screen and the screen of a separate Windows PC in your local network. And it makes no difference whether the Windows PC is an industrial PC (e.g., the IPC 6641) in the machine's control cabinet or a server in the local network.

Potential applications include the central management of job orders, tools, and NC programs—all the way to remote operation of CAD/CAM systems from the machine. In this way, the machine tool operating panel becomes a flexible and efficient workplace for special manufacturing processes, including decentralized order processing.

Remote Desktop Manager can be set up through the control's operating system by IT specialists.

connected + machining

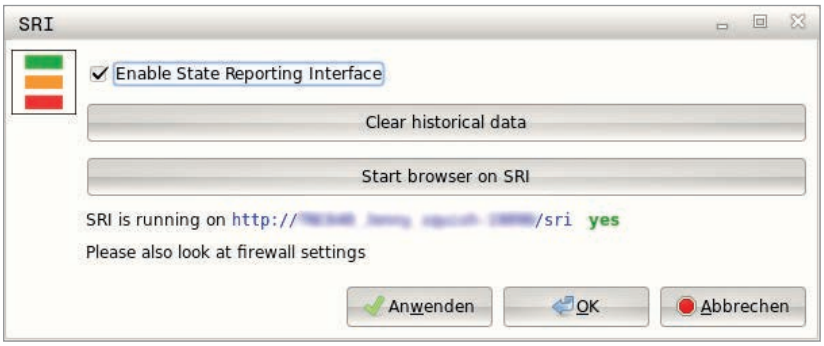
Remote Desktop Manager	Option 133	ID 894423-01
TNC 640 HSCI	As of NC SW 34059x-02	
TNC 620 HSCI	As of NC SW 81760x-02	
TNC 320	As of NC SW 77185x-04	
iTNC 530 HSCI	As of NC SW 60642x-02	
iTNC 530	–	
Installation by IT specialists		
For more information, see the Technical Manuals		

Communication
State Reporting Interface: production data acquisition

With production batch sizes becoming ever smaller and products becoming increasingly customized, systems for production data acquisition are becoming increasingly important. The operating resource data—as one of the most important parts of production data acquisition—describe the states of operating resources along a time scale. For machine tools, the idle times and run times as well as information about faults are recorded. An evaluation per workpiece can also be made that takes the active NC program into account.

One of the most common applications of production data acquisition is the determination of equipment effectiveness. The term Overall Equipment Effectiveness (OEE) is a measure of the value added by equipment. It provides information about the productivity of equipment as well as its losses at a glance.

With the **State Reporting Interface (SRI)**, HEIDENHAIN offers a simple and reliable interface for acquiring the operating states of your machines. In contrast to other interfaces, SRI also provides historical production data. Even if your company network fails for several hours, your valuable operating data will not be lost. A buffer memory that twice contains 10,000 entries is available for storing the historical operating states. In this context an entry corresponds to a status change.



connected machining

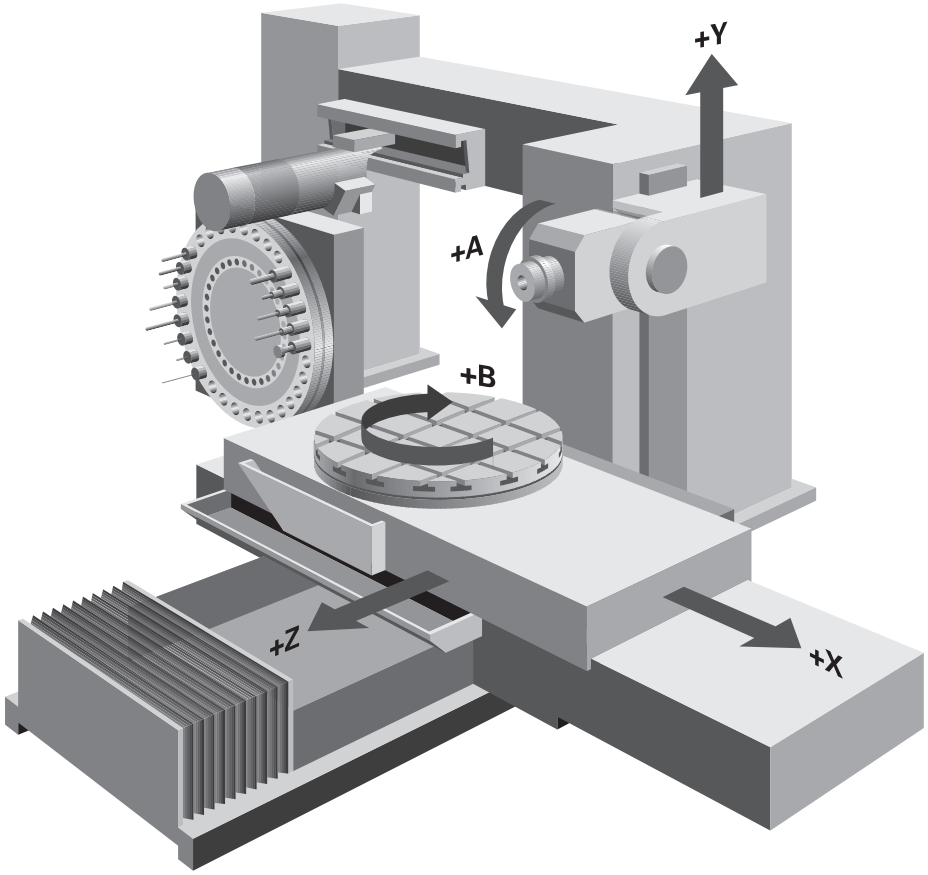
State Reporting Interface	Option 137	ID 1232242-01
TNC 640 HSCI	As of NC SW 34059x-09	
TNC 620 HSCI	As of NC SW 81760x-06	
TNC 320	77185x-06	
iTNC 530 HSCI	–	
iTNC 530	–	
Installation by the machine tool builder		
More information	–	

Interfacing to the machine
Additional Control Loops

The number of enabled control loops depends on the SIK or on additionally enabled control loops, which can also be ordered as needed later.

Further control loops can be enabled either as groups or individually. The combination of control-loop groups and individual control loops makes it possible to enable any number of control loops.

The maximum possible number of control loops depends on the control:
- iTNC 530: 20 control loops
- TNC 640: 24 control loops
- TNC 620: 8 control loops
- TNC 320: 6 control loops



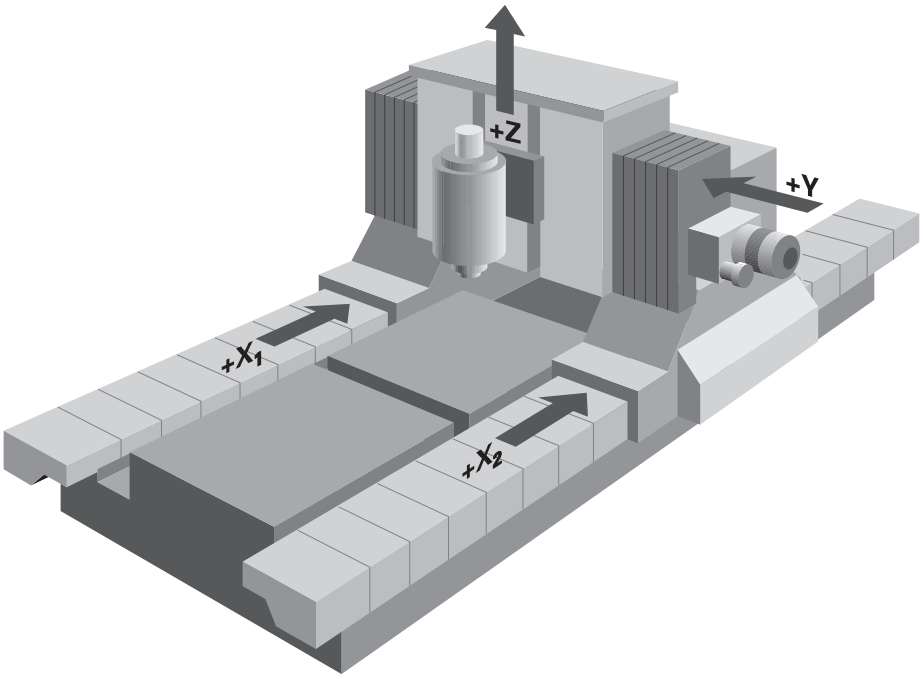
Individual control loops		ID
1st Additional Control Loop	Option 0	354540-01
2nd Additional Control Loop	Option 1	353904-01
3rd Additional Control Loop	Option 2	353905-01
4th Additional Control Loop	Option 3	367867-01
5th Additional Control Loop	Option 4	367868-01
6th Additional Control Loop	Option 5	370291-01
7th Additional Control Loop	Option 6	370292-01
8th Additional Control Loop	Option 7	370293-01
Control-loop groups		
4 Additional Control Loops	Option 77	634613-01
8 Additional Control Loops	Option 78	634614-01
TNC 640 HSCI	As of NC SW 34059x-01	
TNC 620 HSCI	As of NC SW 34056x-01/73498x-01/81760x-01	
TNC 320	As of NC SW 34055x-01/771851-01	
iTNC 530 HSCI	As of NC SW 60642x-01	
iTNC 530	As of NC SW 34049x-01	
Installation by the machine tool builder		
For more information, see the *Information for the Machine Tool Builder* brochures		

Interfacing to the machine
Synchronized Axes: gantry axes, tandem tables

Synchronized axes move in synchronism and are programmed with the same axis designation.

With HEIDENHAIN controls, parallel axis systems (gantry axes) such as on portal-type machines or tilting tables can be moved synchronously to each other through high-accuracy and dynamic position control. Rapid and especially precise positioning movements are coordinated to exactly match each other and permit 5-axis simultaneous movements for very demanding tasks. Multiple slave axes can be assigned to one master gantry axis.

Master-slave torque systems normally come into use when massive parts have to be moved or rack and pinion drive systems have to be prestressed for backlash-free motion. Up to six drives can be combined into one master-slave network and be flexibly stressed against each other. This makes it possible to realize fast and precise positioning movements even on large machine tools.



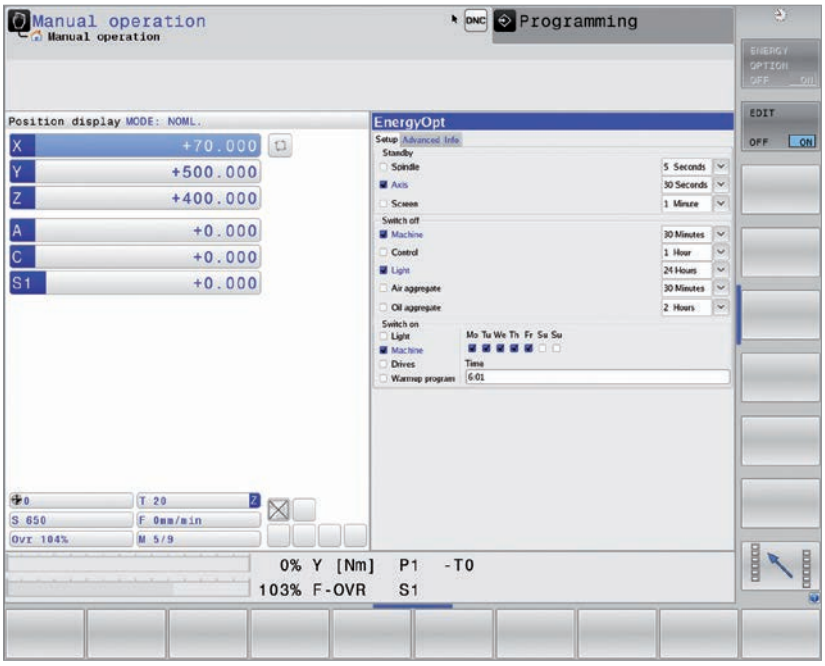
Synchronized Axes	Option 24	ID 634621-01
TNC 640 HSCI	Standard	
TNC 620 HSCI	As of NC SW 34056x-01/73498x-01/81760x-01	
TNC 320	As of NC SW 34055x-01/771851-01	
iTNC 530 HSCI	Standard	
iTNC 530	Standard	
Installation by the machine tool builder		
For more information, see the Technical Manuals		

Interfacing to the machine
Python OEM Process: realization of special functions

The **Python OEM Process** option is an effective tool for the machine tool builder to use an object-oriented high-level programming language in the control. Python is an easy-to-learn script language that supports the use of all necessary high-level language elements.

Python OEM Process can be universally used for machine functions and complex calculations, as well as to display special user interfaces. User-specific or machine-specific solutions can be efficiently implemented. A large number of functions on the basis of Python and GTK are available, regardless of whether you want to create special algorithms for special functions or separate solutions such as an interface for machine maintenance software.

You can use the PLC to integrate the applications you created and display them in the customary PLC windows. The applications can also be displayed in separate windows that are freely integrated in the TNC user interface and that can be made as large as the TNC screen itself.

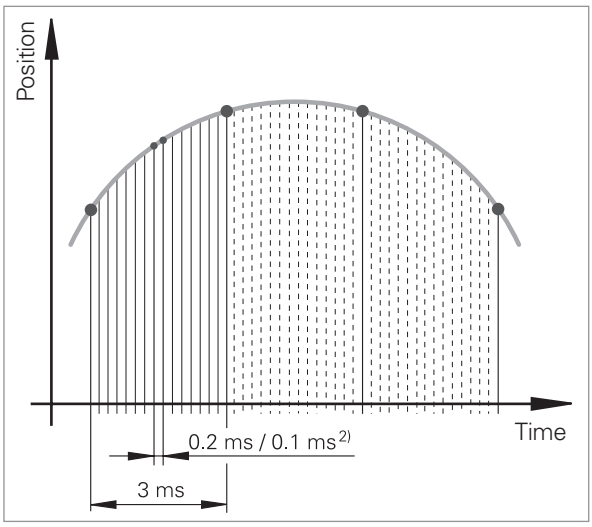


Python OEM Process	Option 46	ID 579650-01
TNC 640 HSCI	As of NC SW 34059x-01	
TNC 620 HSCI	As of NC SW 34056x-01/73498x-01/81760x-01	
TNC 320	As of NC SW 34055x-04/771851-01	
iTNC 530 HSCI	As of NC SW 60642x-01	
iTNC 530	As of NC SW 34049x-04	
Installation by IT specialists		
For more information, see the Technical Manuals		

Interfacing to the machine
Double Speed: short control-loop cycle times for direct drives

Single-speed control loops are usually sufficient for linear or torque motors and for conventional axes. **Double-speed control loops** are preferred for HSC spindles and axes that are difficult to control. In the default setting, 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. This requires option 49 to be enabled.

Double-speed control loops permit higher PWM frequencies as well as shorter cycle times for the speed controller. This makes improved current control for spindles possible as well as higher control performance for linear and torque motors.



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

Control loop cycle times
Fine interpolation
Single speed: 0.2 ms
Double speed: 0.1 ms (with option 49)

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

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

Current controller

f_{PWM}	T_{INT}
3 333 Hz	150 µs
4 000 Hz	125 µs
5 000 Hz	100 µs
6 666 Hz	75 µs with option 49
8 000 Hz	60 µs with option 49
10 000 Hz	50 µs with option 49

Double Speed Axes	Option 49	ID 632223-01
TNC 640 HSCI	As of NC SW 34059x-01	
TNC 620 HSCI	As of NC SW 34056x-01/73498x-01/81760x-01	
TNC 320	–	
iTNC 530 HSCI	As of NC SW 60642x-01	
iTNC 530	Standard	

Installation by the machine tool builder

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

Interfacing to the machine
OEM Option

Machine tools are often equipped by the manufacturer with useful and convenient additional functions that are saved in the control configuration (e.g., PLC). These functions are then offered to the user as options. To provide the user with the greatest possible flexibility in enabling these options, HEIDENHAIN provides a reserved range in the option menu (SIK menu) that can be used for the machine tool builder's purposes.

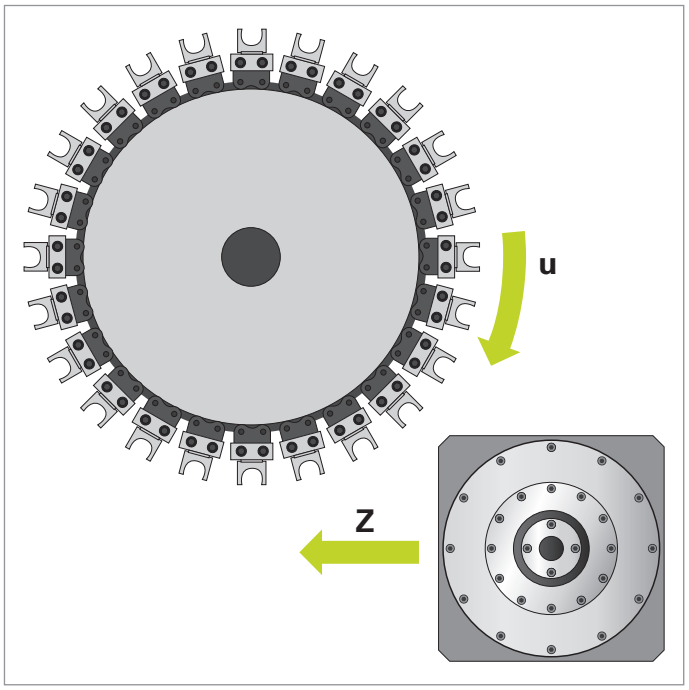
Options 101 to 130 provide 30 activatable options that the machine tool builder can have activated and that he can enable through his own PLC program via verification. One advantage is the simple activation procedure performed by the user over the SIK menu without needing on-site support by the machine tool builder.

OEM Option	Options 101 to 130	ID 579651-01 to ID 579651-30
TNC 640 HSCI	As of NC SW 34059x-02	
TNC 620 HSCI	–	
TNC 320	–	
iTNC 530 HSCI	As of NC SW 60642x-01	
iTNC 530	As of NC SW 34049x-06	
Installation by the machine tool builder		
More information	–	

Interfacing to the machine

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

The **Real-Time Coupling** function (RTC) allows the cyclic calculation of a position offset for an axis from the actual and nominal values of any other axes in the system. This enables you to realize complex simultaneous movements of several NC or PLC axes. The mutual dependence of the axes is defined in mathematical formulas. Applications include, for example, PLC axes whose movements need to be synchronized with those of an NC axis during a tool change in order to avoid collisions with the tool holders. The machine tool builder can use RTC to define these movements. The real-time coupling function enables you to realize complex traverse movements by coupling the principal and secondary axes. It therefore provides a number of new solutions—from process-specific movements to tool changes with specific requirements.



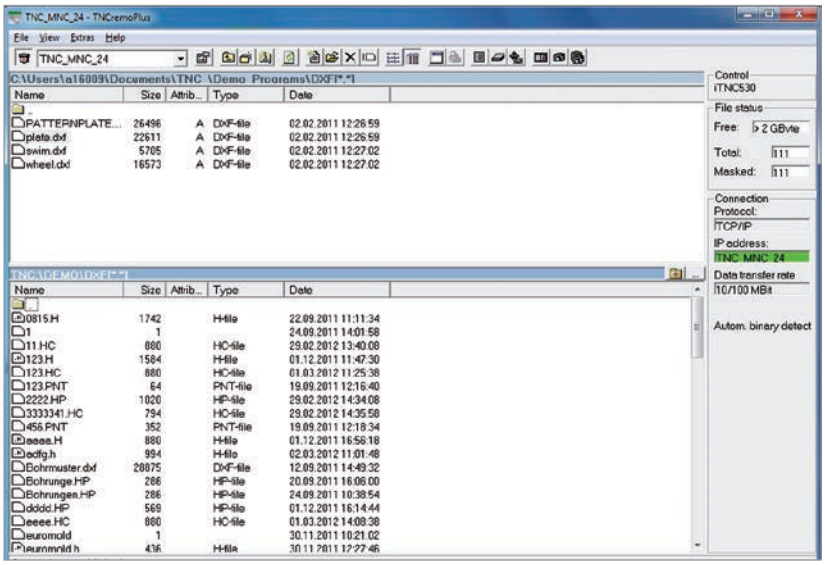
RTC	Option 135	ID 1085731-01
TNC 640 HSCI	As of NC SW 34059x-04	
TNC 620 HSCI	–	
TNC 320	–	
iTNC 530 HSCI	–	
iTNC 530	–	
Installation by the machine tool builder		
More information	–	

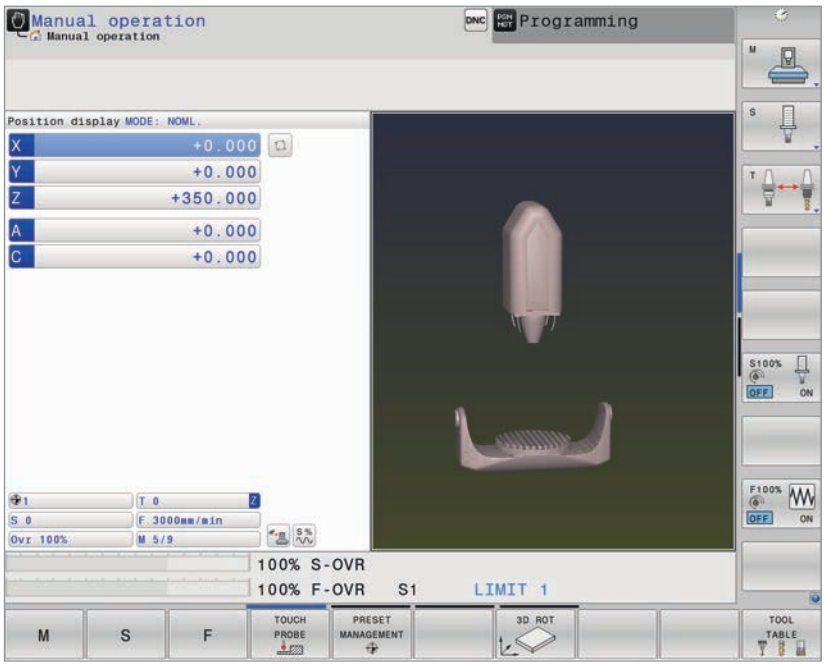
PC software
TNCremo: programs for data transfer

The free PC software package **TNCremo** supports the user during data transfer from the PC to the TNC. The software transfers data blockwise with block check characters (BCC). Using TNCremo and an Ethernet or other data interface, you can bidirectionally transfer externally saved part programs, tool tables, and pallet tables, and you can start the machine, create backups of the hard disk, and sample the operating condition of the machine.

Functions:
- Data transfer (also blockwise)
- Remote control (only serial)
- TNC file management
- TNC data backup
- Reading out of the log
- Print-out of screen contents
- Text editor
- Management of more than one machine

In addition to the features already familiar from TNCremo, **TNCremoPlus** can also transfer the current content of the control's screen to the PC (live screen). This makes it very simple to monitor the machine. TNCremo uses the LSV2 protocol to control the TNC remotely.





TNCremo	Free download
TNCremoPlus	ID 340447-xx
TNC 640 HSCI	As of NC SW 34059x-01
TNC 620 HSCI	As of NC SW 34056x-01/73498x-01/81760x-01
TNC 320	As of NC SW 34055x-01/771851-01
iTNC 530 HSCI	As of NC SW 60642x-01
iTNC 530	As of NC SW 34049x-01
Installation by the user	
More information	–

PC software
TeleService: remote diagnostics for HEIDENHAIN controls

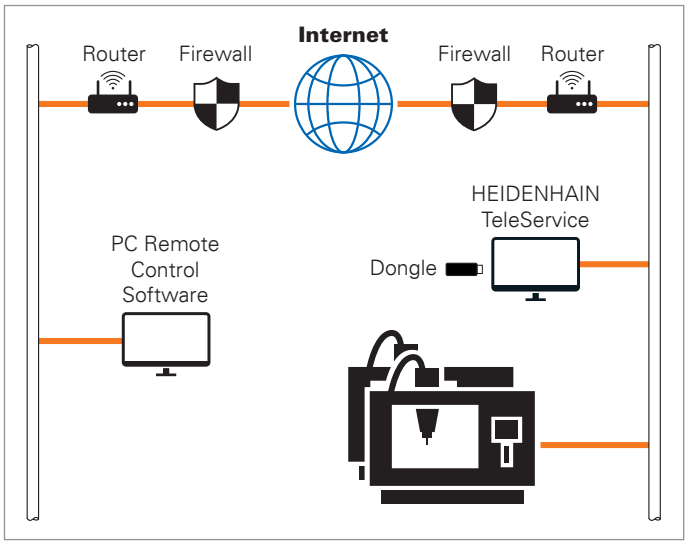
The PC software **TeleService** permits comprehensive remote diagnostics as well as extensive remote operation and remote monitoring of HEIDENHAIN controls. This makes in-depth troubleshooting possible. The service technician communicates with the control online, analyzes the control, and, if possible, repairs the issue immediately.

The machine tool builder creates the necessary diagnostic user interface according to his service requirements for checking the desired information. The TeleService control panel is used for operation.

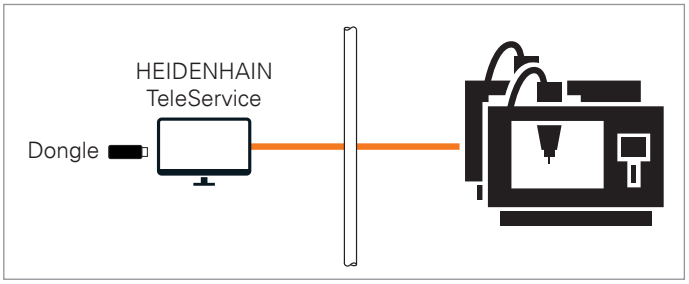
Functions
- Remote operation of the TNC with online screen transfer and virtual TNC keyboard
- Transmission of machining and PLC programs, machine parameters, tool and datum tables, etc.
- Display of machine and PLC data through TNCscope or TNCexplorer. The data are adapted to TNCexplorer by the machine tool builder through mask files
- Motor diagnostics with DriveDiag
- The machine tool builder can add his own applications to the TeleService control panel. HEIDENHAIN supplies the RemoTools SDK software development package for this purpose

TeleService by the machine tool builder
The machine manufacturer builds a network consisting of his machines equipped with TNCs on-site at the end customer and a network made up of service PCs (with TeleService installed) in the machine manufacturer's own service department. Routers connect the two networks over the public telephone and data network. When the customer presses the "Service" or "Support" soft key, the routers automatically establish a connection between the customer's network and that of the machine manufacturer. Through TeleService, the service technician has access to all of the machine data and PLC data saved on the control. The online screen transmission and a virtual TNC keyboard make the TNC completely remotely operable.

TeleService by the customer
TeleService can also be implemented within the customer's intranet. Here, a PC with TeleService installed on it is connected directly (without a router) to the network of TNCs. This enables remote operation, remote monitoring, and remote diagnostics of the machines within the customer's own network.



Remote connection over PC remote control software



TeleService in the company network

TeleService	CD-ROM with dongle
Single station license	ID 340449-xx
Network license for up to 14 participants	ID 340454-xx
Network license for up to 20 participants	ID 340455-xx
TNC 640 HSCI	As of NC SW 34059x-01
TNC 620 HSCI	As of NC SW 34056x-01/73498x-01/ 81760x-01
TNC 320	As of NC SW 34055x-01/771851-01
iTNC 530 HSCI	As of NC SW 60642x-01
iTNC 530	As of NC SW 34049x-01

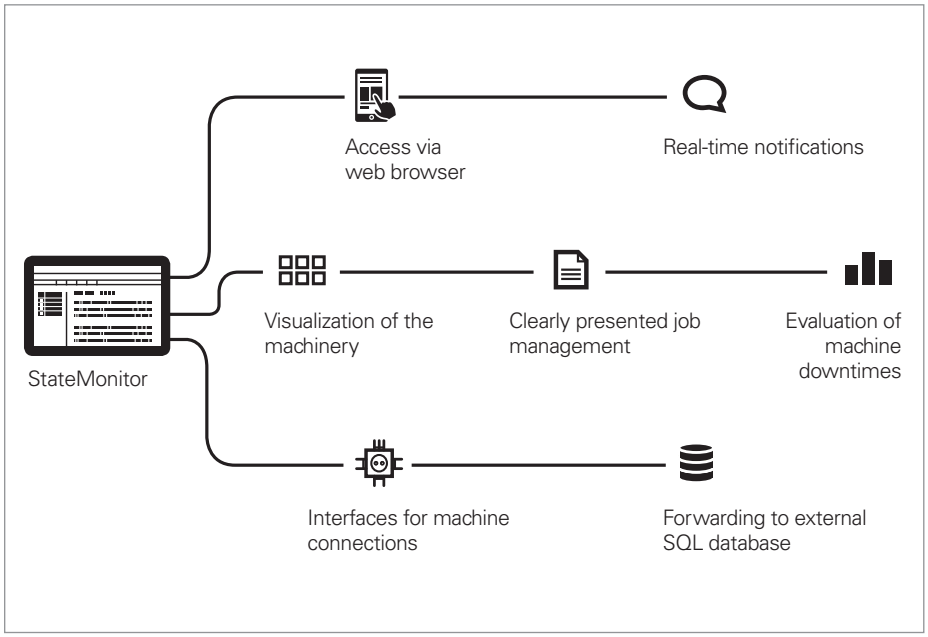
Installation by the machine tool builder

For more information, see the *Diagnostics for HEIDENHAIN Controls* Product Information document

PC software

StateMonitor: record, visualize, and evaluate machine data

StateMonitor records and visualizes the statuses of production machines. By evaluating important data such as the current machine status, machine messages, override positions, and utilization history, StateMonitor provides in-depth information on the machine's degree of utilization. StateMonitor also uses the collected data to reveal optimization possibilities. The user can enter comments about machine downtimes and setup times in order to uncover machine-specific and organizational optimization potential. Via the messenger function, StateMonitor notifies the responsible person by e-mail based on individually combinable machine signals and statuses.



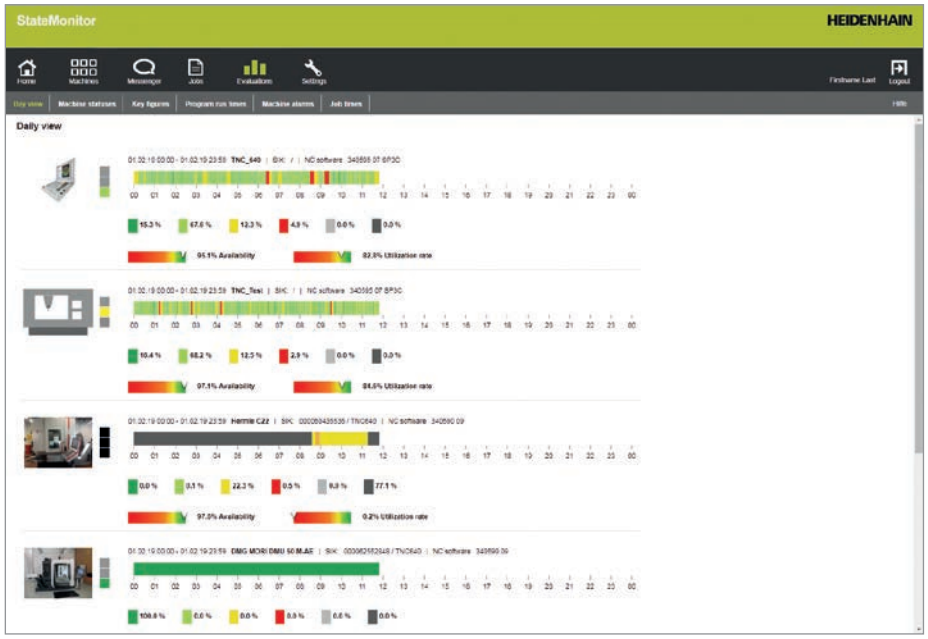
StateMonitor captures and visualizes the following information from the networked machines:

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

Active support for production planning with comprehensive functions for recording job data:

- Creating and assigning jobs
- Starting and terminating jobs
- Entering setup times and interruptions
- Storing additional job data, such as quantities produced

You can use StateMonitor to connect your machines with various types of controls. StateMonitor supports the protocol types HEIDENHAIN DNC, OPC UA, MTConnect, and Modbus TCP.



For more information, please contact HEIDENHAIN.

StateMonitor	ID 1218930-xx	**TNC 640 HSCI**	As of NC SW 34059x-01
StateMonitor for an additional five machines	ID 1220884-xx	**TNC 620 HSCI**	As of NC SW 34056x-01/73498x-01/ 81760x-01
Modbus Interface	ID 1268670-xx		
OPC UA Interface	ID 1268673-xx	**TNC 320**	As of NC SW 34055x-01/771851-01
JobTerminal	ID 1268674-xx	**iTNC 530 HSCI**	As of NC SW 60642x-01
MTConnect Interface	ID 1268675-xx	**iTNC 530**	As of NC SW 34049x-03
		Prerequisite: option 18 (DNC)	

Installation by the machine tool builder or end user

For more information, see the *Connected Machining* brochure and visit *www.klartext-portal.com*

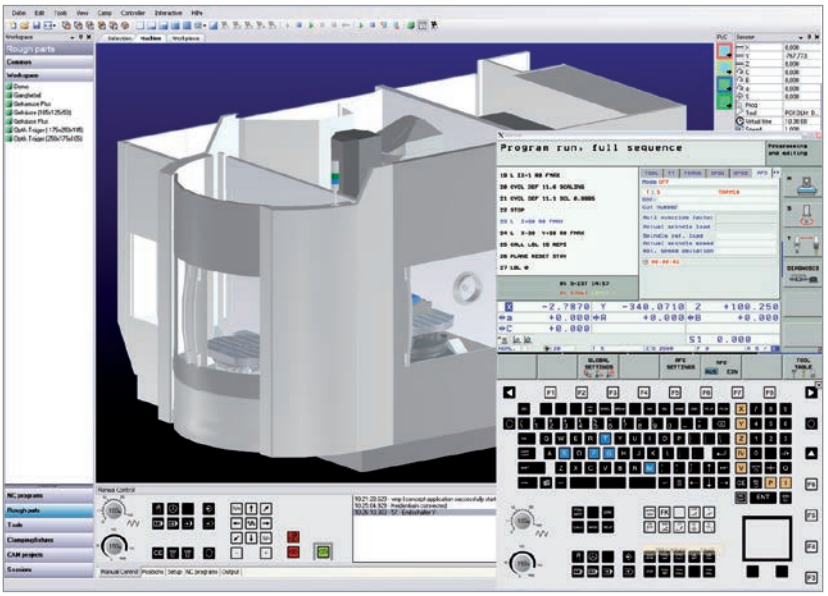
PC software
virtualTNC: control of virtual machines

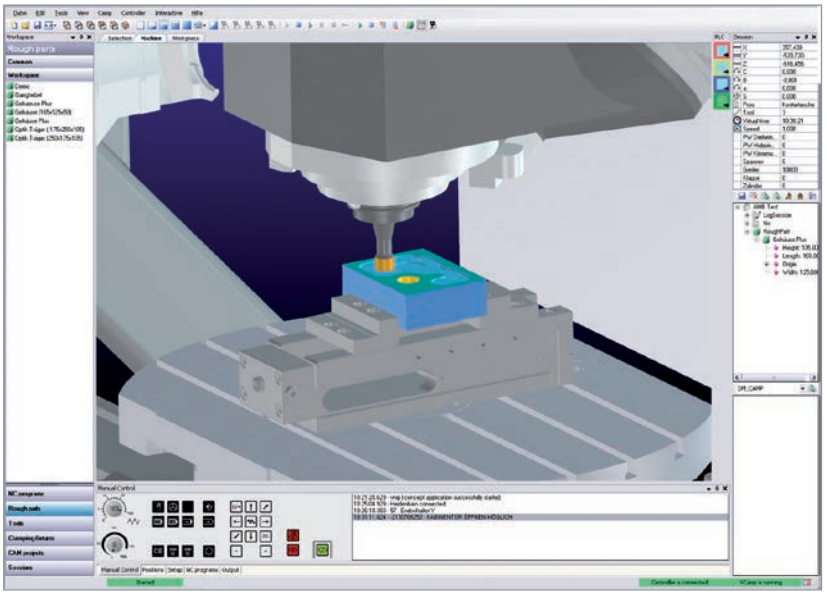
The simulation of NC programs on the control has been a feature of HEIDENHAIN contouring controls for some time now. The PC software **virtualTNC** now makes it possible to use the TNC as a control component for machine-simulation applications (virtual machine) on external computer systems.

How a virtual machine with virtualTNC works
Machine-simulation applications (virtual machines) can completely simulate production units in order to optimize production processes in the field ahead of time. 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 TNC.

The virtualTNC solution is the programming station software of the TNC with a special interface that enables the machine simulation software to identify the current axis positions of the running "virtual" control.

Interfacing of virtualTNC over HEIDENHAIN DNC
Software manufacturers who would like to simulate a production system can connect their virtual machine to virtualTNC over HEIDENHAIN DNC. The **COM component** (Object Axis Streaming) required for programming and configuring the interface to virtualTNC and its interface description are included in the **RemoTools SDK 3.0** software development package and its help system.





virtualTNC	TNC 640 HSCI	iTNC 530 HSCI iTNC 530
Single station license	ID 1113933-03	
Network license, 14 stations	ID 1113935-03	
Network license, 20 stations	ID 1113936-03	
Installation for manufacturers of machine-simulation applications		
For more information, see the *HEIDENHAIN DNC* brochure		

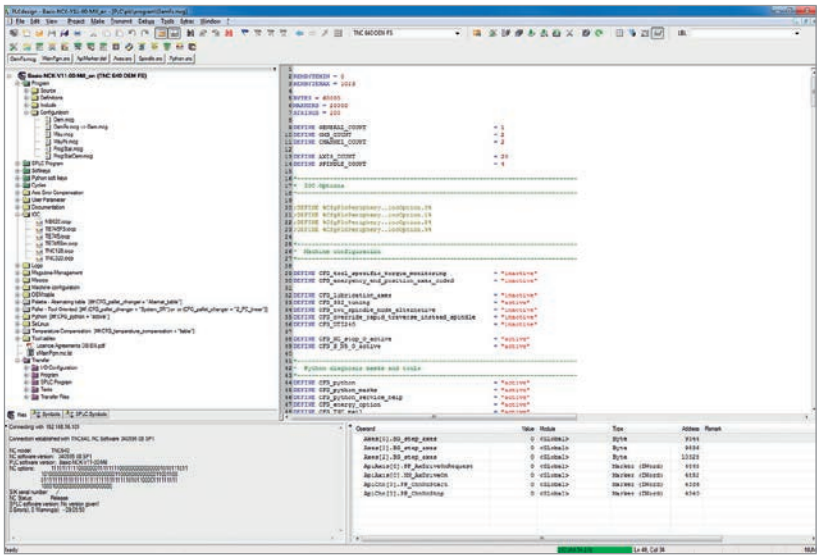
PC software

PLCdesign: software for PLC program development

The **PLCdesign** software can be used to easily create PLC programs. Extensive examples of PLC programs are included with the product.

Functions

- Easy-to-use text editor
- Menu-guided operation
- Programming of symbolic operands
- Modular programming method
- "Compiling" and "linking" of PLC source files
- Operand commenting, creation of a documentation file
- Comprehensive help system
- Data transfer between the PC and TNC
- Creation of PLC soft keys



PLCdesign	ID 284686-xx
TNC 640 HSCI	As of NC SW 34059x-01
TNC 620 HSCI	As of NC SW 34056x-01/73498x-01/81760x-01
TNC 320	As of NC SW 34055x-01/771851-01
iTNC 530 HSCI	As of NC SW 60642x-01
iTNC 530	As of NC SW 34049x-01
Installation for machine manufacturers and service	
For more information, see the *Information for the Machine Tool Builder* brochures	

PC software
KinematicsDesign: for creating machine kinematic models

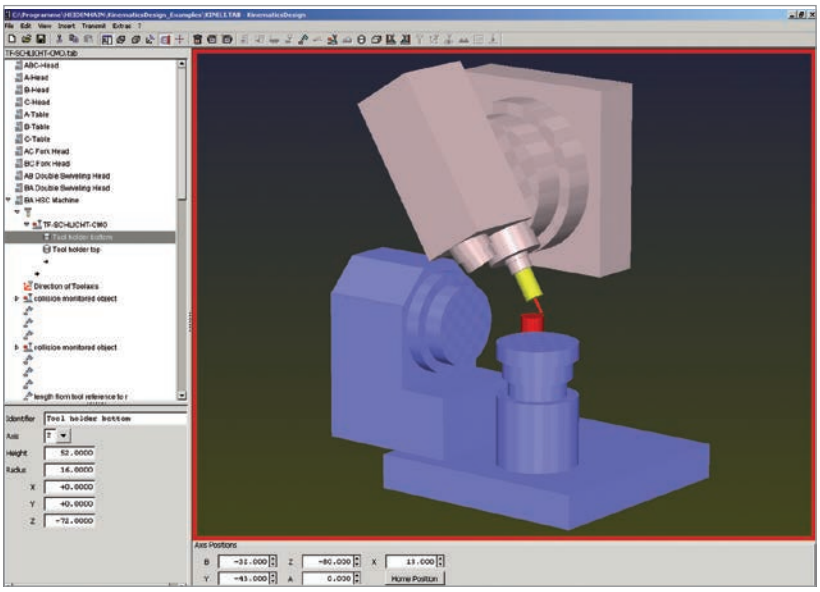
KinematicsDesign is a PC software application for the creation of flexible machine kinematic descriptions. KinematicsDesign also provides a convenient way to configure and commission the DCM collision monitoring function.

The software enables complete generation of the
- Assignment table
- Kinematics description table
- Kinematics subfile description table
- Tool-carrier kinematics description table
- Definition table for collision-monitored objects (CMOs) and the transfer of configuration files between control and PC

If KinematicsDesign is connected to a control online (operation is also possible with the programming station software of the TNC), then machine movements and the working space can be simulated when DCM is active and the axes are moved. Collisions that occur between defined machine objects or machine components in danger of collision are displayed in a color that you define.

The extensive visualization possibilities range from a depiction of the transformation chain by itself or a wire model all the way to a visualization of the entire work envelope.

With the TNC 640, you can transfer collision objects from a CAD file and integrate them into the machine kinematics using the M3D format. The M3D data format from HEIDENHAIN permits a very finely detailed depiction of high-resolution collision objects. The M3D converter, which is capable of performing tasks such as checking, repairing, simplifying, merging, and optimizing CAD data for collision objects, is used to generate the M3D data. As an independent PC tool, the M3D converter is part of the KinematicsDesign installation package (as of version 3.1). The M3D converter requires a software release module (ID 1124969-01).



KinematicsDesign	ID 340448-07
TNC 640 HSCI	As of NC SW 34059x-01
TNC 620 HSCI	As of NC SW 34056x-01/73498x-01/81760x-01
TNC 320	As of NC SW 34055x-01/771851-01
iTNC 530 HSCI	As of NC SW 60642x-01
iTNC 530	As of NC SW 34049x-01
Installation for machine manufacturers and service	
More information	–

PC software

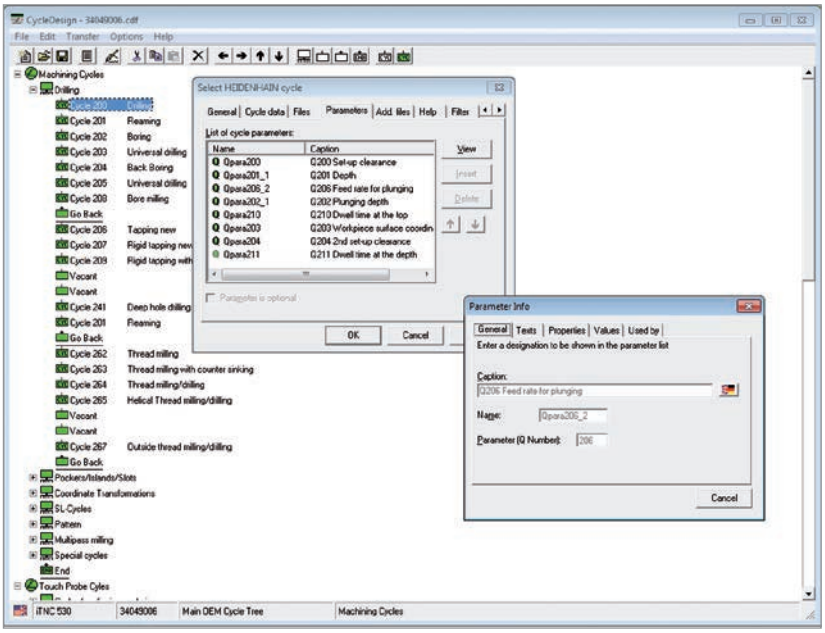
CycleDesign: saving NC subprograms as cycles

For frequently recurring operations, the HEIDENHAIN controls provide you with NC subprograms, also referred to as cycles, featuring configurable parameters. As you enter the parameters, the TNC supports you with prompts, questions, and help graphics.

You can call the cycles via soft keys. When the CYCL DEF key on the control is pressed, the soft-key row for HEIDENHAIN cycles appears.

With **CycleDesign** you can include your own NC subprograms as cycles in the soft-key structure of the control. You decide whether to add your cycles to the side of the HEIDENHAIN cycle row or to completely replace the HEIDENHAIN cycle row.

You use CycleDesign to transfer the cycle data to the control's memory

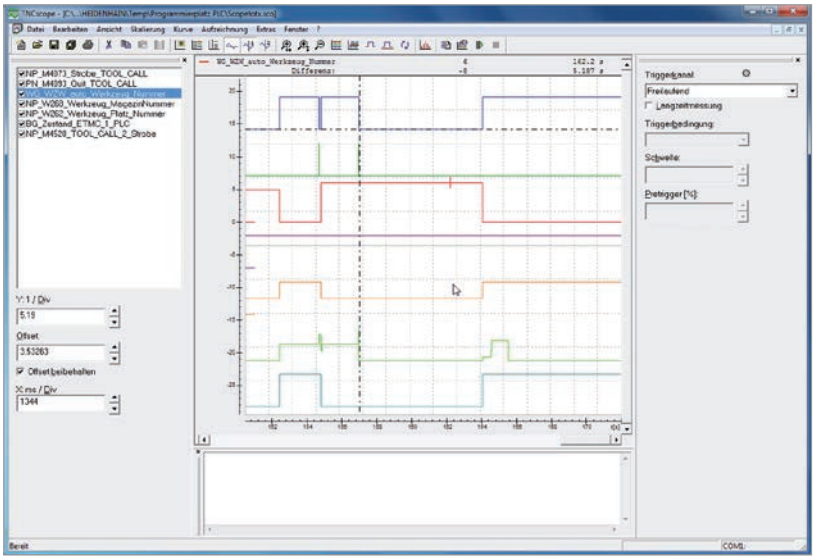


CycleDesign	Free download
TNC 640 HSCI	As of NC SW 34059x-01
TNC 620 HSCI	As of NC SW 34056x-01/73498x-01/81760x-01
TNC 320	As of NC SW 34055x-01/771851-01
iTNC 530 HSCI	As of NC SW 60642x-01
iTNC 530	As of NC SW 34049x-01
Installation for users and machine manufacturers	
For more information, see the *Information for the Machine Tool Builder* brochures	

PC software

TNCscope: for transferring the oscilloscope files to a PC

TNCscope is a PC oscilloscope software used to record and evaluate control signals and drive signals as well as PLC operands from HEIDENHAIN controls. TNCscope conveniently enables you to make a diagnosis from a Windows PC. Through a simple Ethernet connection, you can also use TNCscope to work with the control via remote maintenance. TNCscope can also display oscilloscope files that were recorded on the control (offline mode). Comprehensive mathematical functions are available for data evaluation.



- Multi-channel recording
- Various trigger possibilities
- Convenient evaluation of measured values
- Display of the control's circle form test and X/Y files
- Display of logic-trace and HSCI-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
- X/Y 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 for the configuration of various special functions of the control loop (PAC, LAC, CTC, MAC, ACC)

TNCscope	Download
TNC 640 HSCI	As of NC SW 34059x-01
TNC 620 HSCI	As of NC SW 34056x-01/73498x-01/81760x-01
TNC 320	As of NC SW 34055x-01/771851-01
iTNC 530 HSCI	As of NC SW 60642x-01
iTNC 530	As of NC SW 34049x-01
Installation for machine manufacturers and service	
For more information, see the Technical Manuals and the integrated help system	

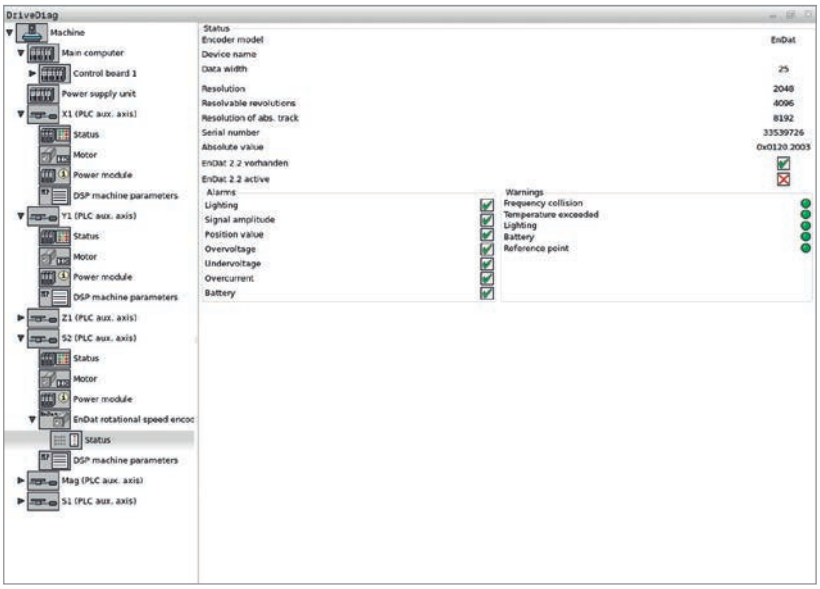
PC software
DriveDiag: diagnosis of digital drive systems

The **DriveDiag** software for PCs allows the service technician to quickly and easily diagnose the drives—from the motor all the way to the drive control. After a connection is established between DriveDiag and the control, various signals can be called from the control. In particular, with the dynamic display of status signals, you can even examine ambient conditions that lead to errors. DriveDiag can be used through the serial interface or Ethernet.

Functions

- Graphically supported, dynamic display of status signals
- Display of position encoder signals
- Display of analog signals that are also available to the controller (e.g., motor temperature and DC-link voltage, etc.)
- Display of speed encoder signals as well as monitoring of the motor's direction of rotation
- Test of the motor's power connection
- Automatic test for proper function of motors, inverters, position encoders, and speed encoders
- Reading and displaying the electronic ID labels of QSY motors with EQN 13xx or ECN 13xx as well as the inverter modules UVR 1xxD and UM 1xxD
- Displaying and evaluating the internal control conditions and the status signals of the inverter components
- Comprehensive help system

DriveDiag is available on the Internet for downloading as PC software (accessory) from the HEIDENHAIN Filebase. End users have read access, whereas the code number for the machine tool builder provides access to comprehensive testing possibilities with DriveDiag.



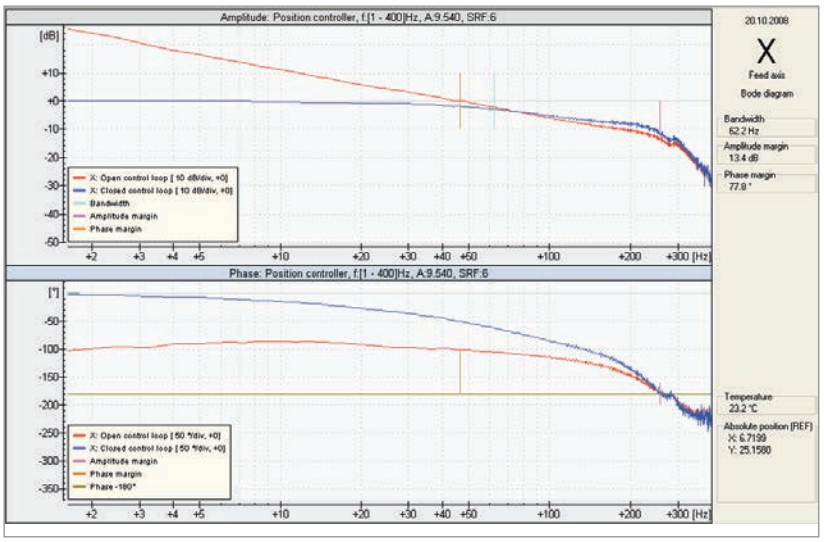
Graphically supported, dynamic display of status signals

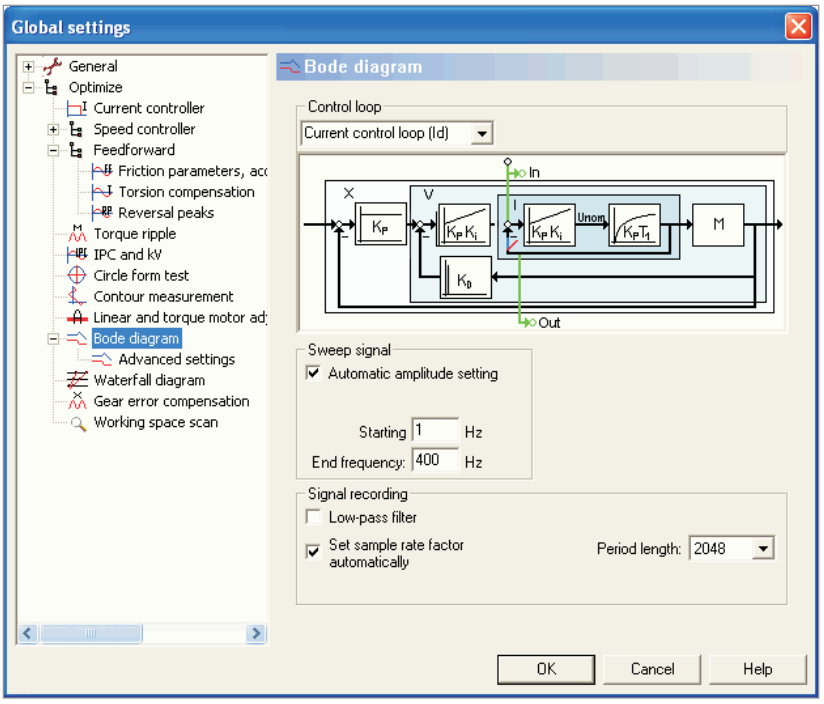
DriveDiag	Download
TNC 640 HSCI	As of NC SW 34059x-01
TNC 620 HSCI	As of NC SW 34056x-01/73498x-01/81760x-01
TNC 320	–
iTNC 530 HSCI	As of NC SW 60642x-01
iTNC 530	As of NC SW 34049x-01
Installation for machine manufacturers and service	
For more information, see the *Diagnostics for HEIDENHAIN Controls* Product Information document	

PC software
TNCopt: for commissioning digital control loops

High-end machine tools must operate with increasing speed and accuracy. High performance is demanded from the drive system with servo motors and spindles. This is why HEIDENHAIN assigns highest priority to efficient and intelligent control technology. This makes the correct functional and optimization settings in the control's servo loop particularly important. **TNCopt** helps you to easily maintain an overview and comply with the correct sequence during commissioning of all axes.

Functions:
- Commissioning of the current controller
- (Automatic) commissioning of the speed controller
- (Automatic) optimization of sliding-friction compensation
- (Automatic) optimization of the reversal spike compensation
- (Automatic) optimization of the k_V factor
- Circular interpolation test, contour test





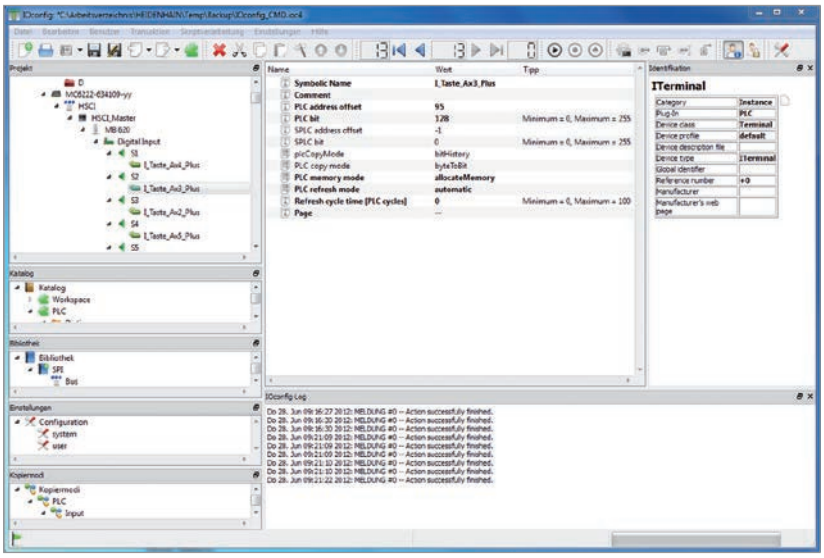
TNCopt	Download
TNC 640 HSCI	As of NC SW 34059x-01
TNC 620 HSCI	As of NC SW 34056x-01/73498x-01/81760x-01
TNC 320	–
iTNC 530 HSCI	As of NC SW 60642x-01
iTNC 530	As of NC SW 34049x-01
Installation for machine manufacturers and service	
For more information, see the *Information for the Machine Tool Builder* brochures	

PC software

IOconfig: configuration of the I/O and HSCI components

Modern machine tools are becoming ever more complex and extensively equipped. **IOconfig** supports you in the configuration of HSCI control components and peripheral devices (e.g., PROFIBUS, PROFINET) so that you can quickly and conveniently integrate all of the components (drive system, PLC) into the control system.

IOconfig can be integrated into PLCdesign's overall project and forms an essential component in the HSCI system during the commissioning of HSCI components and SPI additional modules.



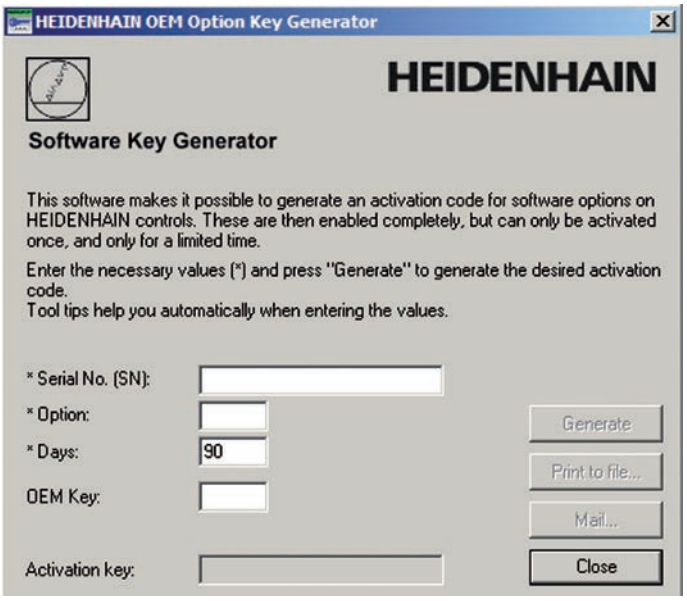
IOconfig	Download
TNC 640 HSCI	As of NC SW 34059x-01
TNC 620 HSCI	As of NC SW 34056x-01/73498x-01/81760x-01
TNC 320	As of NC SW 771851-01
iTNC 530 HSCI	As of NC SW 60642x-01
iTNC 530	As of NC SW 34049x-01
Installation for machine manufacturers and service	
For more information, see the Technical Manuals and the integrated help system	

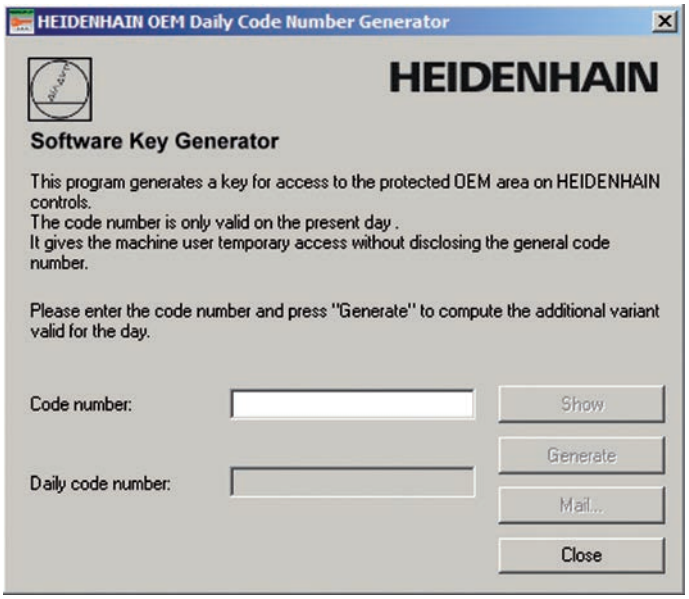
PC software
TNCkeygen: enabling keys for software options

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

OEM key generator
The OEM key generator PC tool makes it possible to generate an enabling key for software options on HEIDENHAIN controls. The selected option is enabled for a limited time (10 to 90 days) and can be enabled only once. You generate the desired enabling key by entering the SIK number, the option to be enabled, the duration, and a manufacturer-specific password. This allows the customer to extensively test the available options (e.g., the DXF Converter) without having to buy them. Upon successful testing, the customer can purchase the desired option from HEIDENHAIN.

OEM daily key generator
This application generates an enabling key for the protected area of the machine tool builder on HEIDENHAIN controls. The key gives the user access on the day the key is generated.





Software key generator	Free download
TNC 640 HSCI	As of NC SW 34059x-01
TNC 620 HSCI	As of NC SW 34056x-03/73498x-01/81760x-01
TNC 320	As of NC SW 34055x-05/771851-01
iTNC 530 HSCI	As of NC SW 60642x-01
iTNC 530	As of NC SW 34049x-04

Installation for the machine tool builder

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

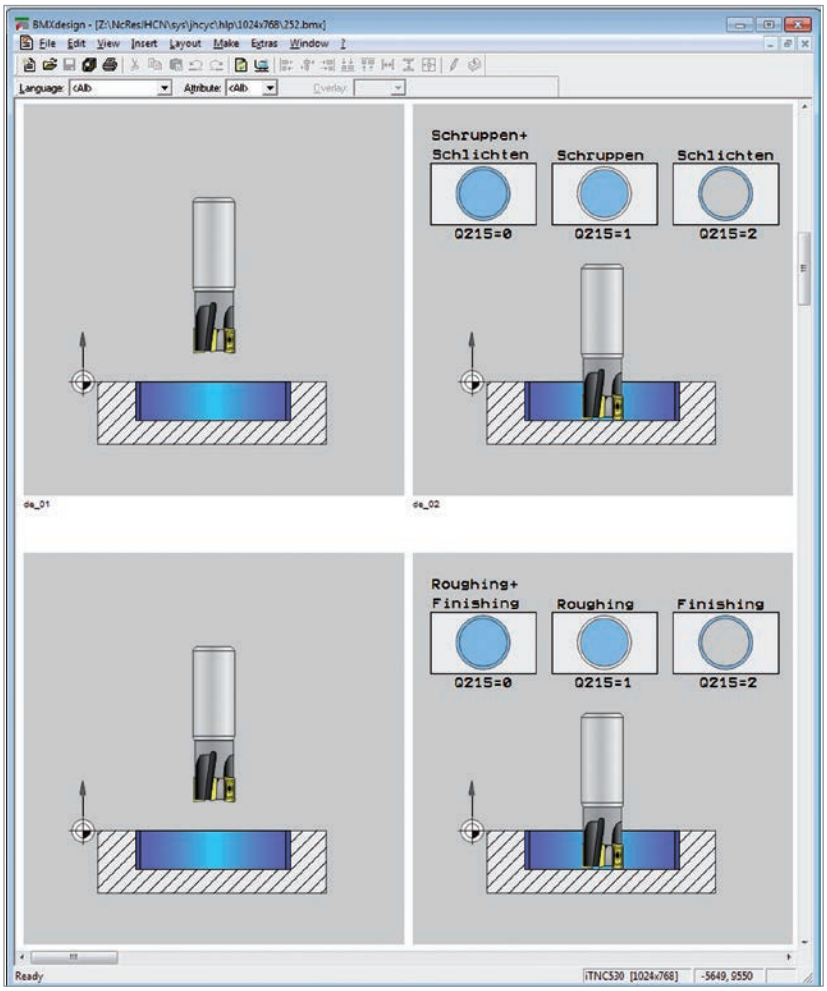
PC software
BMXdesign

BMXdesign is used to interactively create BMX project files and finished BMX files.

BMX files contain bitmap and text elements that the control can combine into variants during runtime. This permits the combination of help graphics or soft keys with language-sensitive texts or status-sensitive variants in only one 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 BPJ files (WYSIWYG)
- Display of BMX files as on the control
- Insertion of text fields and background bitmaps
- 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
- Bitmap (*.BMP) export function
- Integration of PLCtext for managing BMX files



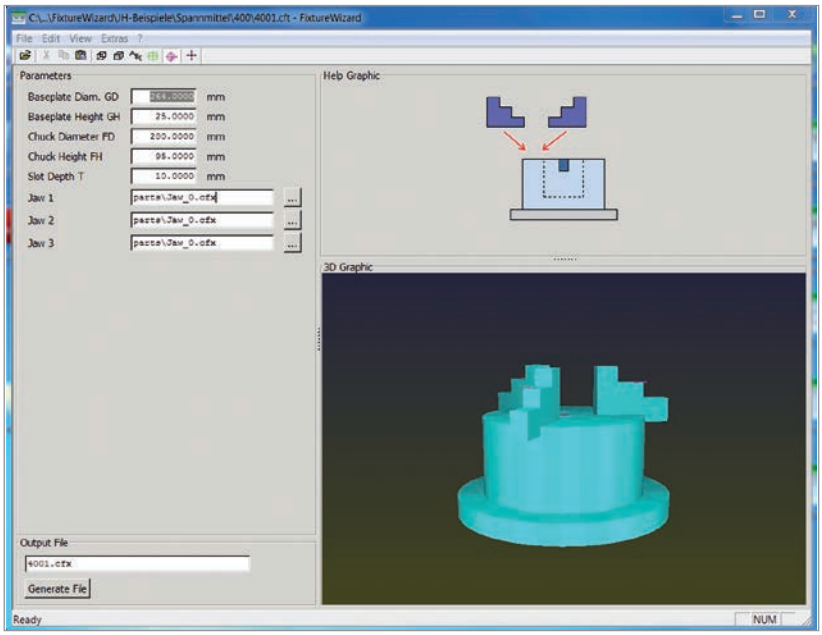
BMXdesign	ID 340443-xx
TNC 640 HSCI	As of NC SW 34059x-01
TNC 620 HSCI	As of NC SW 34056x-01/73498x-01/81760x-01
TNC 320	As of NC SW 34055x-01/771851-01
iTNC 530 HSCI	As of NC SW 60642x-01
iTNC 530	As of NC SW 34049x-01
Installation for the machine tool builder	
For more information, see the *Information for the Machine Tool Builder* brochures	

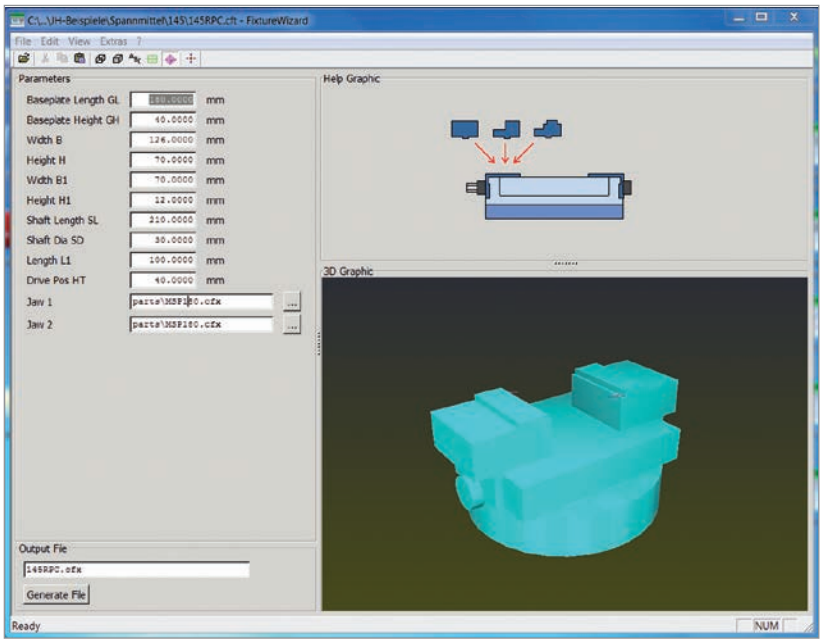
PC software
FixtureWizard

The **FixtureWizard** makes it easy for you to create specific kinematic models for fixtures and tool carriers based on fixture templates or tool carrier kinematics descriptions. You can then integrate the files you have created into the collision monitoring function of the TNC.

A large number of fixture templates are provided with the FixtureWizard. The installation program asks you whether you wish to include the accompanying fixture templates.

After opening a fixture template that matches your actual fixture, you simply enter the real dimensions in the corresponding dialog fields. The FixtureWizard immediately depicts the entered dimensions in the graphic window to help you avoid incorrect entries. Then you create the required control file, transfer it to the TNC, and use the fixture management to integrate the file into the collision monitoring function.





FixtureWizard	Free download
TNC 640 HSCI	–
TNC 620 HSCI	–
TNC 320	–
iTNC 530 HSCI	As of NC SW 60642x-01
iTNC 530	As of NC SW 34049x-05
Installation for users	
More information	–

PC software
Programming station

Why a programming station?
It is, of course, easy to write a part program with the TNC at the machine, even while the machine tool is machining a different part. Nevertheless, short reloading times and other machining tasks can often hinder any prolonged or concentrated programming work. With the programming station, you have the capability of programming just as you do at the machine, but away from all the noise and distractions of the shop floor.

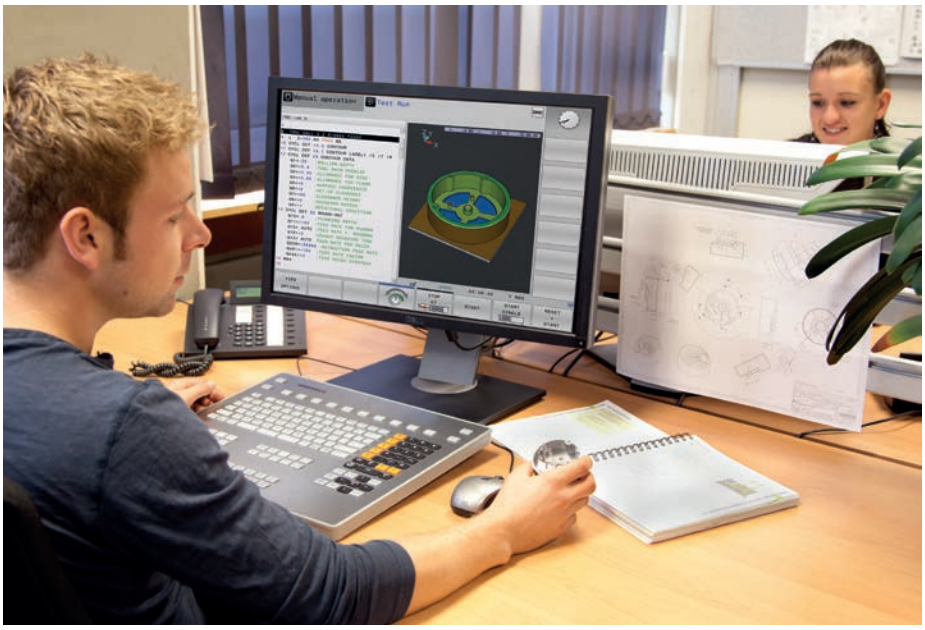
Creating programs
Programming, testing, and optimizing your smarT.NC programs (on the iTNC 530), HEIDENHAIN Klartext programs, or G-code programs for the TNC with the programming station substantially reduces machine idle times. This does not require a shift in thinking, since every keystroke feels the same. This is because, on the programming station, you program on the same keyboard as the one on the machine.

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

Training with the TNC programming station
Because the programming station is based on the same software as the TNC, it is ideally suited for apprentice and advanced training. The program is entered on the original keyboard unit. Even the Test Run mode functions exactly as it does on the machine. This gives the trainee the confidence needed to later operate the machine. Because it can be programmed with smarT.NC, in HEIDENHAIN Klartext format, and with G codes, the programming station can also be used in schools for TNC programming training.

The workstation
The programming station software runs on a PC. The programming station is only slightly different from a TNC built onto a machine tool. The TNC keyboard remains the same except that it now includes the soft keys that are normally integrated into the visual display unit. You connect the TNC keyboard to the USB port of your PC. The PC screen displays the familiar TNC user interface. Or as an alternative, you can even operate the programming station without a keyboard. You can use a virtual keyboard instead—it is displayed together with the TNC control panel and features the most important dialog initiation keys of the TNC.

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



Programming station	TNC 640	TNC 620/ TNC 320	iTNC 530
Demo version	ID 1114029-xx	ID 1114030-xx	ID 1114028-xx
With TNC operating panel	ID 1113967-02		
With virtual keyboard			
Single station license	ID 1113924-03		
Network license, 1 station	ID 1125955-03		
Network license, 14 stations	ID 1113926-03		
Network license, 20 stations	ID 1113928-03		
Operating panel without software release module	ID 1113967-52		
Installation by the user			
For more information, see the *Programming Stations* brochure			

Hardware enhancements
HR: electronic handwheel

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

You can move the axis slide via the feed motors in direct relation to the rotation of the handwheel. For delicate operations, you can set the transmission ratio incrementally to a defined distance per handwheel revolution.

HR 130 panel-mounted handwheels
The panel-mounted handwheels from HEIDENHAIN can be integrated in 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 in close proximity to the machine's working space. The axis keys and certain function keys are integrated in the housing. In this way you can switch axes and set up the machine at any time—regardless of where you happen to be standing. The integrated display of the HR 520 and HR 550 handwheels keeps you directly 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 unit with integrated charging device).

HR 130

HR 510

HR 520

HR 550

Electronic handwheel	With/without detent	TNC 640 HSCI As of NC SW	TNC 620 HSCI As of NC SW	TNC 320 As of NC SW	iTNC 530 HSCI As of NC SW	iTNC 530 As of NC SW
HR 130	ID 540940-01/540940-03	34059x-01	34056x-01/73498x-01/81760x-01	34055x-01	60642x-01	34049x-01
HR 510 **HR 510 FS** **HR 520** **HR 520 FS** **HR 550 FS**	ID 1120313-xx/1119971-xx ID 1119974-xx/1120311-xx ID 670303-xx/670302-xx ID 670305-xx/670304-xx ID 1183021-xx/1200495-xx	34059x-01 34059x-02 34059x-02 34059x-02 34059x-02	34056x-01/73498x-01/81760x-01 34056x-02/73498x-02/81760x-01 34056x-04/73498x-02/81760x-01 34056x-04/73498x-02/81760x-01 34056x-04/73498x-02/81760x-01	34055x-01 34055x-06 34055x-06 34055x-06 34055x-06	60642x-01 60642x-01 60642x-01 60642x-01 60642x-01	34049x-01 34049x-07 34049x-01 34049x-07 34049x-07
HRA 551 FS for HR 550 FS	ID 1119052-xx	34059x-02	34056x-04/73498x-02/81760x-01	34055x-06	60642x-01	34049x-07
Installation by the machine tool builder						
For more information, see the *Information for the Machine Tool Builder* brochures						

Hardware enhancements
TS: workpiece touch probes

Workpiece touch probes from HEIDENHAIN help you to reduce costs on the shopfloor and in series production. Together with the TNC, touch probes can manually or automatically perform setup, measuring, and inspection functions:
- Workpiece alignment
- Preset setting
- Workpiece measurement

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

Cable-connected touch probes
For machines with manual tool changing, 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 an automatic tool changer:
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 force, infrared transmission

Transceiver unit
Radio and infrared communication is established between the TS or TT touch probe and the SE transceiver.
SE 660 for radio and infrared transmission (hybrid technology); SE unit for both the TS 460 and TT 460.

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

SE 660

SE 661

TS 460

TS 260

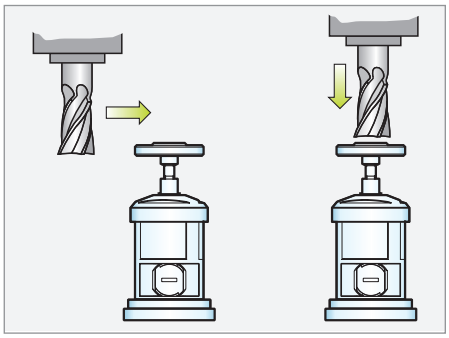
Workpiece touch probes		TNC 640 HSCI As of NC SW	TNC 620 HSCI As of NC SW	TNC 320 As of NC SW	iTNC 530 HSCI As of NC SW	iTNC 530 As of NC SW
TS 260	ID 738283-xx	34059x-02	81760x-02	771851-02	60642x-01	34049x-01
TS 460	ID 1178530-xx	34059x-01 SP1	81760x-01 SP1	771851-01	60642x-01	34049x-01
TS 740	ID 573757-xx	34059x-01	34056x-01/73498x-01/81760x-01	34055x-01	60642x-01	34049x-01
TS 642	ID 653217-xx	34059x-08	81760x-05	771851-05	60642x-01	34049x-01
Installation by the machine tool builder						
For more information, see the *Touch Probes* brochure						

Hardware enhancements
TT: tool touch probes

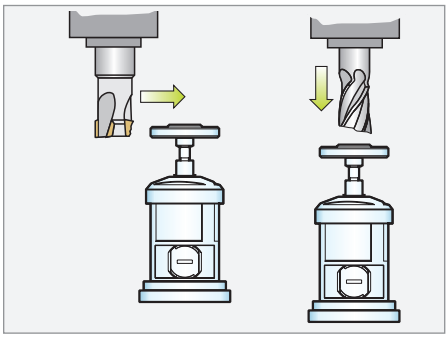
The tool is of course a decisive factor in ensuring a consistently high level of production quality. This means that an exact measurement of the tool dimensions and the periodic inspection of the tool for breakage, wear, and the shape of each tooth are necessary. For tool measurement, HEIDENHAIN offers the **TT triggering tool touch probes**.

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

The **TT tool touch probes** measure the tool length and radius. During probing of the rotating or stationary tool, such as for individual tooth measurement, the contact plate is deflected and a trigger signal is transmitted directly to the TNC. The **TT 160** uses cable-bound signal transmission, whereas the **TT 460** sends its signals via radio or infrared transmission. It is thus particularly well-suited for use on rotary 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		TNC 640 HSCI As of NC SW	TNC 620 HSCI As of NC SW	TNC 320 As of NC SW	iTNC 530 HSCI As of NC SW	iTNC 530 As of NC SW
TT 160 **TT 460**	ID 729763-xx ID 1192582-xx	34059x-05 34059x-04 SP1	81760x-02 81760x-01 SP1	771851-02 771851-01	60642x-01 60642x-01	34049x-01 34049x-05
Installation by the machine tool builder						
For more information, see the *Touch Probes* brochure						

Hardware enhancements
ITC: additional operating station

The additional ITC (Industrial Thin Client) operating stations from HEIDENHAIN are convenient solutions for an additional, remote station for operating the machine or a machine unit, such as a tool-changing station. Its remote operation design, which is tailored to the TNC, permits very simple connection of the ITC via a standard Ethernet connection with a cable length of up to 100 meters.

The **ITC 755** is a compact additional operating station for control systems with a 15-inch or 19-inch main screen. It provides the most important function keys of the TNC in addition to an ASCII keyboard and a touchscreen. The ITC 755 adjusts its resolution automatically to fit the size of the main screen. The soft keys are operated using the touchscreen.

Together with the TE 73x or TE 74x keyboard unit, the **ITC 750** (15-inch screen) or **ITC 860** (19-inch touchscreen) each make up a complete second operating station. They are operated in the same manner as the TNC. The ITC 860 can also be used as a second-screen solution for an extended workspace.

Convenience through plug-and-play technology
As soon as the TNC detects an ITC, the control provides the ITC with a current operating system. After booting of the ITC, the complete content of the TNC's screen is mirrored 1:1 on the ITC's screen. Due to this plug-and-play capability, it is not necessary for the hardware manufacturer to preconfigure the operating station. With a standard configuration of the X116 Ethernet port, the TNC automatically integrates the ITC into the system.

You can switch between the TNC and the ITC either by direct takeover or based on a handover principle (selectable). Both the booting and shutdown processes are entirely controlled by the TNC, thereby providing maximum operational reliability.

ITC 755

ITC 860

ITC 860
As "Extended Workspace" solution

Additional operating station		TNC 640 HSCI As of NC SW	TNC 620 HSCI As of NC SW	TNC 320 As of NC SW	iTNC 530 HSCI As of NC SW	iTNC 530 As of NC SW
ITC 755	ID 1039527-01	34059x-04	81760x-01	–	60642x-04	–
ITC 750	ID 1039544-01	34059x-04	81760x-01	–	60642x-04	–
ITC 860	ID 1174935-01	34059x-04 34059x-07 SP2	– –	– –	– –	– –
Installation by the machine tool builder						
More information			–			

Hardware enhancements
IPC: industrial PC

With the **IPC 6641** industrial PC you can start and remotely operate Windows-based applications via the TNC's user interface. The user interface is displayed on the control screen. Software option 133 is required for this.

Since Windows runs on the industrial PC, it does not influence 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 accesses.

In addition to the IPC 6641, a separately ordered hard disk is required for operation. The operating systems Windows 7, Windows 8, or Windows 10 can be installed on this empty data medium.

IPC 6641

Controls		**TNC 640 HSCI**	**TNC 620 HSCI**	**TNC 320**	**iTNC 530 HSCI**	**iTNC 530**
Industrial PC		Prerequisite: Remote Desktop Manager (option 133)				
IPC 6641 with 8 GB of RAM with 16 GB of RAM	 ID 1039543-01 ID 1039543-02					
HDR IPC	ID 1074770-51	Data medium for operating system				
Installation by the machine tool builder						
For more information, see the *Information for the Machine Tool Builder* brochures						

Hardware enhancements
VS 101: camera system for monitoring the working space

The **VS 101** camera system, in conjunction with software option 136 (Visual Setup Control) enables you to monitor the working space of your machine. The sealed and extremely sturdy VS 101 camera system is designed for integration in the machine's workspace. The protective housing features a closable cover and connections for sealing air to prevent damage to the camera optics. The VS 101 camera system can be connected directly to the control's main computer over a Gigabit Ethernet interface.

The closing cover and the sealing air are controlled via the TNC's integrated PLC. This enables you to optimally adapt the VSC inspection procedure to your actual conditions.

VS 101

Camera system for monitoring the working space	**TNC 640 HSCI** As of NC SW	**TNC 620 HSCI** As of NC SW	**TNC 320** As of NC SW	**iTNC 530 HSCI** As of NC SW	**iTNC 530** As of NC SW
VS 101 ID 1137063-01	340590-06	–	–	–	–
Installation by the machine tool builder					
More information		–			

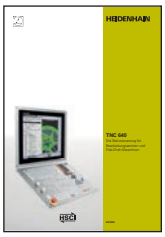
For more information

Brochures, data sheets, and CD-ROMs
The products shown here are described in more detail in separate documentation, including complete specifications, signal descriptions and dimension drawings in English and German (other languages available upon request).

HEIDENHAIN on the Internet
At www.heidenhain.de you will find not only these brochures in various languages, but also a great deal of further up-to-date information on the company and its products.

Our website also includes:
- Technical articles
- Press releases
- Addresses
- TNC training programs

Machine tool control



Brochures
TNC 128 Straight-Cut Control
TNC 320 Contouring Control
iTNC 530 Contouring Control
TNC 620 Contouring Control
TNC 640 Contouring Control

Contents:
Information for the user

Brochures
MANUALplus 620 Contouring Control
CNC PLIOT 640 Contouring Control

Contents:
Information for the user

OEM brochures
TNC 128 Straight-Cut Control
TNC 320 Contouring Control
iTNC 530 Contouring Control
TNC 620 Contouring Control
TNC 640 Contouring Control

Contents:
Information for the machine tool builder

OEM brochures
MANUALplus 620 Contouring Control
CNC PLIOT 640 Contouring Control

Contents:
Information for the machine tool builder

Angle measurement

Brochure
Rotary Encoders

Contents:
Absolute rotary encoders
ECN, EQN, ROC, ROQ
Incremental rotary encoders
ERN, ROD

Brochure
Encoders for Servo Drives

Contents:
Rotary encoders
Angle encoders
Linear encoders

Brochure
Modular Angle Encoders
With Magnetic Scanning

Contents:
Incremental encoders
ERM

Brochure
Modular Angle Encoders with Integral Bearing

Contents:
Absolute angle encoders
RCN, ECN
Incremental angle encoders
RON, RPN, ROD

Brochure
Modular Angle Encoders
With Optical Scanning

Contents:
Incremental angle encoders
ERP, ERO, ERA

Brochure
Angle Encoder Modules

Contents:
Angle encoder modules
MRP 2000/MRP 5000/MRP 8000
Angle encoder modules with integrated torque motor
SRP 5000, AccurET

Length measurement

Brochure
Linear Encoders
For Numerically Controlled Machine Tools

Contents:
Absolute linear encoders
LC
Incremental linear encoders
LB, LF, LS

Brochure
Exposed Linear Encoders

Contents:
Absolute linear encoders
LIC
Incremental linear encoders
LIP, PP, LIF, LIDA

Brochure
Length Gauges

Contents:
HEIDENHAIN-ACANTO
HEIDENHAIN-SPECTO
HEIDENHAIN-METRO
HEIDENHAIN-CERTO

Measured value acquisition and display

Brochure
Evaluation Electronics
For Metrology Applications

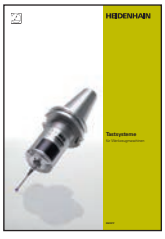
Contents:
ND, QUADRA-CHEK, MSE, EIB, IK

Product Overview
Interface Electronics

Brochure
Digital Readouts/Linear Encoders
For Manually Operated Machine Tools

Contents:
Digital readouts
ND, POSITIP
Linear encoders
LS

Setup and measurement



Brochure
Touch Probes

Contents:
Tool touch probes
TT
Workpiece touch probes
TS

Brochure
Measuring Devices For Machine Tool Inspection and Acceptance Testing

Contents:
Incremental linear encoders
KGM, VM

Connecting encoders and touch probes

Brochure
Cables and Connectors

Contents:
Technical properties, cable overviews, and cable lists

HEIDENHAIN

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

www.heidenhain.de

Vollständige und weitere Adressen siehe www.heidenhain.de
For complete and further addresses see www.heidenhain.de

DE **HEIDENHAIN Vertrieb Deutschland**
83301 Traunreut, Deutschland
☏ 08669 31-3132
FAX 08669 32-3132
E-Mail: hd@heidenhain.de

HEIDENHAIN Technisches Büro Nord
12681 Berlin, Deutschland
☏ 030 54705-240

HEIDENHAIN Technisches Büro Mitte
07751 Jena, Deutschland
☏ 03641 4728-250

HEIDENHAIN Technisches Büro West
44379 Dortmund, Deutschland
☏ 0231 618083-0

HEIDENHAIN Technisches Büro Südwest
70771 Leinfelden-Echterdingen, Deutschland
☏ 0711 993395-0

HEIDENHAIN Technisches Büro Südost
83301 Traunreut, Deutschland
☏ 08669 31-1337

AR **NAKASE SRL.**
B1653AOX Villa Ballester, Argentina
www.heidenhain.com.ar

AT **HEIDENHAIN Techn. Büro Österreich**
83301 Traunreut, Germany
www.heidenhain.de

AU **FCR MOTION TECHNOLOGY PTY LTD**
Laverton North Victoria 3026, Australia
E-mail: sales@fcrmotion.com

BE **HEIDENHAIN N.V.**
1760 Roosdaal, Belgium
www.heidenhain.be

BG **ESD Bulgaria Ltd.**
Sofia 1172, Bulgaria
www.esd.bg

BR **HEIDENHAIN Brasil Ltda.**
04763-070 – São Paulo – SP, Brazil
www.heidenhain.com.br

BY **GERTNER Service GmbH**
220026 Minsk, Belarus
www.heidenhain.by

CA **HEIDENHAIN CORPORATION**
Mississauga, OntarioL5T2N2, Canada
www.heidenhain.com

CH **HEIDENHAIN (SCHWEIZ) AG**
8603 Schwerzenbach, Switzerland
www.heidenhain.ch

CN **DR. JOHANNES HEIDENHAIN (CHINA) Co., Ltd.**
Beijing 101312, China
www.heidenhain.com.cn

CZ **HEIDENHAIN s.r.o.**
102 00 Praha 10, Czech Republic
www.heidenhain.cz

DK **TP TEKNIK A/S**
2670 Greve, Denmark
www.tp-gruppen.dk

ES **FARRESA ELECTRONICA S.A.**
08028 Barcelona, Spain
www.farresa.es

FI **HEIDENHAIN Scandinavia AB**
01740 Vantaa, Finland
www.heidenhain.fi

FR **HEIDENHAIN FRANCE sarl**
92310 Sèvres, France
www.heidenhain.fr

GB **HEIDENHAIN (G.B.) Limited**
Burgess Hill RH15 9RD, United Kingdom
www.heidenhain.co.uk

GR **MB Milionis Vassilis**
17341 Athens, Greece
www.heidenhain.gr

HR Croatia → **SL**

HU **HEIDENHAIN Kereskedelmi Képviselet**
1239 Budapest, Hungary
www.heidenhain.hu

ID **PT Servitama Era Toolsindo**
Jakarta 13930, Indonesia
E-mail: ptset@group.gts.co.id

IL **NEUMO VARGUS MARKETING LTD.**
Holon, 58859, Israel
E-mail: neumo@neumo-vargus.co.il

IN **HEIDENHAIN Optics & Electronics India Private Limited**
Chetpet, Chennai 600 031, India
www.heidenhain.in

IT **HEIDENHAIN ITALIANA S.r.l.**
20128 Milano, Italy
www.heidenhain.it

JP **HEIDENHAIN K.K.**
Tokyo 102-0083, Japan
www.heidenhain.co.jp

KR **HEIDENHAIN Korea LTD..**
Gasan-Dong, Seoul, Korea, 153-782
www.heidenhain.co.kr

MX **HEIDENHAIN CORPORATION MEXICO**
20290 Aguascalientes, AGS., Mexico
E-mail: info@heidenhain.com

MY **ISOSERVE SDN. BHD.**
43200 Balakong, Selangor
E-mail: sales@isoserve.com.my

NL **HEIDENHAIN NEDERLAND B.V.**
6716 BM Ede, Netherlands
www.heidenhain.nl

NO **HEIDENHAIN Scandinavia AB**
7300 Orkanger, Norway
www.heidenhain.no

NZ **Llama ENGINEERING Ltd**
5012 Wellington, New Zealand
E-mail: info@llamaengineering.co.nz

PH **MACHINEBANKS' CORPORATION**
Quezon City, Philippines 1113
E-mail: info@machinebanks.com

PL **APS**
02-384 Warszawa, Poland
www.heidenhain.pl

PT **FARRESA ELECTRÓNICA, LDA.**
4470 - 177 Maia, Portugal
www.farresa.pt

RO **HEIDENHAIN Reprezentanţă Romania**
Braşov, 500407, Romania
www.heidenhain.ro

RS Serbia → **BG**

RU **OOO HEIDENHAIN**
115172 Moscow, Russia
www.heidenhain.ru

SE **HEIDENHAIN Scandinavia AB**
12739 Skärholmen, Sweden
www.heidenhain.se

SG **HEIDENHAIN PACIFIC PTE LTD**
Singapore 408593
www.heidenhain.com.sg

SK **KOPRETINA TN s.r.o.**
91101 Trencin, Slovakia
www.kopretina.sk

SL **NAVO d.o.o.**
2000 Maribor, Slovenia
www.heidenhain.si

TH **HEIDENHAIN (THAILAND) LTD**
Bangkok 10250, Thailand
www.heidenhain.co.th

TR **T&M Mühendislik San. ve Tic. LTD. ŞTİ.**
34775 Y. Dudullu –
Ümraniye-Istanbul, Turkey
www.heidenhain.com.tr

TW **HEIDENHAIN Co., Ltd.**
Taichung 40768, Taiwan R.O.C.
www.heidenhain.com.tw

UA **Gertner Service GmbH Büro Kiev**
02094 Kiev, Ukraine
www.heidenhain.ua

US **HEIDENHAIN CORPORATION**
Schaumburg, IL 60173-5337, USA
www.heidenhain.com

VN **AMS Co. Ltd**
HCM City, Vietnam
E-mail: davidgoh@amsvn.com

ZA **MAFEMA SALES SERVICES C.C.**
Midrand 1685, South Africa
www.heidenhain.co.za

827222-27 · 10 · 07/2019 · CD · Printed in Germany