HEIDENHAIN

Options and Accessories

For TNC Controls

September 2016

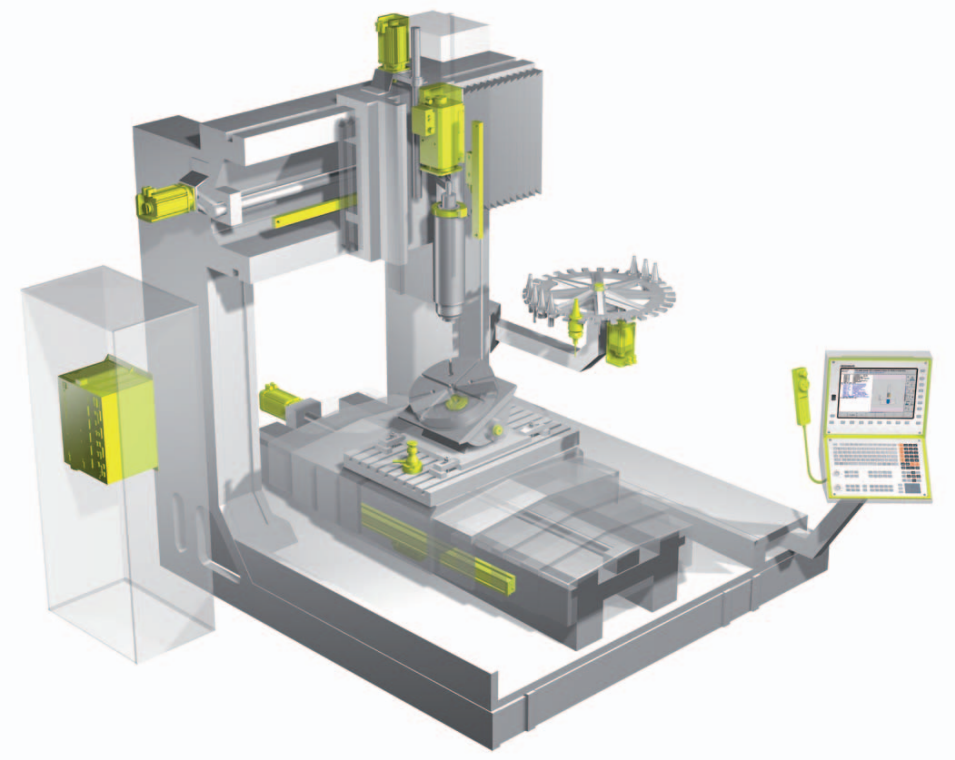
Options and accessories for TNC controls

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 integrated in the control with which you can adapt the feature range of the TNC, retroactively if necessary, to your actual requirements. 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**, e.g. for supporting data transfer or for creating a PLC program, or for a 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 machine tool builder	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	•	•	•	•	X	12
Coordinate transformation Tilting the working plane, PLANE function	**8**	617920-01 *iTNC 530:* 367591-01	•	•	•	•	X	13
Display step to 0.01 µm or 0,00001°	**23**	632986-01	–	•	•	–		–
Touch probe cycles • Compensation of workpiece misalignment, datum setting • Automatic tool and workpiece measurement • Touch probe input enabled for non-HEIDENHAIN systems	**17**	634063-01	S	•	S	S	X	14
Extended programming functions • FK free contour programming • Fixed cycles • Peck drilling, reaming, boring, counterboring, centering • Milling internal and external threads • Clearing level and oblique surfaces • Multioperation machining of straight and circular slots • Multioperation 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: Engrave text or numbers in a straight line or on an arc • Contour slot with trochoidal milling	**19**	628252-01	S	•	S	S		15
Program-verification graphics, program-run graphics • Plan view • Projection in three planes • 3-D view	**20**	628253-01	S	•	S	S		16
Finely detailed 3-D view	**20**	628253-01	S	•	S	–		17
Pallet editor	**22**	628255-01	–	•	S	S	X	18
DXF converter – importing contours and machining options from DXF files	**42**	526450-01	•	•	•	•		19
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	–	–	•	–	X	20
Eccentric turning	**50**	634608-01	–	–	•	–	X	21

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

Option	Option number	ID	TNC 320	TNC 620	TNC 640	iTNC 530	Adaptation by machine tool builder	Page
Programming and operation								
Synchronization of two or more spindles • Hobbing cycle	**131** **50**	806270-01	–	–	•	–	X	24
Extended tool management	**93**	676938-01	•	•	•	•	X	22
Interpolating spindle – interpolation turning	**96**	751653-01	–	–	•	•	X	23
CAD viewer – opening 3-D CAD data directly on the TNC	**98**	800553-01	S	S	S	•		25
Machine accuracy								
KinematicsOpt – touch probe cycles for automatic measurement of rotary axes	**48**	630916-01	–	•	•	•		26
KinematicsComp – 3-D spatial compensation	**52**	661879-01	–	–	•	•	X	27
CTC (Cross Talk Compensation) – compensation of position error through axis coupling	**141**	800542-01	–	•	•	•	X	28
PAC (Position Adaptive Control) – position-dependent adaptation of control parameters	**142**	800544-01	–	•	•	•	X	29
LAC (Load Adaptive Control) – load-dependent adaptation of control parameters	**143**	800545-01	–	•	•	•	X	30
LAC (Load Adaptive Control) – load-dependent adaptation of control parameters	**144**	800546-01	–	•	•	•	X	31
AVD (Active Vibration Damping) – active vibration amplitude reduction	**146**	800548-01	–	•	•	•	X	32

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

Overview

Option	Option number	ID	TNC 320	TNC 620	TNC 640	iTNC 530	Adaptation by machine tool builder	Page
Machining functions								
Interpolation – circular in 3 axes with tilted working plane	**8**	617920-01 *iTNC 530:* 367591-01	•	•	•	•		13
Interpolation – linear in 5 axes	**9**	617921-01 *iTNC 530:* 367590-01	–	•	•	•		–
Spline interpolation – processing third-degree polynomials	**9**	367590-01	–	–	–	•		–
5-axis simultaneous machining • 3-D tool compensation through surface normal vectors • Using 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 normal to the tool direction • Manual traverse in the active tool-axis system	**9**	617921-01 *iTNC 530:* 367590-01	–	•	•	•	X	33
Handwheel superimpositioning – superimposing handwheel positioning during program run	**21**	628254-01	–	•	S	S	X	34
Tool compensation – radius-compensated contour precalculation (LOOK-AHEAD)	**21**	628254-01	S	•	S	S		35
Dynamic Collision Monitoring (DCM)	**40**	526452-01	–	–	•	•	X	36
Global program settings	**44**	576057-01	–	–	–	•	X	37
Adaptive Feed Control (AFC)	**45**	579648-01	–	–	•	•	X	38
3D-ToolComp – 3-D radius compensation depending on the tool's inclination angle (only with option 9)	**92**	679678-01	–	–	–	•		39
ACC (Active Chatter Control) – active suppression of tool chatter	**145**	800547-01	–	•	•	•	X	40
Visual Setup Control (VSC) – camera-based monitoring of the setup situation	**136**	1099457-01	–	–	•	–	X	41
Communication								
HEIDENHAIN-DNC – communication with external Windows applications over COM component	**18**	526451-01	•	•	•	•	X	42
Remote Desktop Manager – display and remote operation of external computer units (e.g. a Windows PC)	**133**	894423-01	–	•	•	•	X	43

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

Option		Option number	ID	TNC 320	TNC 620	TNC 640	iTNC 530	Adaptation by machine tool builder	Page
Interfacing to the machine									
Additional control loops	Additional axis 1	**0**	354540-01	•	•	•	•	X	44
	Additional axis 2	**1**	353904-01	•	•	•	•	X	
	Additional axis 3	**2**	353905-01	–	–	•	•	X	
	Additional axis 4	**3**	367867-01	–	–	•	•	X	
	Additional axis 5	**4**	367868-01	–	–	•	•	X	
	Additional axis 6	**5**	370291-01	–	–	•	•	X	
	Additional axis 7	**6**	370292-01	–	–	•	•	X	
	Additional axis 8	**7**	370293-01	–	–	•	•	X	
	4 additional control loops	**77**	634613-01	–	–	•	•	X	
	8 additional control loops	**78**	634614-01	–	–	•	•	X	
Synchronized axes – gantry axes, tandem tables		**24**	634621-01	•	•	S	S	X	45
Python OEM process – realizing special functions		**46**	579650-01	•	•	•	•	X	46
Double speed – short control-loop cycle times for direct drives		**49**	632223-01	–	•	•	•	X	47
OEM option		**101 – 130**	579651-01 – 579651-30	–	–	•	•	X	48
Real-Time Coupling (RTC) – real-time coupling function for synchronizing axes and spindles		**135**	1085731-01	–	–	•	–	X	49

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

Overview

PC software		TNC 320	TNC 620	TNC 640	iTNC 530	Adaptation by machine tool builder	Page
TNCremo		•	•	•	•		50
TNCremoPlus		•	•	•	•		50
TeleService		•	•	•	•	X	51
RemoTools SDK		•	•	•	•	X	42
virtualTNC		–	–	•	•	X	52
PLCdesign		•	•	•	•	X	53
KinematicsDesign		•	•	•	•	X	54
CycleDesign		•	•	•	•	X	55
TNCscope		•	•	•	•	X	56
DriveDiag		–	•	•	•	X	57
TNCopt		–	•	•	•	X	58
IOconfig		–	•	•	•	X	59
Software Key Generator		•	•	•	•	X	60
BMXdesign		•	•	•	•	X	61
Fixture Wizard		–	–	–	•	X	62
Programming station	**Single-user license for demo version**	•	•	•	•		63
	Single-user license with TNC keyboard	•	•	•	•		
	Single-user license with virtual keyboard	•	•	•	•		
	Network license with virtual keyboard For 1 workstation	•	•	•	•		
	Network license with virtual keyboard For 14 workstations	•	•	•	•		
	Network license with virtual keyboard For 20 workstations	•	•	•	•		

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

Hardware enhancements			ID	TNC 320	TNC 620	TNC 640	iTNC 530	Adaptation by machine tool builder	Page
Handwheel	**HR 130** TTL panel mounted	With detent W/o detent	540940-01 540940-03	•	•	•	•		64
	HR 150 11 μA_{PP} panel-mounted	With detent W/o detent	540940-06 540940-07	•	•	•	•		
	HR 510 portable handwheel	With detent W/o detent	1120313-xx 1119971-xx	•	•	•	•		
	HR 510 FS portable handwheel	With detent W/o detent	1119974-xx 1120311-xx	•	•	•	•		
	HR 520 portable handwheel with display	With detent W/o detent	670303-01 670302-01	•	•	•	•		
	HR 520 FS portable handwheel with display	With detent W/o detent	670305-01 670304-01	•	•	•	•		
	HR 550 FS portable wireless handwheel with display	With detent W/o detent	606622-03 598515-03	•	•	•	•		
	HRA 551 FS handwheel holder for HR 550 FS		731928-02	•	•	•	•		
Workpiece touch probe	**TS 260** with cable		738283-xx	•	•	•	•	X	65
	TS 460 with radio or infrared transmission		737624-xx	•	•	•	•	X	
	TS 444 with infrared transmission		588008-xx	•	•	•	•	X	
	TS 740 with infrared transmission		573757-xx	•	•	•	•	X	
Tool touch probe	**TT 160** with cable		729763-xx	•	•	•	•	X	66
	TT 460 with radio or infrared transmission		728346-xx	•	•	•	•	X	
	TL Nano with laser scanning **TL Micro 150** with laser scanning **TL Micro 200** with laser scanning **TL Micro 300** with laser scanning		557690-xx 557684-xx 557685-xx 557686-xx	•	•	•	•	X	
Additional operating station	**ITC 755** with touch screen and ASCII keyboard		1039527-01	–	–	•	•	X	67
	ITC 750 15“ screen; separate TNC keyboard required		1039544-01	–	•	•	•	X	
	ITC 760 19“ screen; separate TNC keyboard required		827086-01	–	–	•	•	X	
Industrial PC	**IPC 6641** for Windows interface on TNC		749963-01	–	•	•	•	X	68
Camera system	**VS 101** for workspace monitoring		1137063-01	–	–	•	–	X	69

• = Available as option
– = Not available
S = 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 machining position. And all this is done without any modification of the machine's mechanics.

It's no longer necessary to work slowly on accurate parts with high surface definition. Machine tools that operate with Dynamic Precision can machine **quickly and precisely** at the same time.

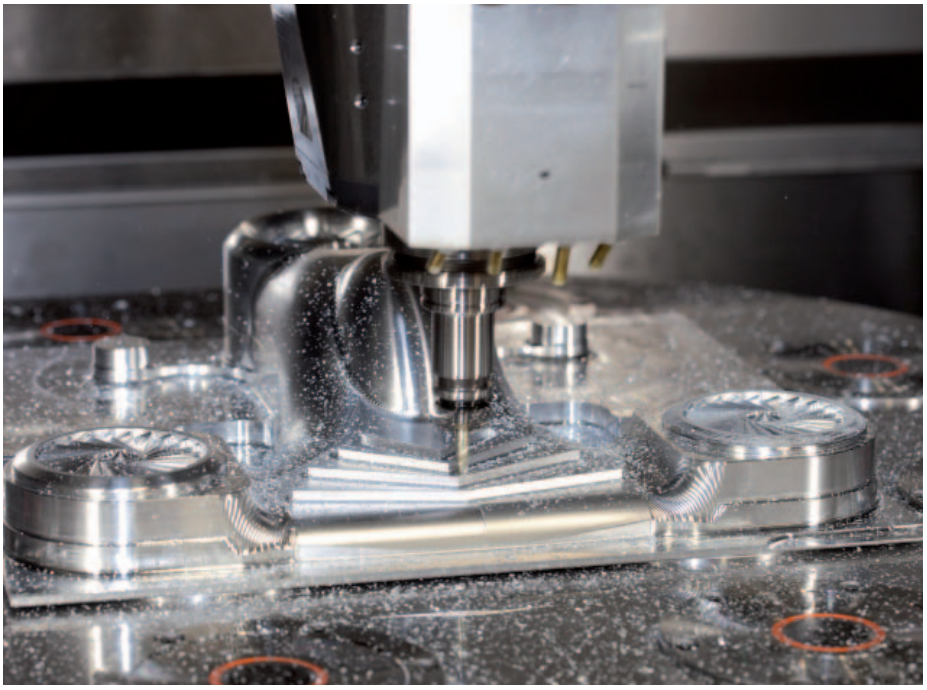
High precision together with fast machining is the basis for an increase in productivity. Unit costs are reduced without compromises in accuracy and surface quality. Dynamic Precision also ensures that accuracy is retained regardless of operating time and weight. It isn't necessary to reduce feed rates 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 at 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 has to enable the individual functions, enter their parameters and adapt them to the machine.



	TNC 640	iTNC 530	TNC 620	Page
Dynamic Precision	**x**	**x**	**x**	
CTC – compensation of position errors through mechanical compliance	Option	Option	Option	28
AVD – active vibration damping	Option	Option	Option	32
PAC – position-dependent adaptation of control parameters	Option	Option	Option	29
LAC – load-dependent adaptation of control parameters	Option	Option	Option	30
MAC – motion-dependent adaptation of control parameters	Option	Option	Option	31
Installation by the machine tool builder				

For more information, see the Technical Information on *Dynamic Precision, and 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.

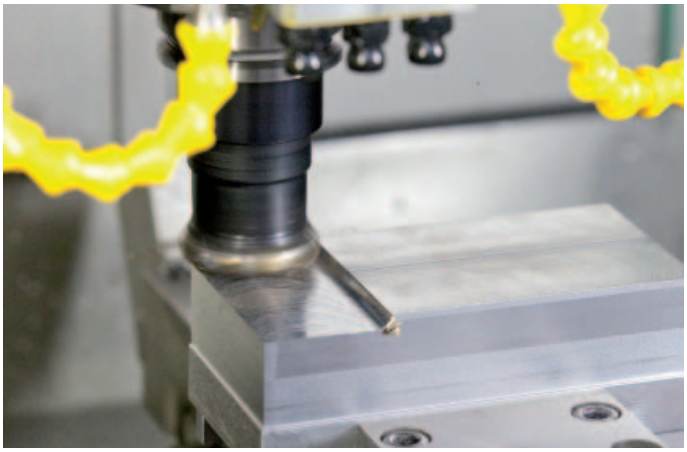
Dynamic Efficiency comprises three software functions:

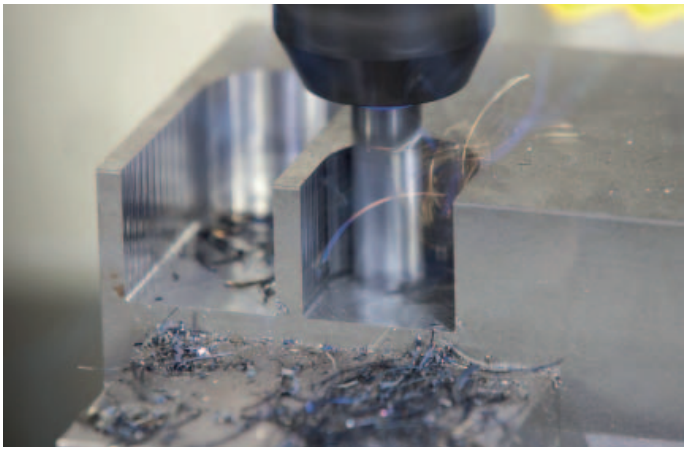
- ACC – reduces chatter tendencies and permits higher feed rates and greater infeeds
- AFC – controls the feed rate depending on the machining situation
- Trochoidal milling – a function for the roughing of slots and pockets that eases the load on the tool

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

Each solution in itself offers decisive advantages in the machining process. But the combination of these TNC features, in particular, exploits the potential of the machine and tool and at the same time reduces the mechanical load. Changing machining conditions, such as interrupted cuts, various material plunging procedures or simple clear-out also show that these features pay for themselves. 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 economy.





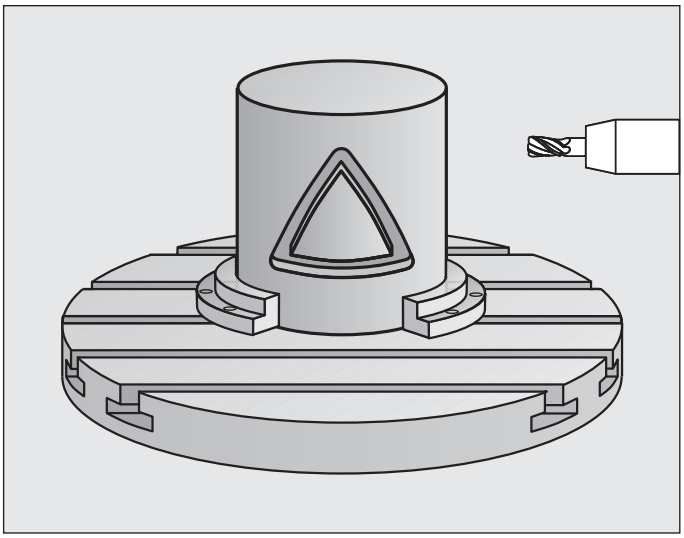
	TNC 640	iTNC 530	TNC 620	Page
Dynamic Efficiency	**x**	**x**	**–**	
ACC – Active Chatter Control	Option	Option	Option	40
AFC – Adaptive Feed Control	Option	Option	–	38
Trochoidal milling	Standard	Standard	Standard	–

Installation by the machine tool builder/user

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

Programming and operation
Machining with a rotary table

Many five-axis operations that at first glance may seem very complex can be reduced to conventional 2-D movements that are simply wrapped onto 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 quite easy to program contours (which consist of straight lines and arcs) on cylindrical surfaces using rotary and tilting tables: You simply program the contour in a plane (independent of the axis on the TNC 640, TNC 620 and TNC 320) as if the cylinder surface were unrolled. and 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
In the standard version, the feed rate of rotary axes is programmed in degrees/minutes. 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.

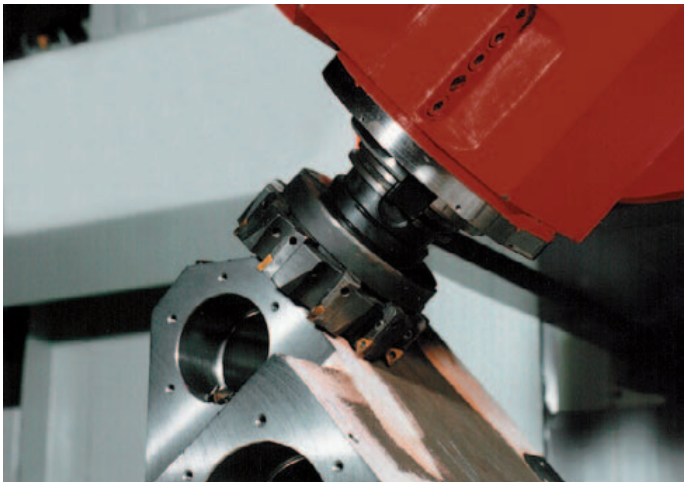
Machining with a rotary table TNC 640 HSCI/TNC 620 HSCI/TNC 320 iTNC 530 HSCI/iTNC 530	Option 8	 ID 617920-01 ID 367591-01
TNC 640 HSCI	As of NC SW 34059x-01	
TNC 620 HSCI	As of NC SW 34056x-01/73498x-01/81760x-01	
TNC 320	As of NC SW 34055x-01/771851-01	
iTNC 530 HSCI	As of NC SW 60642x-01	
iTNC 530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

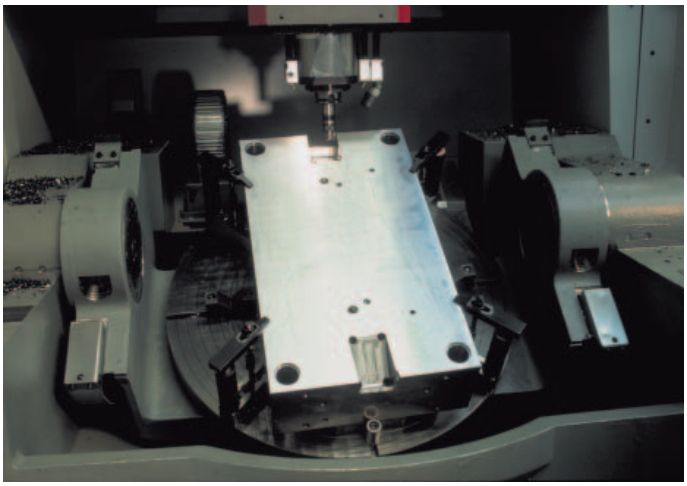
Programming and operation
Coordinate transformation – tilting the working plane, PLANE function

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 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 on the workpiece drawing. In order to keep the use of this complex function as simple as possible, a separate animation is available for each possible plane definition, so that you can view it 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 *www.klartext-portal.com*		

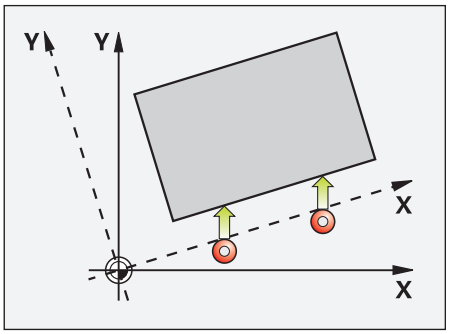
Programming and operation
Touch probe cycles

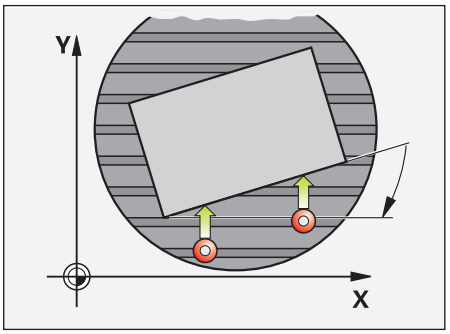
Workpiece alignment
With HEIDENHAIN touch probes and the probing functions of the TNC, you can forgo any tedious manual alignment of the workpiece:
- Clamp the workpiece in any position.
- The touch probe ascertains the workpiece misalignment by probing a surface, two holes, or two studs.
- The TNC compensates the misalignment with a "basic rotation," which means that in the NC program the part is rotated by the measured misalignment.

Datum setting
Finding this point quickly and reliably reduces nonproductive time and increases machining accuracy. The TNC features a large number of probing cycles for automatic presetting.

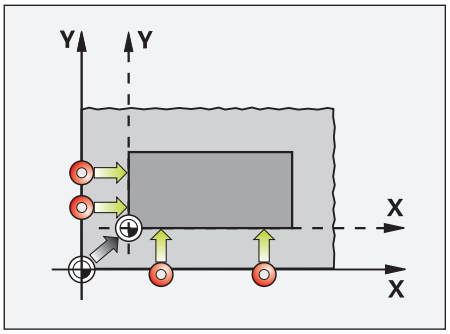
Workpiece inspection
The TNC features a number of measuring cycles for checking the geometry of the machined workpieces. This enables you to
- recognize a workpiece and call an appropriate part program,
- check whether all machining operations were conducted correctly,
- detect and compensate tool wear,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

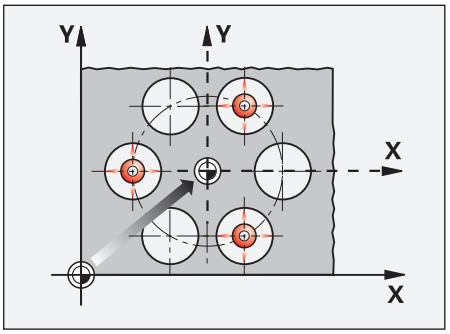
Tool measurement
Together with the TT or TL touch probes for tool measurement, the TNC makes it possible to measure tools automatically while they are in the machine spindle. The TNC saves the ascertained values of tool length and radius in the central tool file. By inspecting the tool during machining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



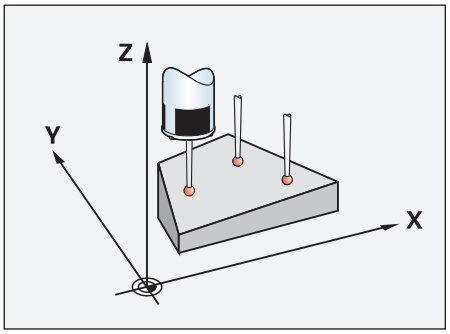


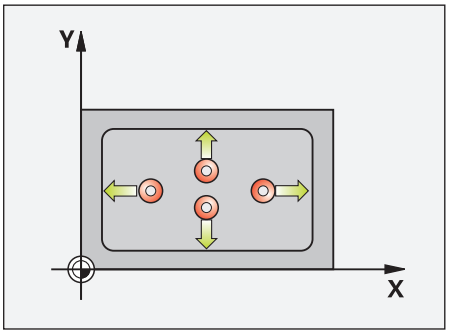
Compensating for workpiece misalignment by rotating the coordinate system or turning the table



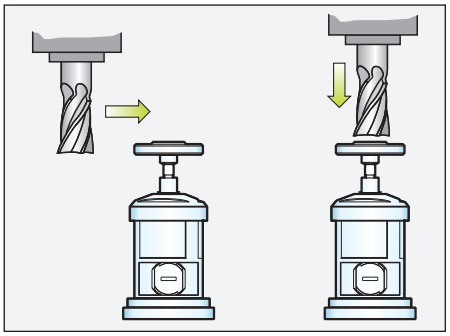
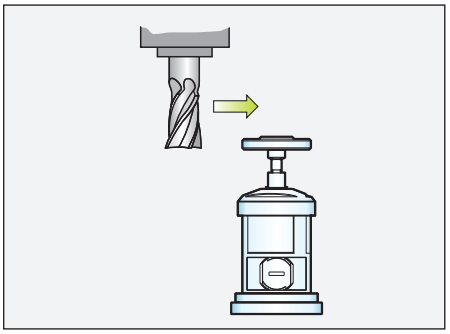


Setting a datum at a corner, for example, or in the center of a circular hole pattern





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

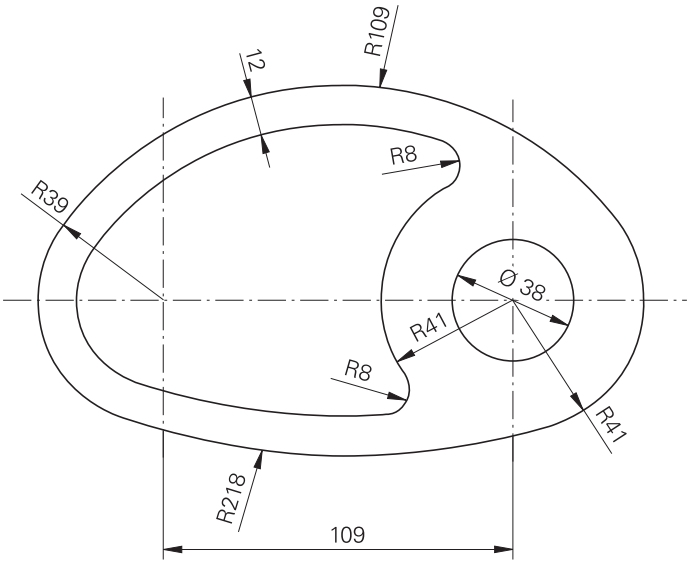
Advanced programming functions – FK free contour programming, fixed cycles

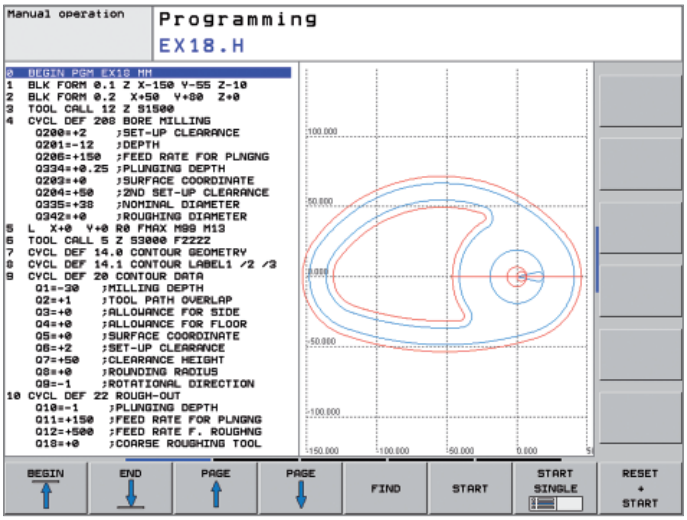
FK free contour programming
Not all workpieces are dimensioned for conventional NC programming. Thanks to FK, the control's free contour programming feature, in such cases you simply type in the data from the drawing—without first having to convert or calculate your data! It does not matter if individual 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, the helpful TNC programming graphics present the possible variants for your 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 cycles (SL) 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 subroutines. 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 *www.klartext-portal.com*		

Programming and operation
Program-verification graphics, program-run graphics

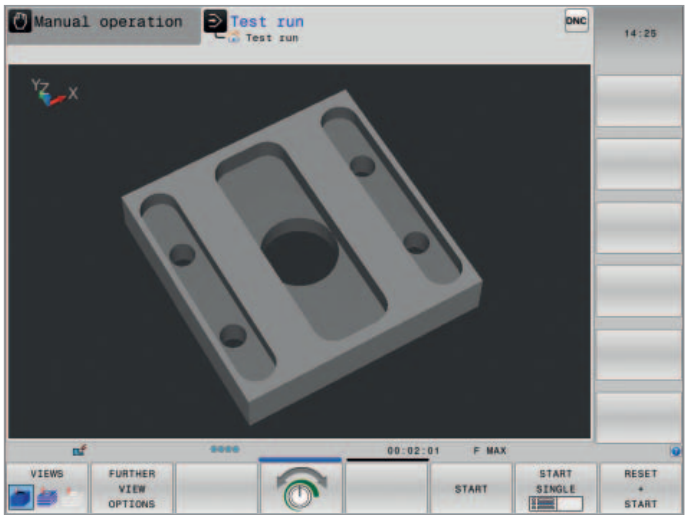
Programming graphics
HEIDENHAIN controls support you with detailed programming graphics. The programming graphics function is available as a standard feature on all controls and is described in the respective product brochures.

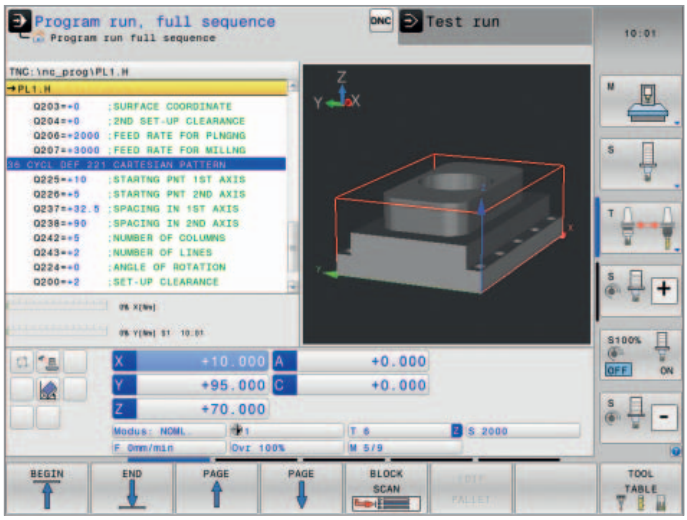
Various other graphic views are optional:

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 a plan view with different shades of depth
- In three planes (as in the workpiece drawing)
- In a solid model, 3-D view

Details can be displayed in 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 *www.klartext-portal.com*		

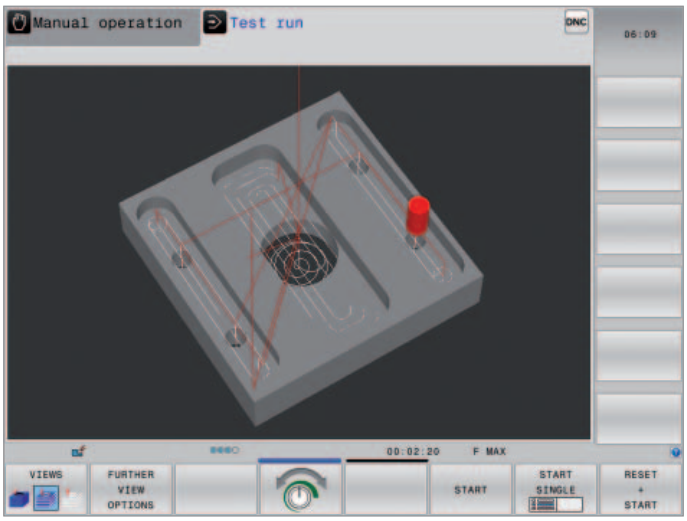
Programming and operation
Detailed 3-D view in the program-verification and program-run graphics

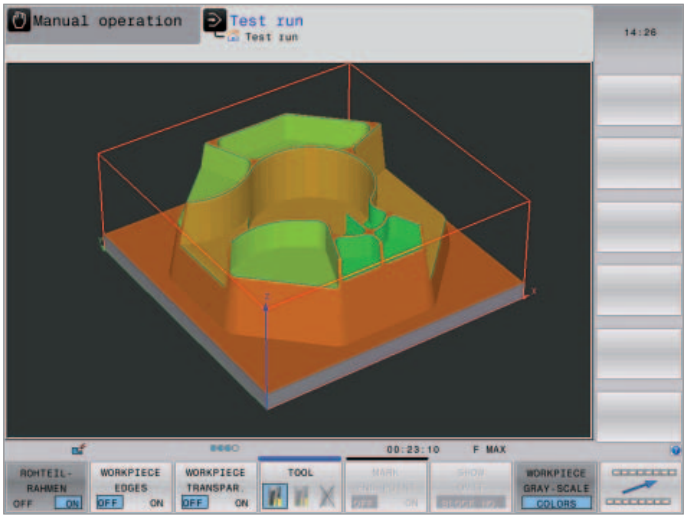
Finely detailed 3-D view
The program-verification and the program-run graphics of the TNC 640, TNC 620 and TNC 320 include further view options with detailed 3-D view. They help you to evaluate the workpiece quality in the simulation even before the actual machining operation begins, as well as during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

The high-resolution 3-D view with high zoom factors shows the most minor 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 graphics with the further view options 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.

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 an 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 *www.klartext-portal.com*		

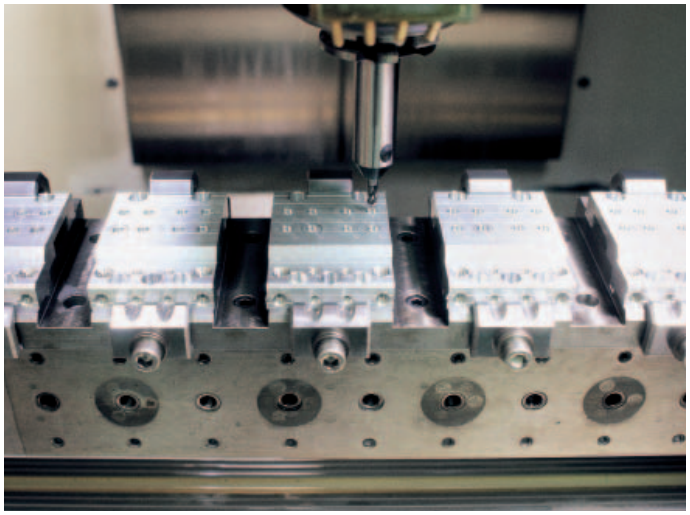
Programming and operation
Pallet editor

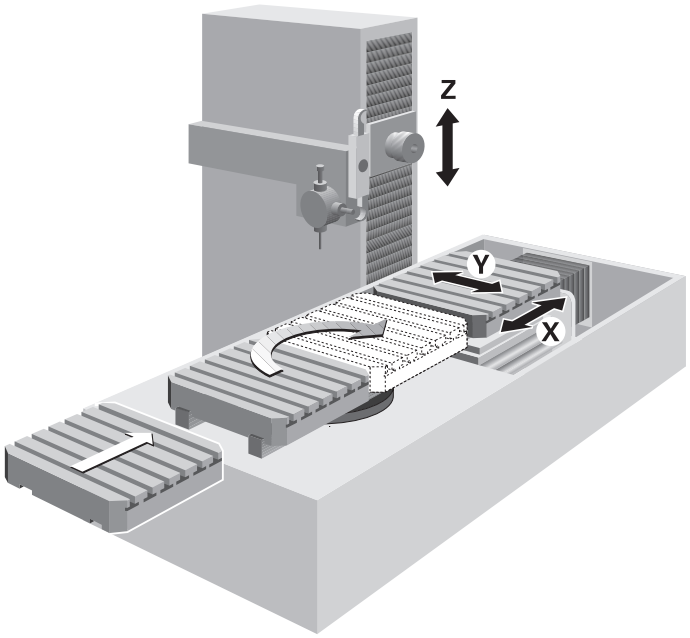
The TNC can assign the appropriate program and datum shift to parts mounted on pallets and brought to the machine in a random sequence.

If a pallet is exchanged 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 pallet and workpiece datums, 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 (only with iTNC 530).





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

DXF converter – importing contours and machining positions from DXF files

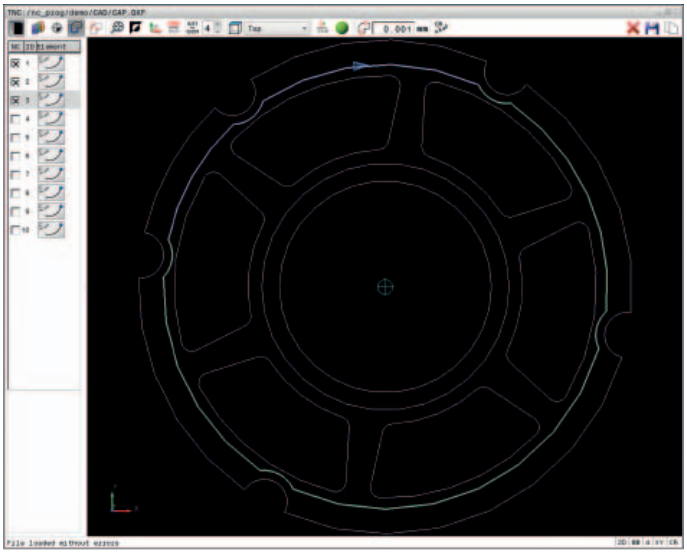
Why program contours when your drawing is already in DXF format anyway? You can open DXF files directly on the TNC in order to extract contours or machining positions from it. This not only saves time otherwise spent on programming and testing, you can also be sure that the finished contour is exactly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

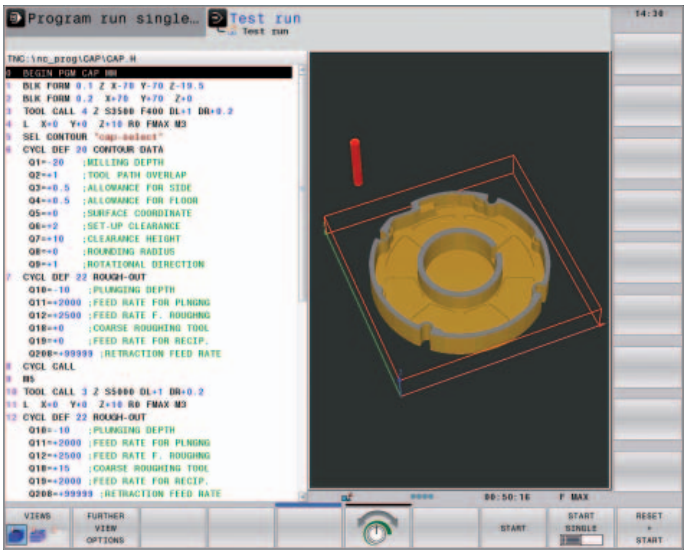
As a rule, DXF files contain multiple layers, with which the design engineer organizes a drawing. So that as little unnecessary information as possible appears on the screen during selection of the contours, with a keystroke you can hide all **excessive layers** contained in the DXF file. This requires a keyboard with touchpad or an external pointing device. The TNC can select a contour train even if it has been saved in **different layers.**

The TNC also supports you when **defining the workpiece datum.** The TNC has a function for this with which you can shift the drawing datum to a suitable location simply by clicking 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 the **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 extensive 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 use drilling positions or starting points for pocket machining. Of course, the TNC saves the machining positions so that they can be reached on the shortest path.





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 *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 the NC program as desired between turning and milling 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 the turning operations—as always—conveniently under dialog guidance in HEIDENHAIN Klartext conversational language. Besides the standard path functions you can also use FK free contour programming to easily create contour elements not otherwise dimensioned for NC. Beyond this, you also have the contour elements recessing and undercutting for turning operations, which are supported by expressive help illustrations.

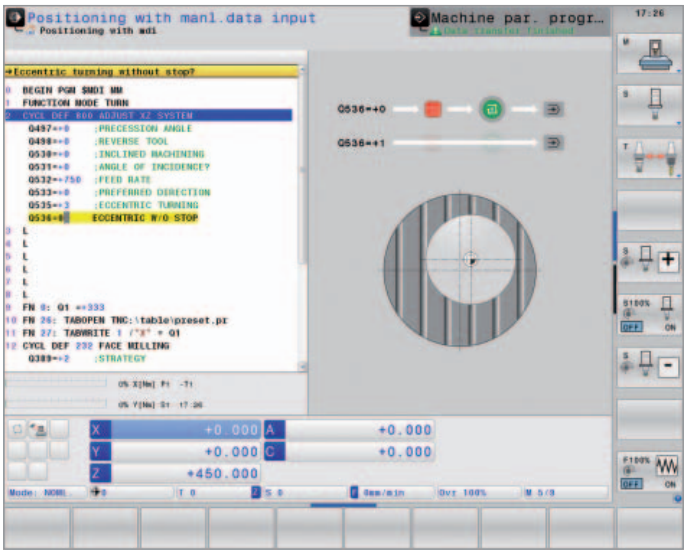
Cycles for milling and turning
HEIDENHAIN controls have always been known for their comprehensive and technologically sophisticated package of cycles. Frequently recurring operations that comprise several steps are also stored in the TNC 640 as cycles. You program them under conversational guidance and 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 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 brochure *TNC 640, and 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, due to the setup situation, is not aligned with the axis of rotation. During machining, the TNC 640 compensates any eccentricity with opposing movements of the linear axis coupled with the rotating spindle. This can significantly reduce the time required for setup.

Option 135 (RTC – Real-time coupling function) is required in addition to option 50 (turning functions) for eccentric turning.



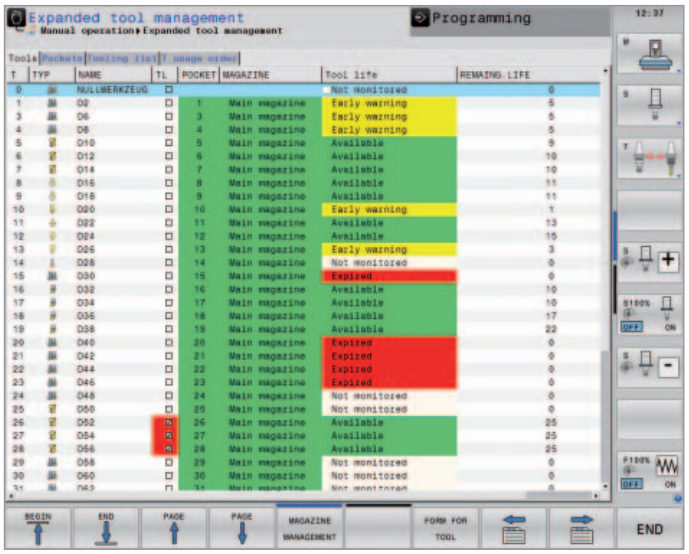
Eccentric turning **RTC** (required only with NC-SW 34059x-04)	Option 50 Option 135	ID 634608-01 ID 1085731-01
TNC 640 HSCI **TNC 620 HSCI** **TNC 320** **iTNC 530 HSCI** **iTNC 530**	As of NC SW 34059x-04 – – – –	
Installation by the machine tool builder		
More information –		

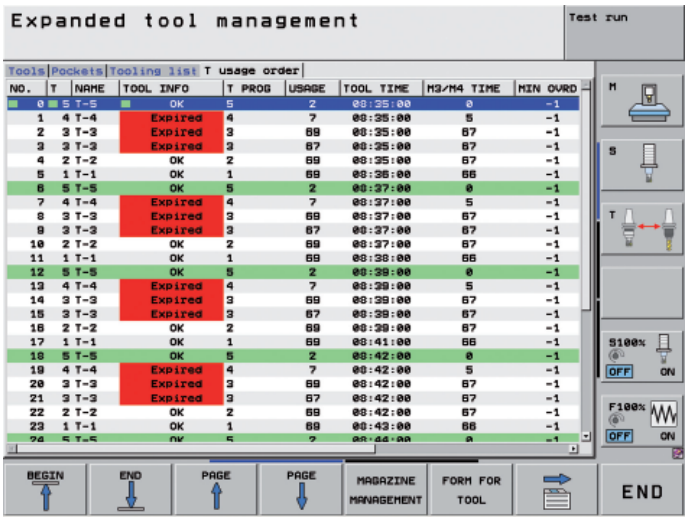
Programming and operation
Extended tool management

Numerous new features that make management of tools and magazines much more transparent are included in the extended tool management. 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.





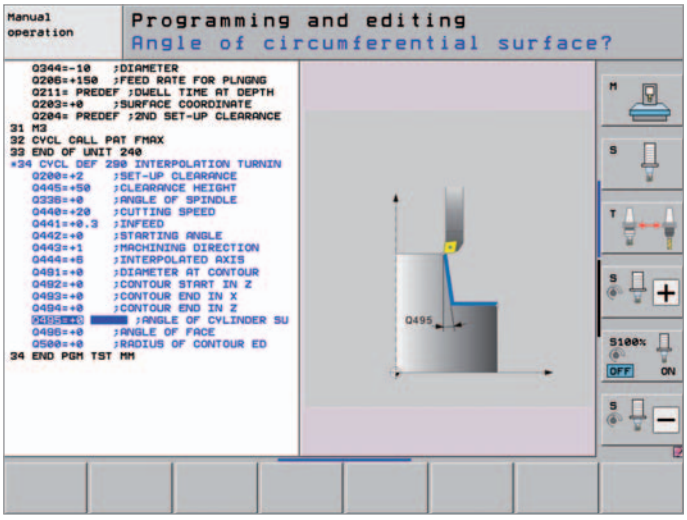
Extended tool management	Option 93	ID 676938-01
TNC 640 HSCI	As of NC SW 34059x-01	
TNC 620 HSCI	As of NC SW 81760x-03	
TNC 320	As of NC SW 771851-02	
iTNC 530 HSCI	As of NC SW 60642x-01	
iTNC 530	As of NC SW 34049x-05	
Installation by the machine tool builder		
More information –		

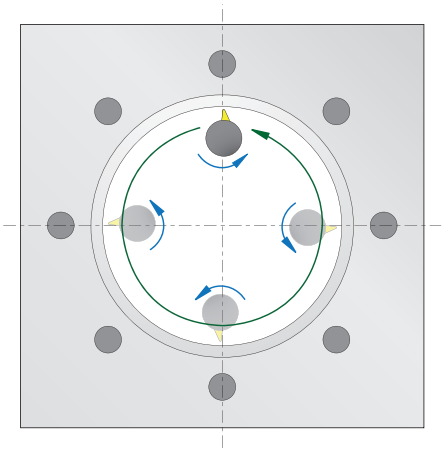
Programming and operation
Interpolating spindle – interpolation turning

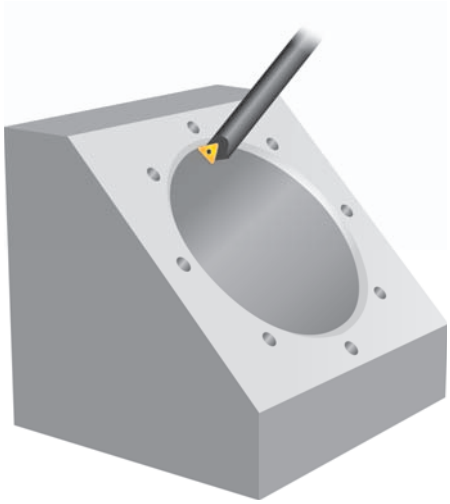
During interpolation turning, the tool's cutting edge moves along a circle, with the cutting edge always oriented to the center of the circle (outside machining) or away from the center (inside machining). By varying the circle radius and the axial position, any rotationally symmetric objects can be 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 tool location 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).





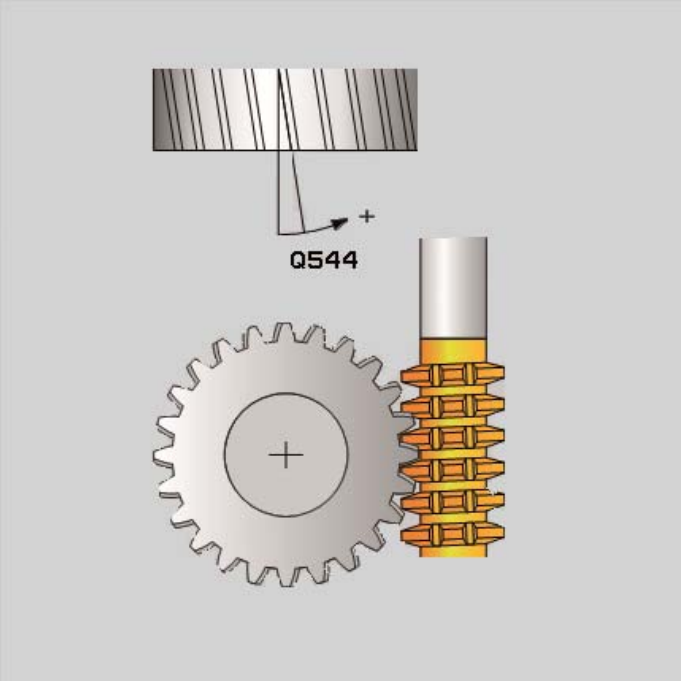


Interpolating spindle	Option 96	ID 751653-01
TNC 640 HSCI	As of NC SW 34059x-05	
TNC 620 HSCI	–	
TNC 320	–	
iTNC 530 HSCI	As of NC SW 60642x-02	
iTNC 530	As of NC SW 34049x-07	
Installation by the machine tool builder		
For more information, see the *iTNC 530, TNC 640* brochures		

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.

The TNC 640 offers you Cycle 880, gear hobbing, in conjunction with option 50 (turning functions), 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. The new Cycle 880 automatically controls these complex movements and provides you with a simple and application-oriented way to enter all relevant values. You can use the tooth parameters directly from your drawing—the cycle calculates from them the course of five-axis movement.



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**		

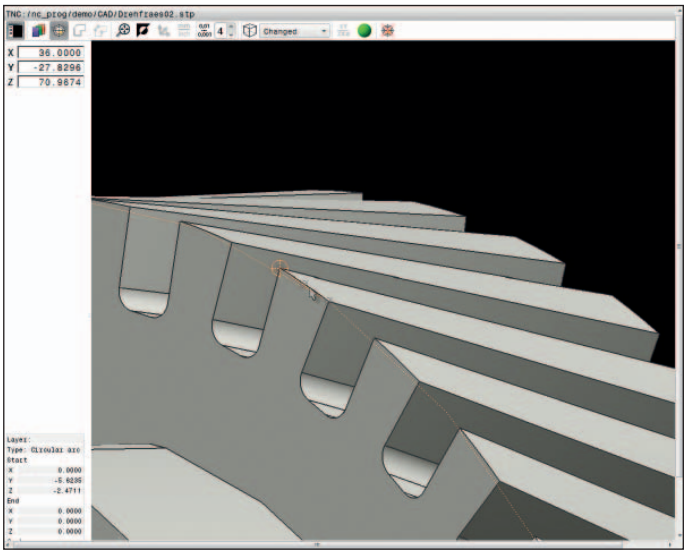
Programming and operation
CAD viewer – display of standardized CAD formats

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. Of course the 3-D CAD viewer includes functions for shifting, rotating and zooming the model so that any problematic locations can be displayed appropriately.

Moreover, you can also use the viewer to find position values and dimensions fro the 3-D model. And you can set the datum 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 –		

Machine accuracy

KinematicsOpt – easy calibration of rotary axes

Accuracy requirements are becoming increasingly stringent, particularly in the area of 5-axis machining. Complex parts need to be manufactured with precision and reproducible accuracy even over long periods.

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. The results of measurement are 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 for each rotary axis the range that you want to measure.

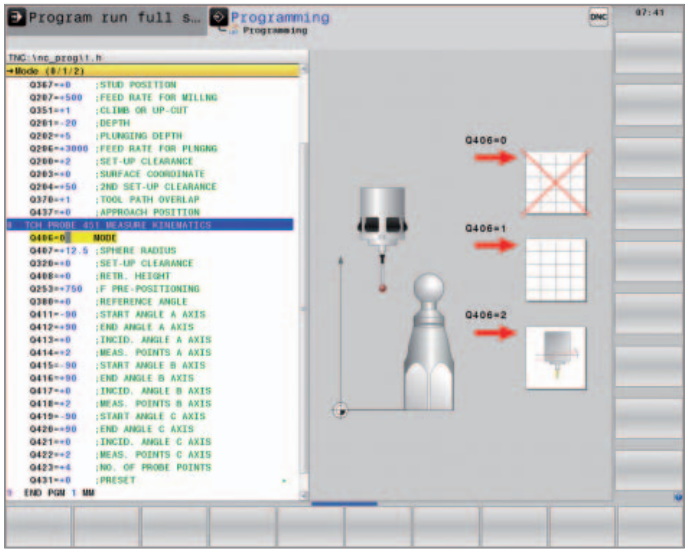
The TNC uses the values measured to determine the spatial errors resulting from the tilting of the axes. The cycle then calculates an optimized kinematic machine description, in which these errors are minimized, and saves it as machine kinematics.

Of course, a comprehensive log file is also saved with the actual measured values and the measured and optimized dispersion (measure for 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. That is why 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		

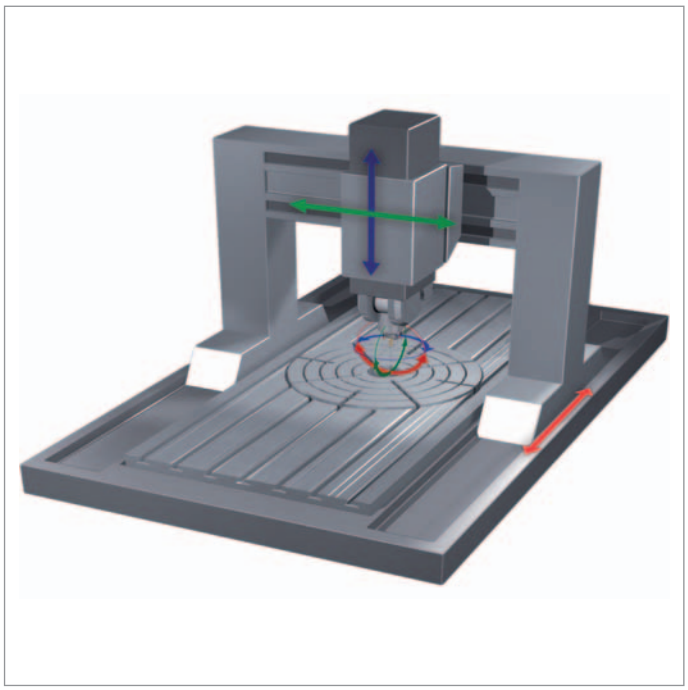
Machine accuracy
KinematicsComp – 3-D spatial compensation

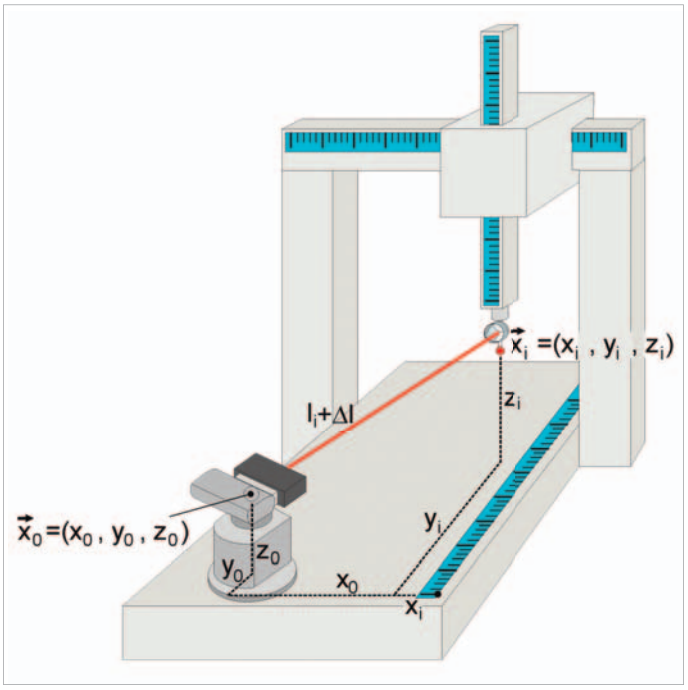
Narrow workpiece tolerances require high machine accuracy. However, machine tools inevitably have errors resulting from their assembly or production.

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 must also be considered, which can cause very complex geometric changes to machine components.

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 so that the Tool Center Point (TCP) can follow the ideal nominal contour exactly. Thermally induced errors can also be measured and compensated 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		

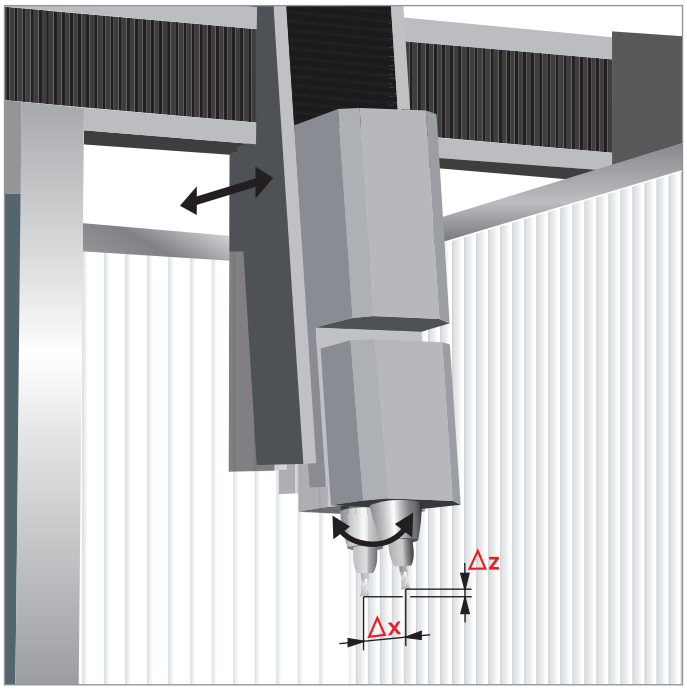
Machine accuracy

CTC – compensation of position errors through axis coupling

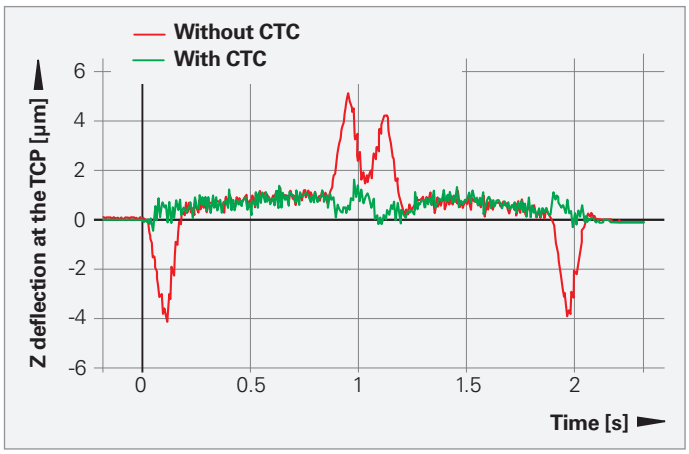
Dynamic acceleration processes introduce forces to the structure of a machine tool. They can deform parts of the machine for a short time and thereby lead to deviations at the tool center point (TCP). Besides deformation in axis direction, the dynamic acceleration of an axis due to mechanical axis coupling can also result in deformation of axes that are perpendicular to the direction of acceleration. This is especially so if the point of force application on an axis does not coincide with its center of gravity, which can cause 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 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 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

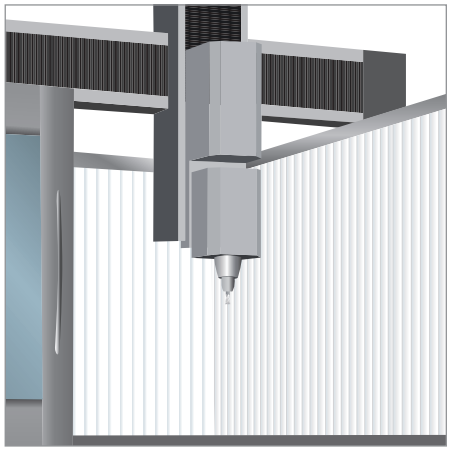
CTC	Option 141	ID 800542-01
TNC 640 HSCI	As of NC SW 34059x-02	
TNC 620 HSCI	As of NC SW 34056x-04/73498x-02/81760x-01	
TNC 320	–	
iTNC 530 HSCI	As of NC SW 60642x-02	
iTNC 530	–	
Installation by the machine tool builder		
For more information, see the *Dynamic Precision* Technical Information		

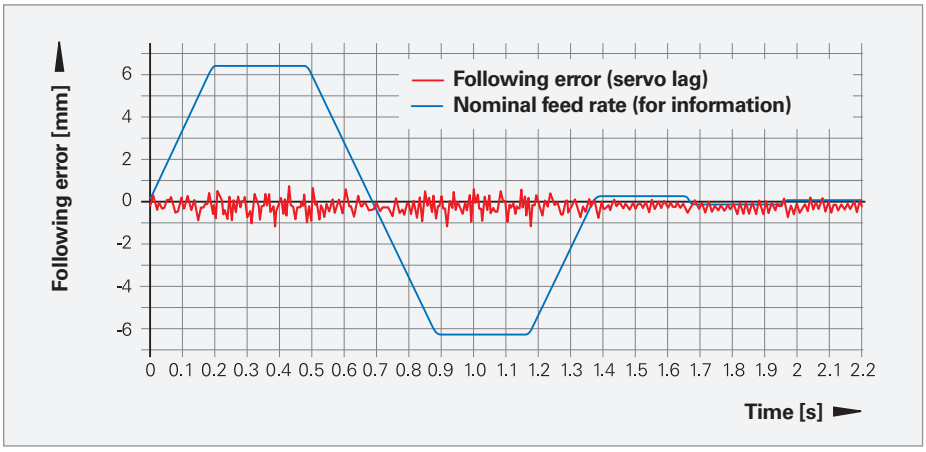
Machine accuracy
PAC – position-dependent adaptation of control parameters

Depending on the positions of the axes in a working space, the kinematic conditions of a machine give it a variable dynamic behavior that can adversely affect the stability of the servo control.

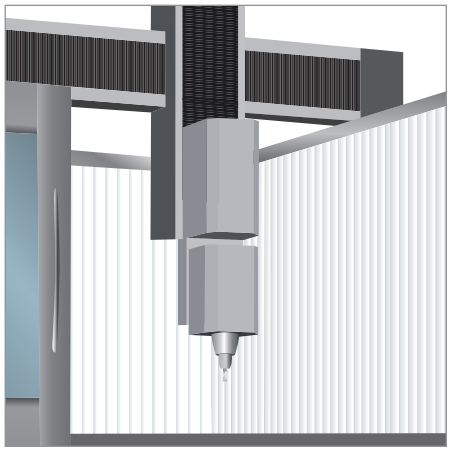
To exploit the machine's dynamic possibilities, you can use the **PAC** option (Position Adaptive Control) to change machine parameters depending on position.

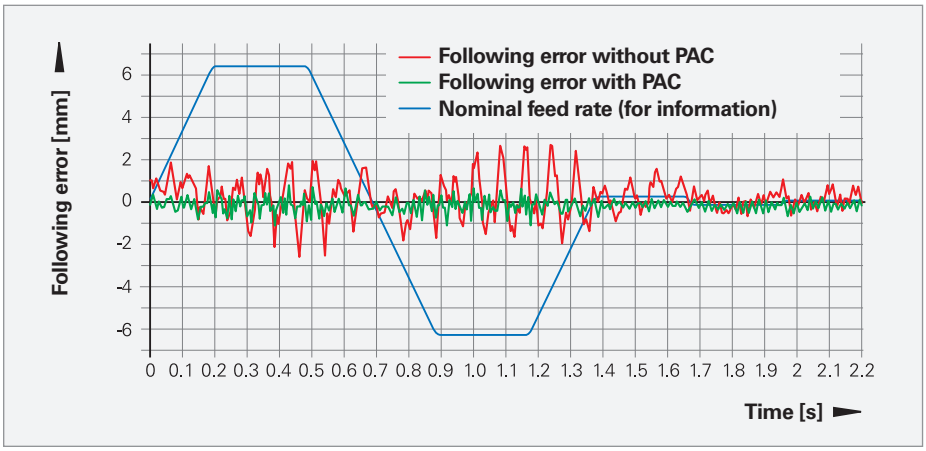
This makes it possible to assign respectively optimal loop gain to defined support points. Additional position-dependent filter parameters can be defined in order to further increase control loop stability.





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

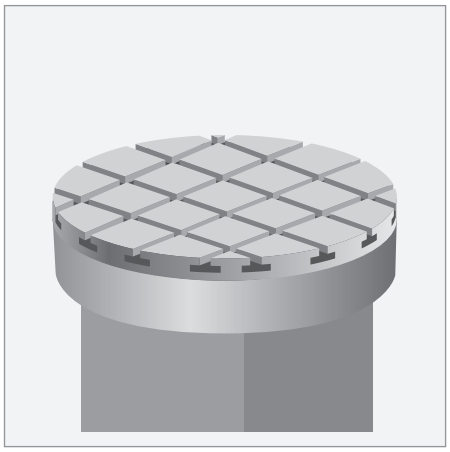
PAC	Option 142	ID 800544-01
TNC 640 HSCI	As of NC SW 34059x-02	
TNC 620 HSCI	As of NC SW 34056x-04/73498x-02/81760x-01	
TNC 320	–	
iTNC 530 HSCI	As of NC SW 60642x-02	
iTNC 530	–	
Installation by the machine tool builder		
For more information, see the *Dynamic Precision* Technical Information		

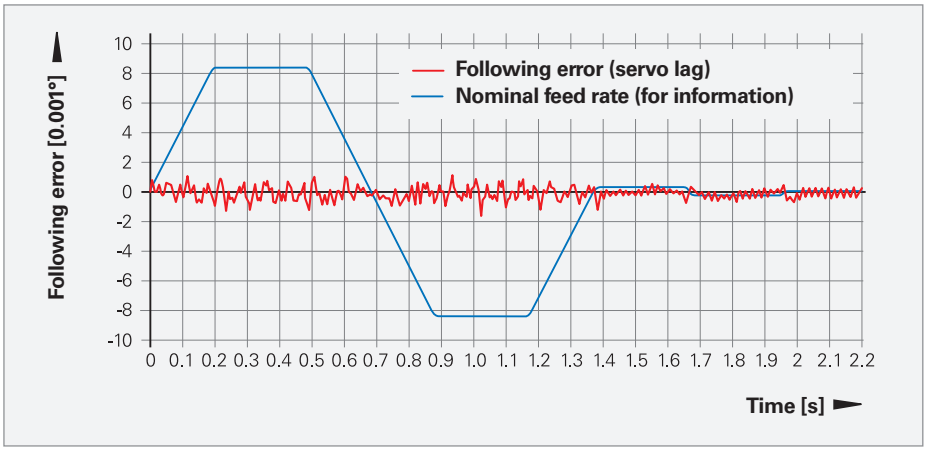
Machine accuracy
LAC – load-dependent adaptation of control parameters

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

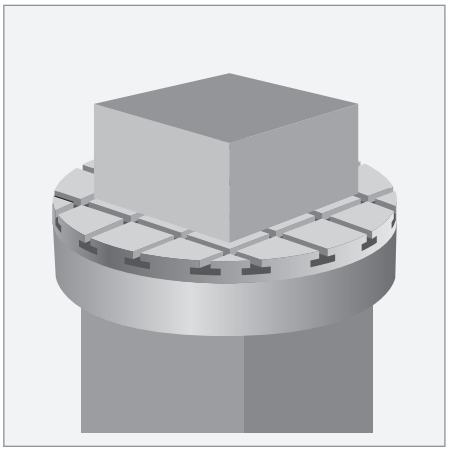
The **LAC** option (Load Adaptive Control), enables the control to automatically ascertain the current workpiece mass or mass moment of inertia as well as current frictional forces. In order to ensure an optimum response to changes in the behavior of the machine at differing loads, adaptive feedforward 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.

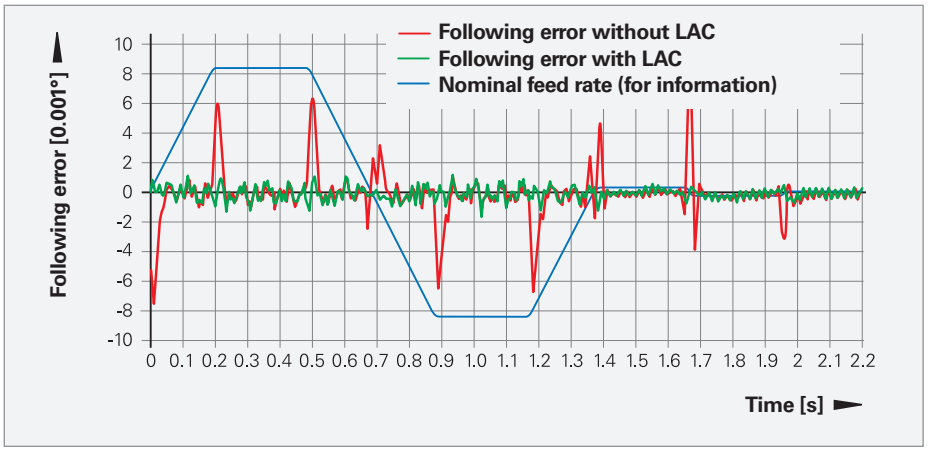
The TNC 620 and the TNC 640 provide Cycle 239 for ascertaining the current load conditions so that fast adaptation to sudden changes in load (e.g. loading and unloading of the workpiece) is ensured.





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





Additional load changed
- 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		

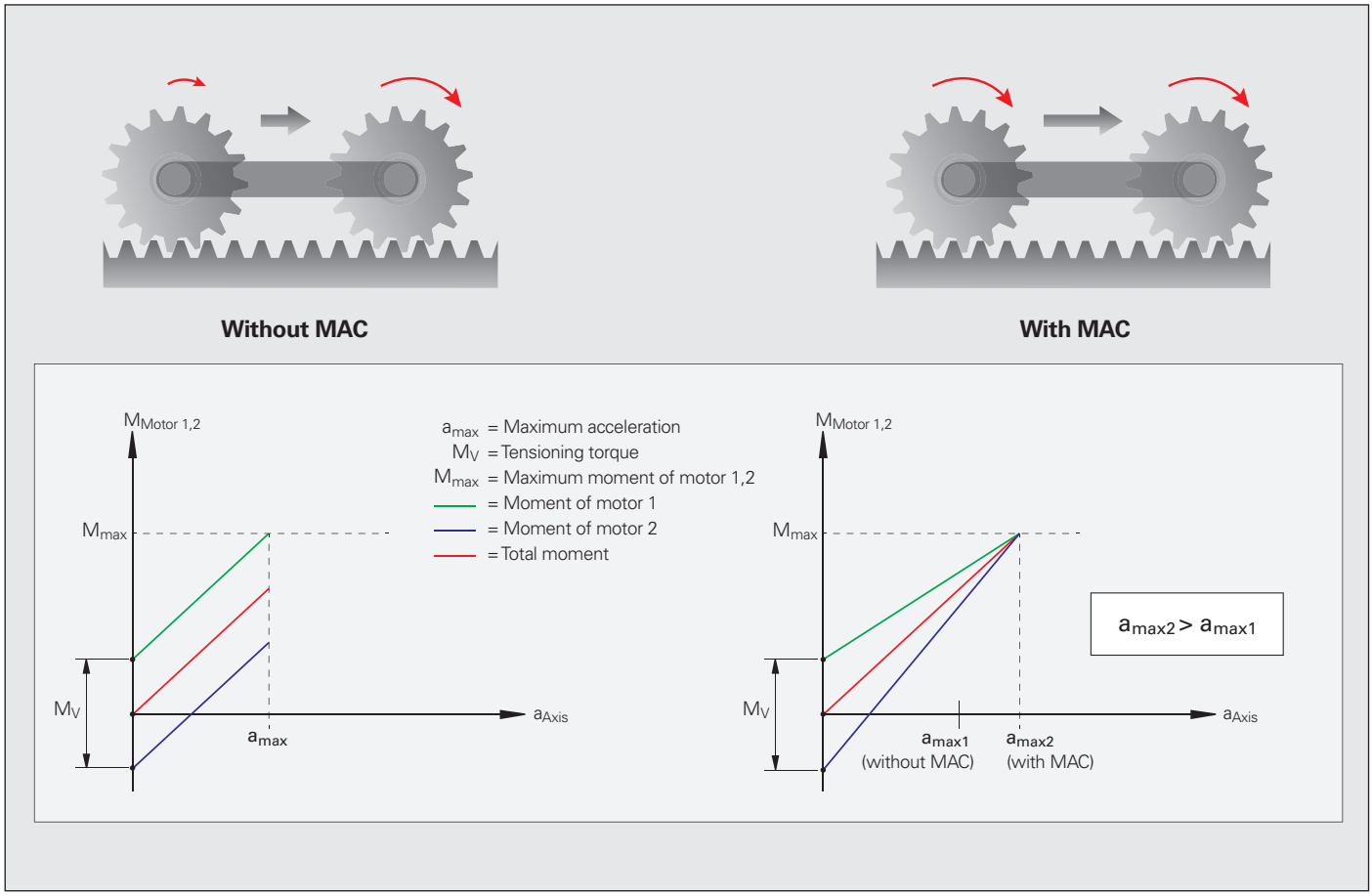
Machine accuracy

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 way to change machine parameter values depending on other input quantities such as velocity, following error or acceleration of a motor. Through this motion-dependent adaptation of the control parameters it is possible, for example, to realize a velocity-dependent adaptation of the kV factor on motors whose stability changes through the various 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 parameterized reduction of the tensioning torque with increasing acceleration.



dynamic + precision

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		

Machine accuracy
AVD – active vibration damping

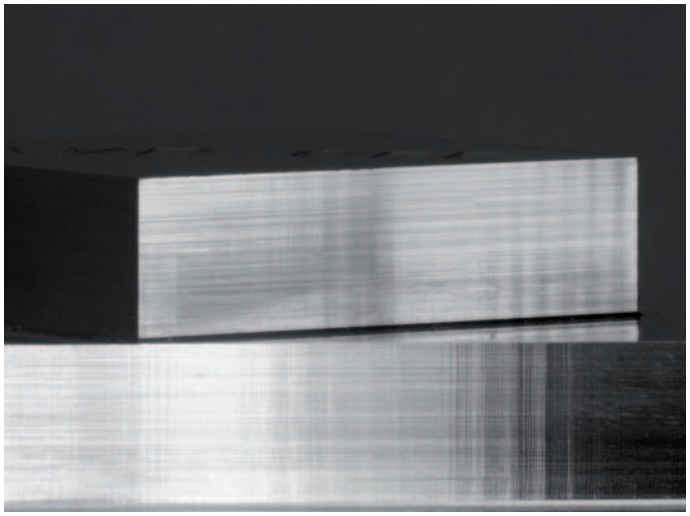
If there are low-frequency vibrations on a machine tool, inclined or curved surfaces can often have finish problems in the form of visible shadows or fluctuations in contrast. Peaks as small as 1 µm or even smaller can be visible on the workpiece surface. These defects often necessitate reworking of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

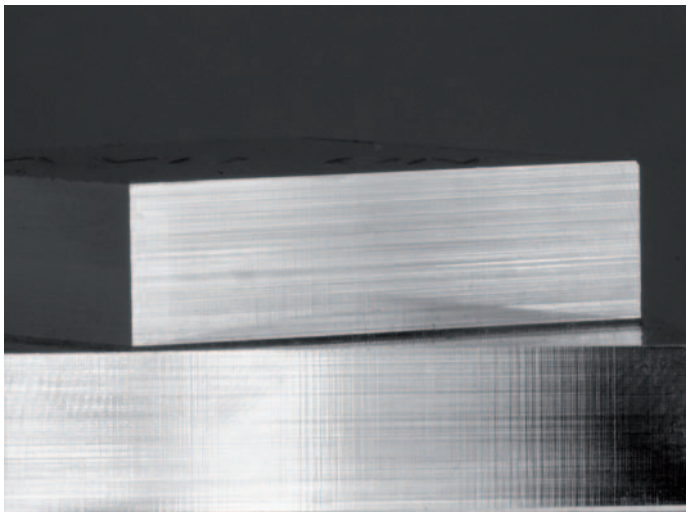
Common causes of low-frequency disturbances are elasticity in the drive train, such as results from vibration between the drive side (motor) and the friction side (slide), as well as vibrations from the machine setup, where high accelerations of the machine tool axes result in disturbances via the fastening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

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

The **AVD** feature (Active Vibration Damping) uses the control loop of the TNC to specifically suppress a dominant low-frequency vibration. AVD 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 makes fast and vibration-free milling operations possible.

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





dynamic + precision

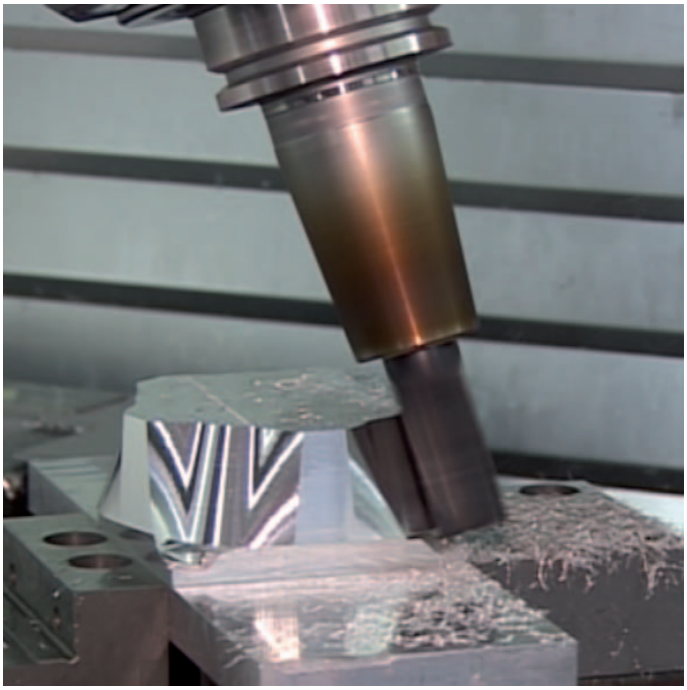
AVD	Option 146	ID 800548-01
TNC 640 HSCI	As of NC SW 34059x-02	
TNC 620 HSCI	As of NC SW 34056x-04/73498x-02/81760x-01	
TNC 320	–	
iTNC 530 HSCI	As of NC SW 60642x-03	
iTNC 530	–	
Installation by the machine tool builder		
For more information, see the *Dynamic Precision* Technical Information		

Machining functions
5-axis simultaneous machining

The TNC provides numerous powerful functions specially 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 five-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 makes all the compensating movements in the linear axes that result from movements in the tilting axes. The TNC's Tool Center Point Management (TCPM) feature—an improvement upon the proven TNC function M128—provides optimal tool guidance and prevents contour gouging.

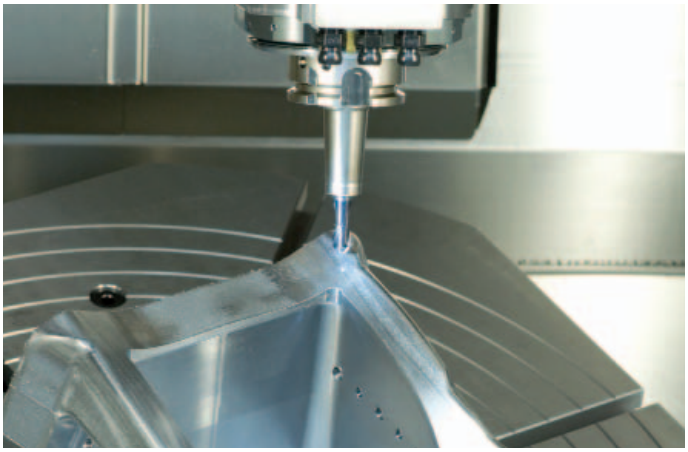


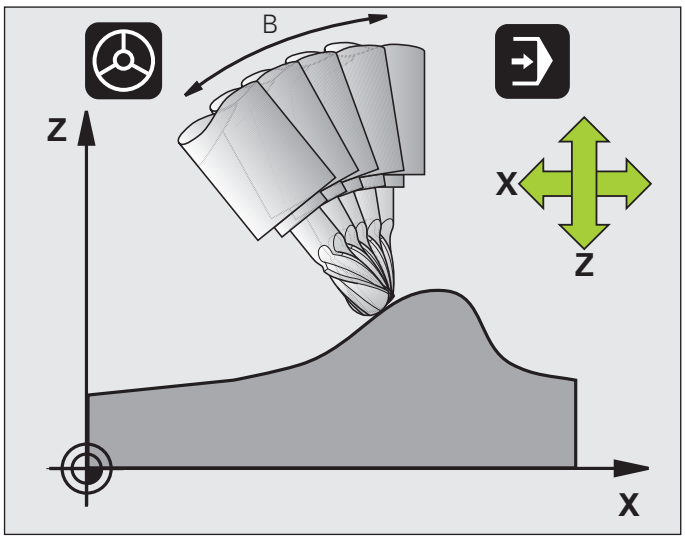
5-axis simultaneous machining	Option 9	
TNC 640 HSCI/TNC 620 HSCI		ID 617921-01
iTNC 530 HSCI/iTNC 530		ID 367590-01
TNC 640 HSCI	As of NC SW 34059x-01	
TNC 620 HSCI	As of NC SW 34056x-01/73498x-01/81760x-01	
TNC 320	–	
iTNC 530 HSCI	As of NC SW 60642x-01	
iTNC 530	As of NC SW 34049x-01	
Installation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

Machining functions

Handwheel superimpositioning – superimposing handwheel positioning during program run

The **handwheel superimposition** function (M118) enables you to make manual corrections by handwheel during program run. This is particularly helpful if you want to change inclination angles of rotary axes in externally written 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.



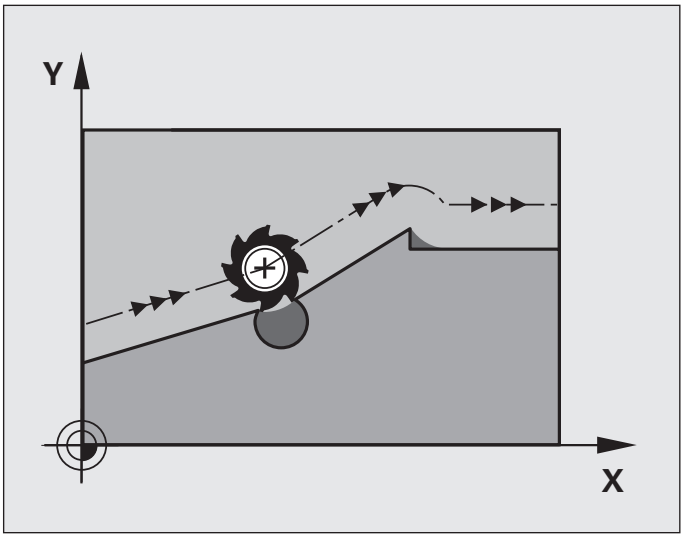


Handwheel superimposition	Option 21	ID 628254-01
TNC 640 HSCI	Standard	
TNC 620 HSCI	As of NC SW 34056x-01/73498x-01/81760x-01	
TNC 320	–	
iTNC 530 HSCI	Standard	
iTNC 530	Standard	
Installation by the machine tool builder		
For more information, see the TNC brochures		

Machining functions

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

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

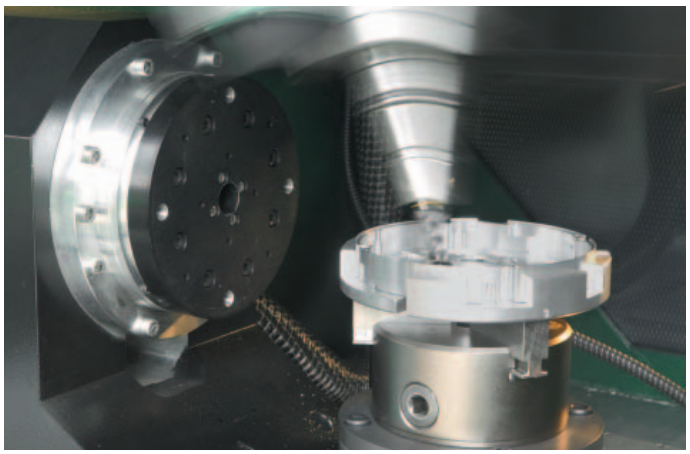
The complex motions and the normally high traversing speeds of 5-axis machining make axis movements difficult to foresee. This makes collision monitoring a valuable function that relieves the machine operator and protects the machine from damage.

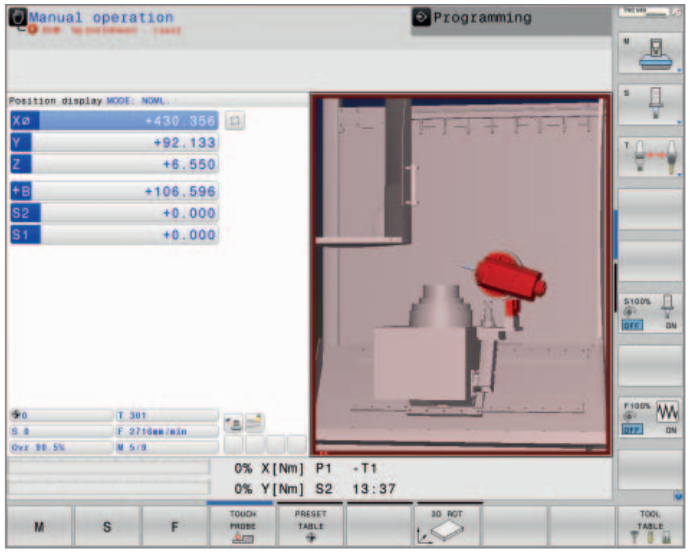
In cases such as these, the machine operator is supported by the **DCM** feature (Dynamic Collision Monitoring) of the TNC. The control interrupts machining whenever a collision is imminent, thereby increasing the safety for the machine and its operator. This helps to prevent machine damage, which can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

However, DCM works not only in **automatic mode.** It is also active in **manual mode.** If, for example, during setup the machine operator takes a collision course, 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 datum and real tools.

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

The TNC 640 additionally offers a convenient way to import collision objects from standard CAD models, such as 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.





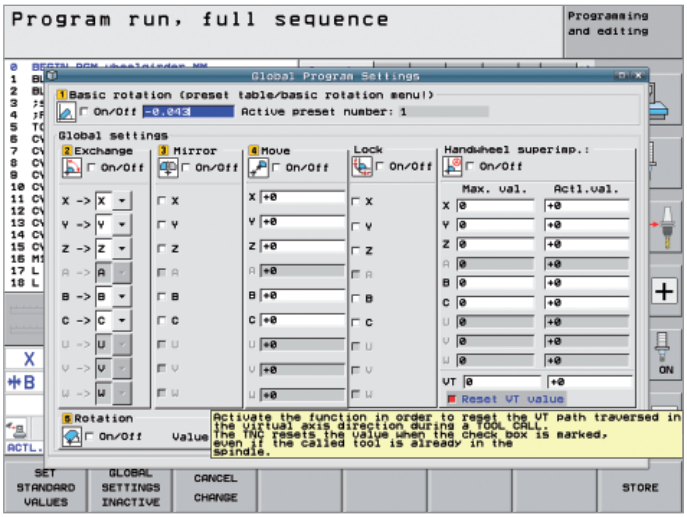
DCM	Option 40	ID 526452-01
TNC 640 HSCI	As of NC SW 34059x-02	
TNC 620 HSCI	–	
TNC 320	–	
iTNC 530 HSCI	As of NC SW 60642x-01	
iTNC 530	As of NC SW 34049x-02	
Installation by the machine tool builder		
For more information, see the *iTNC 530 or TNC 640* **brochure**		

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. They allow you to define various coordinate transformations and settings with global and priority effect for the selected NC program, without having to edit it.

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:
- Swapping axes
- Additional, additive datum shift
- Superimposed mirroring
- Axis locking
- 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
- Limit plane for graphically supported definition of machining limits



Global program settings	Option 44	ID 576057-01
TNC 640 HSCI	–	
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

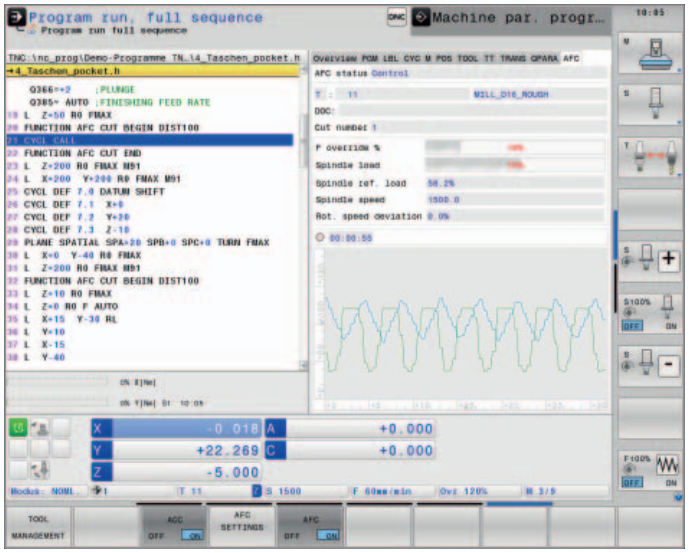
AFC (Adaptive Feed Control) automatically regulates the feed rate of the TNC, taking into consideration the respective spindle power and further process data. In a teach-in cut, the TNC records the maximum spindle power. Then, before actual machining, you define in a table the respective limit values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

The TNC's adaptive feed control offers various advantages:

Optimizing the machining time
Fluctuations in dimensions or material (blowholes) often appear particularly on cast parts. With a corresponding adaptation of the feed rate, the control tries to keep the previously "learned" maximum spindle power during the entire machining time. As a result of increasing the feed rate in zones with less stock removal, the total machining time is shortened.

Tool monitoring
The adaptive feed control permanently 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, a warning or executes a completely automatic tool change to 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. It effectively protects the spindle from 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		

Machining functions

3D-ToolComp – 3-D radius compensation depending on the tool's inclination angle

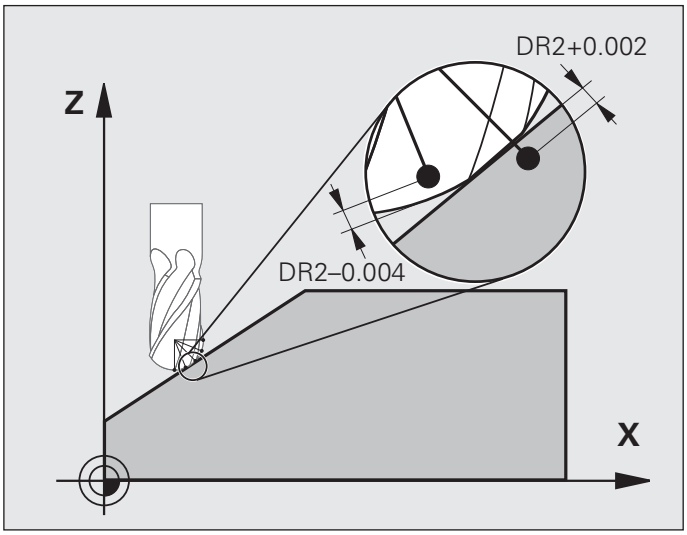
3D-ToolComp is a powerful option for three-dimensional tool radius compensation. A compensation-value table is used to define angle-dependent delta values that describe the tool deviation from the ideal circular form (see image).

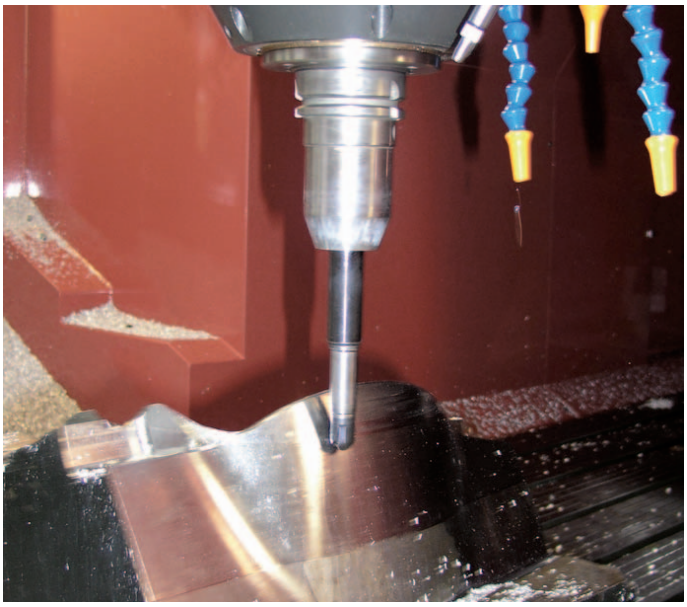
The TNC then corrects the radius value defined for the tool's current point of contact with the workpiece. In order to determine the point of contact exactly, the NC program must be have been created with surface-normal blocks (LN blocks) by a CAM system. The surface-normal blocks specify the theoretical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

Ideally, the compensation-value table is generated fully automatically by way of a special cycle that uses a laser system to measure the form of the tool so that the TNC can use this table directly. If the form errors of the tool used are available as a calibration chart from the tool manufacturer, then you can create the 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, 3-D probing, you enter the respective measured point with its coordinates and the associated normal vectors. After probing the TNC automatically calculates whether the measured points are 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 measuring log 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.



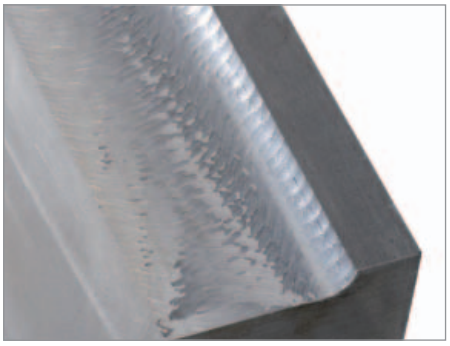


3D-ToolComp	Option 92	ID 679678-01
TNC 640 HSCI	As of NC SW 34059x-07	
TNC 620 HSCI	–	
TNC 320	–	
iTNC 530 HSCI	As of NC SW 60642x-01	
iTNC 530	As of NC SW 34049x-06	
Installation by the machine tool builder		
More information –		

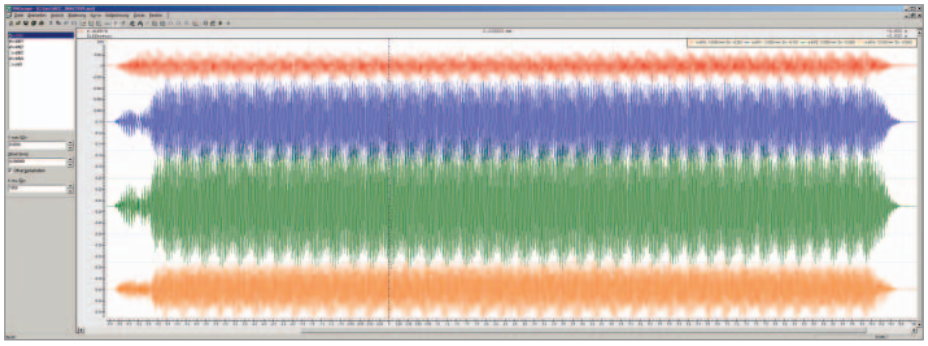
Machining functions
ACC – active chatter control

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." This chattering places heavy strain on the machine, and causes ugly marks on the workpiece surface. The tool, too, is subject to heavy and irregular wear from chattering. In extreme cases it can result in tool breakage.

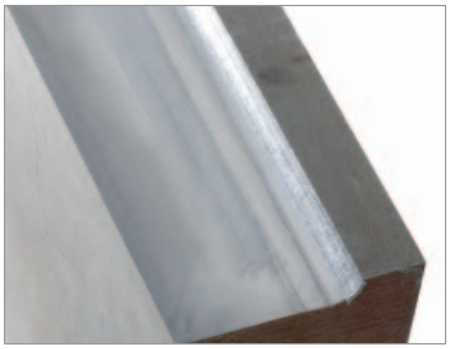
To reduce the inclination to chattering, HEIDENHAIN now offers an effective control function with its **ACC** Active Chatter Control option. The use of this control function is particularly advantageous during heavy cutting. ACC makes substantially higher metal removal rates possible. This enables you to increase your metal removal rate by up to 25 % and more, depending on the type of machine. You reduce the mechanical load on the machine and increase the life of your tools at the same time.



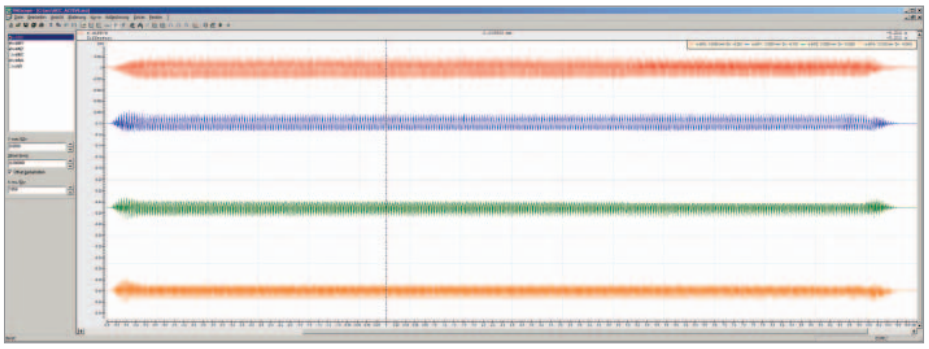
Heavy cutting without ACC



Following error without ACC



Heavy cutting with ACC



Reduced following error with ACC

dynamic + efficiency

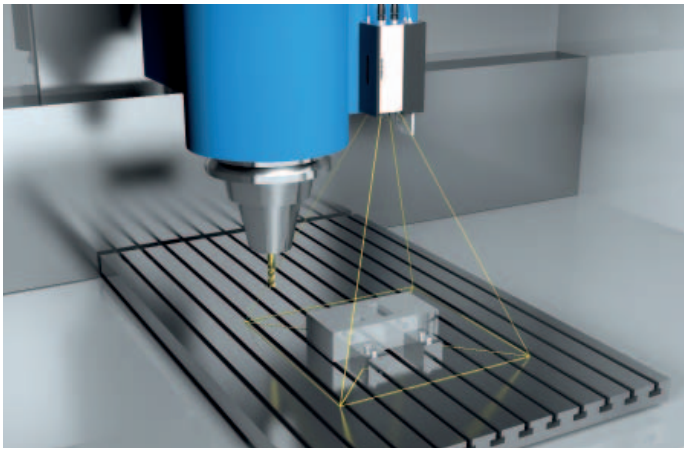
ACC	Option 145	ID 800547-01
TNC 640 HSCI	As of NC SW 34059x-02	
TNC 620 HSCI	As of NC SW 34056x-04/73498x-02/81760x-01	
TNC 320	–	
iTNC 530 HSCI	As of NC SW 60642x-03	
iTNC 530	–	
Installation by the machine tool builder		
For more information, see the *Dynamic Efficiency* Technical Information		

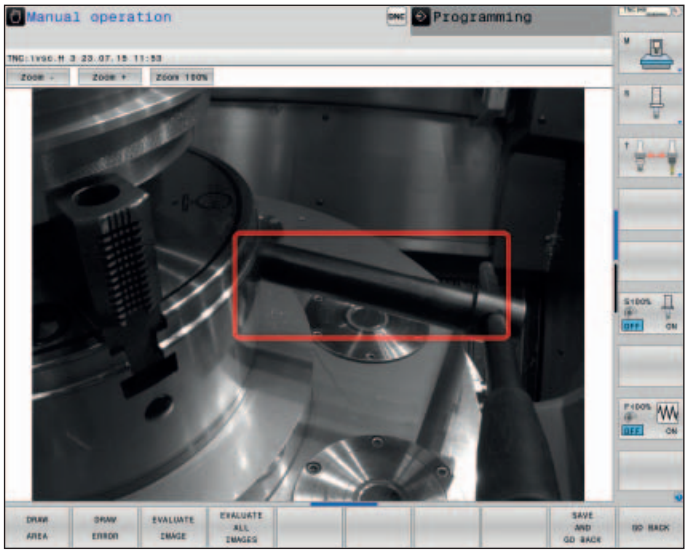
Machining functions
VSC – camera-based monitoring of the setup situation

With the **VSC** option (Visual Setup Control), 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 several places in the NC program at which the control conducts an optical comparison of the actual with the desired condition. If an error is detected, the TNC reacts as previously chosen 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
- Chip residues (before measurements, for example)

VSC not only helps you to avoid expensive damage to the tool, workpiece and machine, you can also use it to document the setup situation by saving photos By saving the 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.



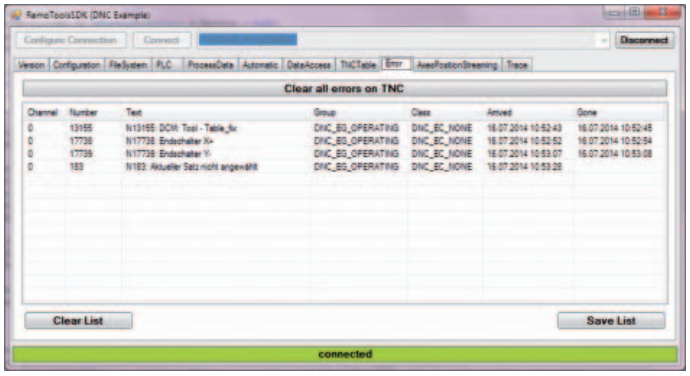


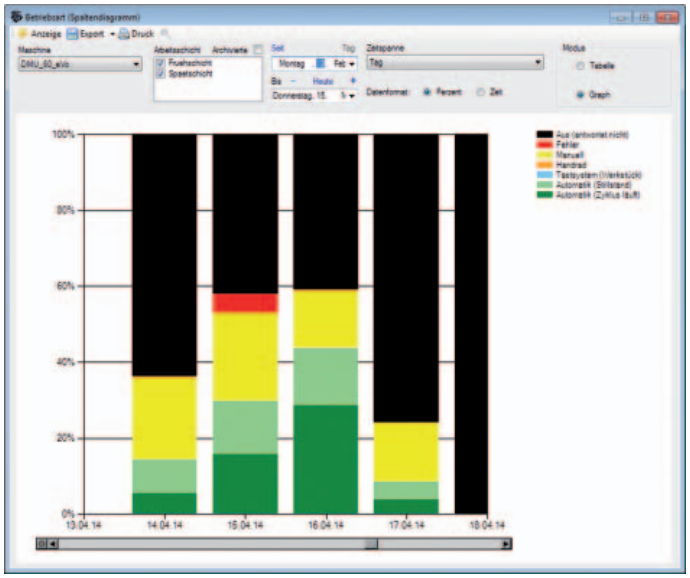
VSC	Option 136	ID 1099457-01
TNC 640 HSCI	As of NC SW 34049x-06	
TNC 620 HSCI	–	
TNC 320	–	
iTNC 530 HSCI	–	
iTNC 530	–	
Installation by the machine tool builder		
For more information, see the *TNC 640* brochure		

Communication
HEIDENHAIN DNC – communication over COM components

The development environments on Windows operating systems are particularly well suited as flexible platforms for application development in order to come to terms with the requirements of the machine's environment. 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 a very short time. The **HEIDENHAIN DNC** option enables a Windows application to access data of the TNC, and to edit the data if required. Possible fields of application include, for example:

- Software solutions controlling the manufacturing process
  - Machine and operating-data acquisition systems
  - 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 text message to his smartphone reporting problems on the currently running machining process
  - Overview plans that inform you about the current condition of all 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. During the installation of RemoTools SDK the COM component is registered in the Windows operating system.

connected + machining

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

Communication

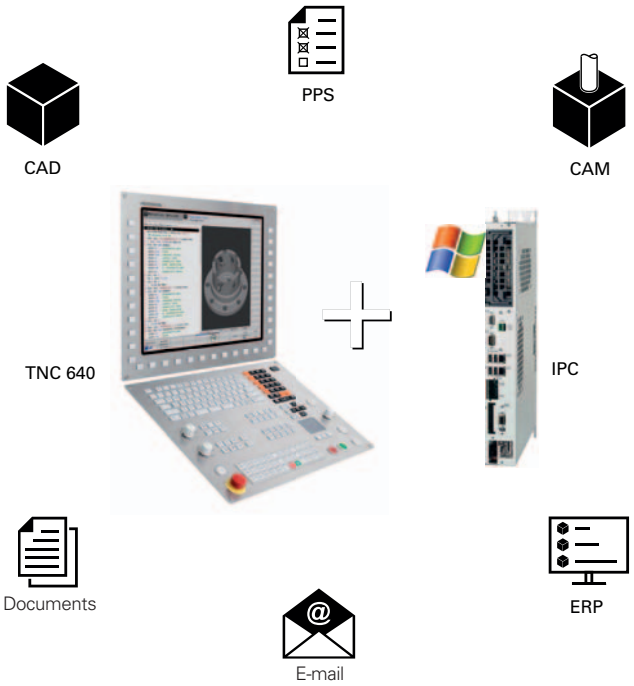
Remote desktop manager – display and remote operation of external computer units

In daily operations it can often be necessary to make entries in planning and control systems or 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 computer operates as an industrial PC (e.g. IPC 6641) in the machine's control cabinet, or as a server in the local network.

Possible applications include the central management of job orders or 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.

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



connected machining

Connected Machining permits uniformly digital order management in networked manufacturing You also profit from:

- Easy data usage
- Time-saving procedures
- Transparent processes

connected machining

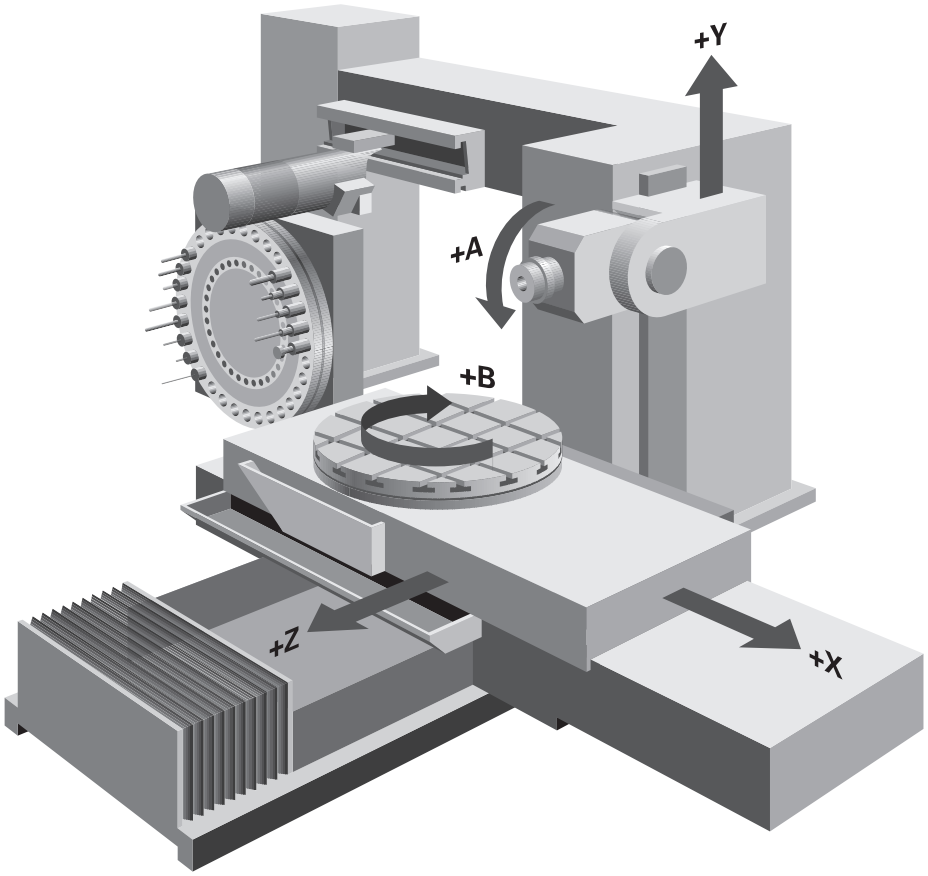
Remote desktop manager	Option 133	ID 894423-01
TNC 640 HSCI	As of NC SW 34059x-01	
TNC 620 HSCI	As of NC SW 81760x-01	
TNC 320	–	
iTNC 530 HSCI	As of NC SW 60642x-02	
iTNC 530	–	
Installation by IT specialists		
For more information, see the Technical Manuals		

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 max. possible number of control loops depends on the control:
- iTNC 530: 20 control loops
- TNC 640: 20 control loops
- TNC 620: 6 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

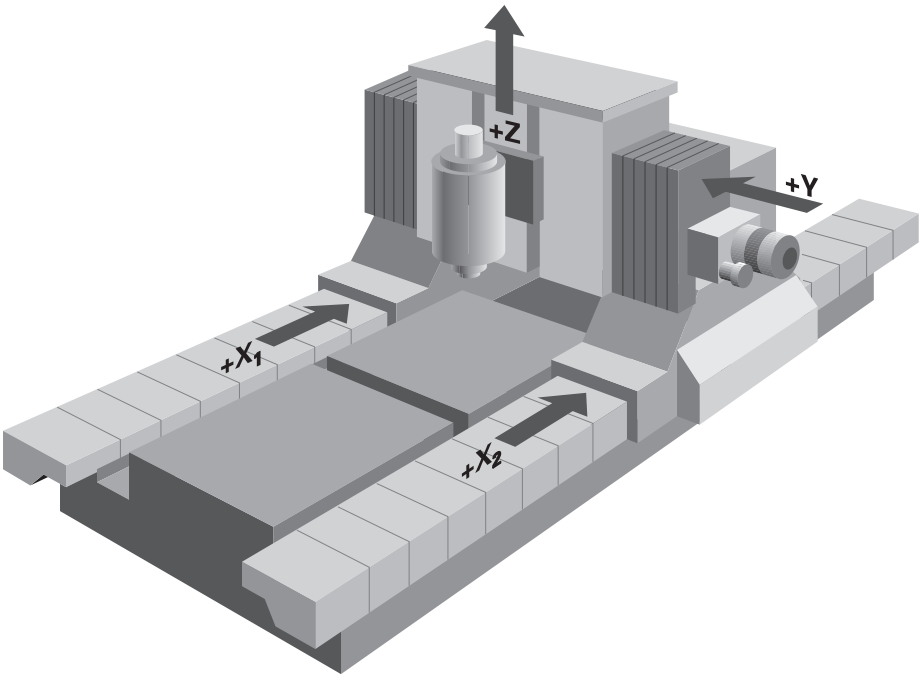
Interfacing to the machine

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. Fast and, above all, precise positioning movements are coordinated to match each other exactly and permit 5-axis simultaneous movements for very demanding tasks. More than one slave axis 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.



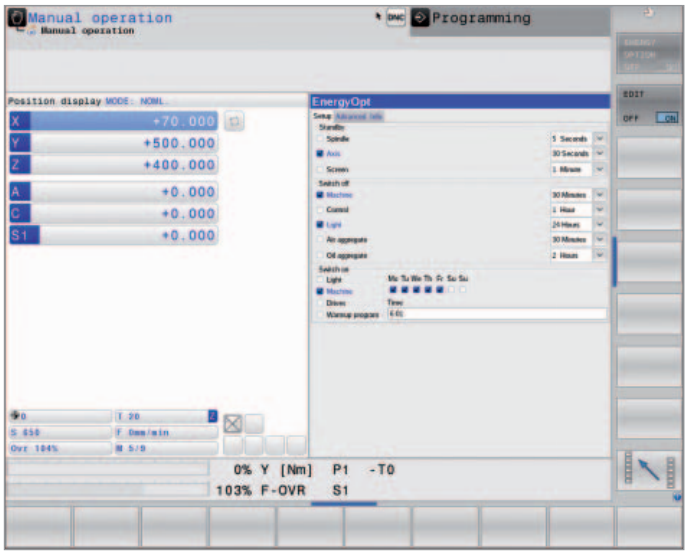
Synchronized axes	Option 24	ID 634621-01
TNC 640 HSCI	Standard	
TNC 620 HSCI	As of NC SW 34056x-01/73498x-01/81760x-01	
TNC 320	As of NC SW 34055x-01/771851-01	
iTNC 530 HSCI	Standard	
iTNC 530	Standard	
Installation by the machine tool builder		
For more information, see the Technical Manuals		

Interfacing to the machine
Python OEM process – realizing 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 used universally 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 PLC windows you are already familiar with. The applications can also be displayed in separate windows that are freely integrated in the TNC user interface up to the size of the entire TNC screen.



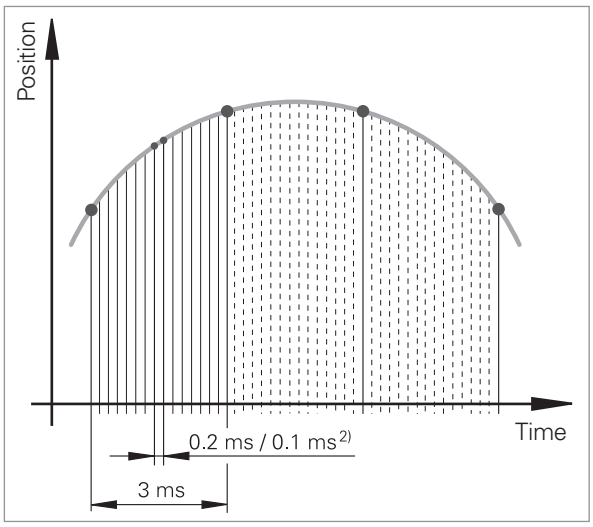
Python OEM Process	Option 46	ID 579650-01
TNC 640 HSCI	As of NC SW 34059x-01	
TNC 620 HSCI	As of NC SW 34056x-01/73498x-01/81760x-01	
TNC 320	As of NC SW 34055x-04/771851-01	
iTNC 530 HSCI	As of NC SW 60642x-01	
iTNC 530	As of NC SW 34049x-04	
Installation by IT specialists		
For more information, see 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, for which option 49 must be enabled.

Double-speed control loops permit higher PWM frequencies as well as shorter cycle times of the speed controller. This makes improved current control for spindles possible, and also 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}
3333 Hz	150 µs
4000 Hz	125 µs
5000 Hz	100 µs
6666 Hz	75 µs with option 49
8000 Hz	60 µs with option 49
10 000 Hz	50 µs with option 49

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 offers a reserved range in the option menu (SIK menu) that can be used for the machine tool builder's purposes.

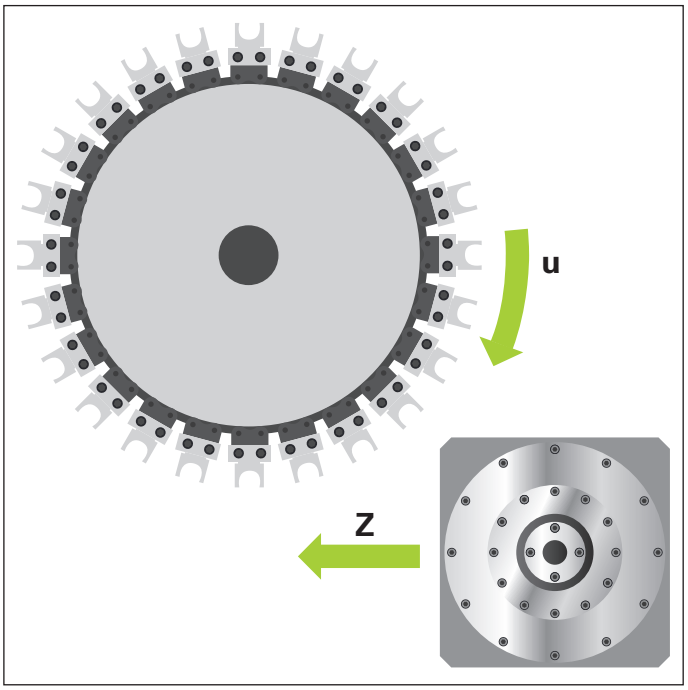
The option range 101 to 130 therefore provides 30 options that the machine tool builder can have activated and enabled by his own PLC program via verification. One advantage is the simple activation through the user by SIK menu without requiring on-site support by the machine tool builder.

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 **RTC** Real-Time Coupling function 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.



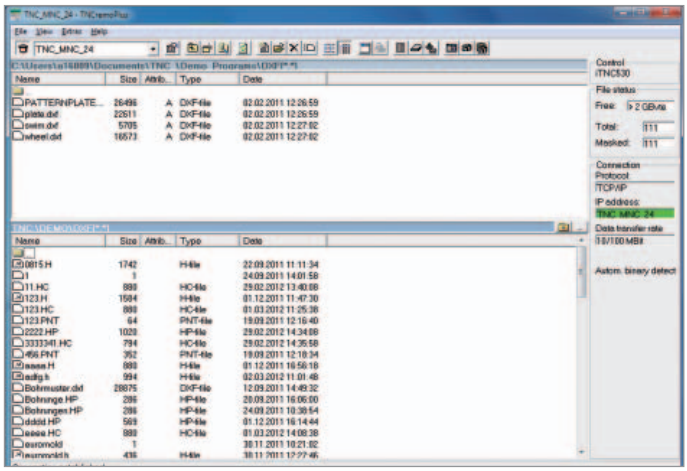
RTC	Option 135	ID 1085731-01
TNC 640 HSCI	As of NC SW 34059x-04	
TNC 620 HSCI	–	
TNC 320	–	
iTNC 530 HSCI	–	
iTNC 530	–	
Installation by the machine tool builder		
More information –		

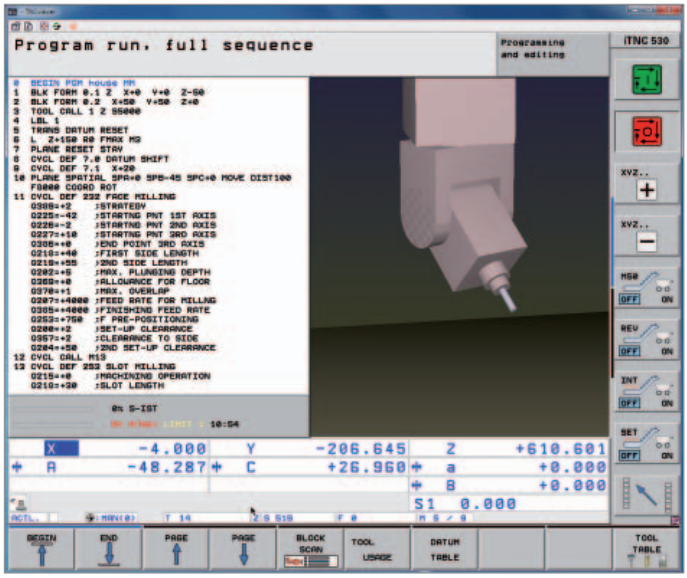
PC software
TNCremo – programs for data transfer

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). With TNCremo and an Ethernet or other data interface, you can bidirectionally transfer externally saved part programs, tool tables and pallet tables, 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 the log
- Print-out of screen contents
- Text editor
- Managing more than one machine

In addition to the features you are already familiar with 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 operate 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

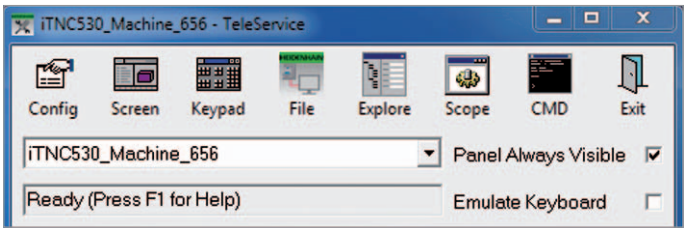
The PC software **TeleService** permits comprehensive remote diagnostics, as well as extensive remote operation and remote monitoring of HEIDENHAIN controls. This makes in-depth troubleshooting possible. The service technician communicates online with the control over modem, ISDN or over the Internet,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

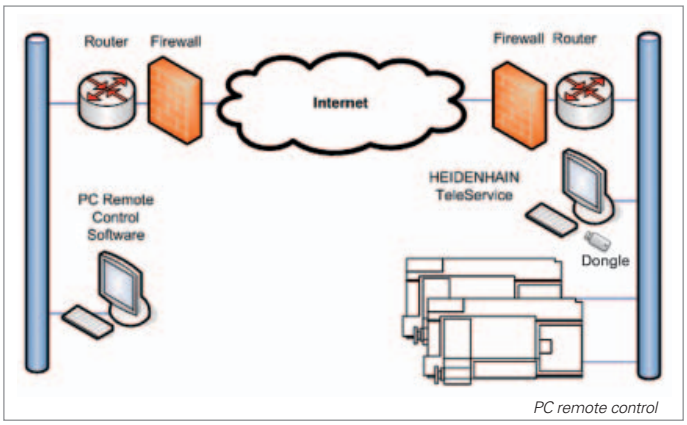
The machine tool builder creates the necessary diagnostic user interface according to his service requirements for checking the desired information. The TeleService control panel is used for operation.

Functions
- Remote operation of the TNC with on-line 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.

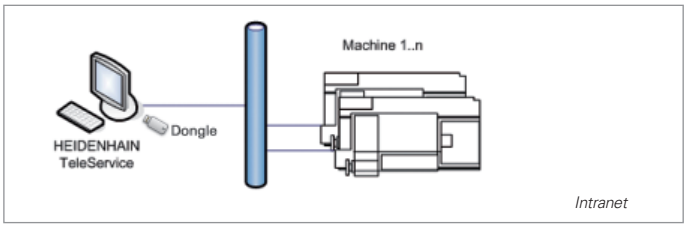
TeleService by the machine tool builder
The machine tool builder builds a network of his machines with TNCs at the customer's site and a network of service PCs in his service department (with TeleService installed). Routers connect the two networks over the public telephone and data lines. When the customer presses the "Service" or "Support" soft key, the routers automatically set up a connection between the customer's network and that of the machine manufacturer. The service technician has access over TeleService to all 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 in the customer's intranet. Here, a PC with TeleService installed is connected directly (without a router) to the network of TNCs. This makes remote operation, remote monitoring and remote diagnostics possible of the machines within the customer's own network.





Remote connection over PC remote control software



TeleService in the company network

TeleService Single station license Network license for up to 14 participants	CD-ROM with dongle ID 340449-xx ID 340454-xx
TNC 640 HSCI **TNC 620 HSCI** **TNC 320** **iTNC 530 HSCI** **iTNC 530**	As of NC SW 34059x-01 As of NC SW 34056x-01/73498x-01/81760x-01 As of NC SW 34055x-01/771851-01 As of NC SW 60642x-01 As of NC SW 34049x-01
Installation by the machine tool builder	
For more information, see the *Diagnostics for HEIDENHAIN Controls* Product Information	

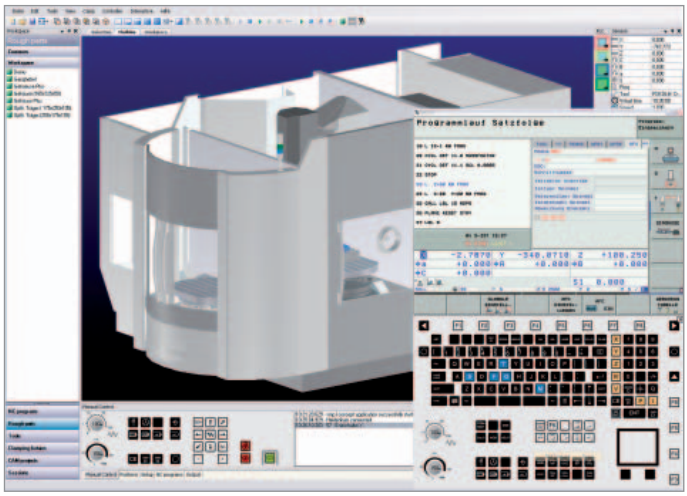
PC software
virtualTNC – control of virtual machines

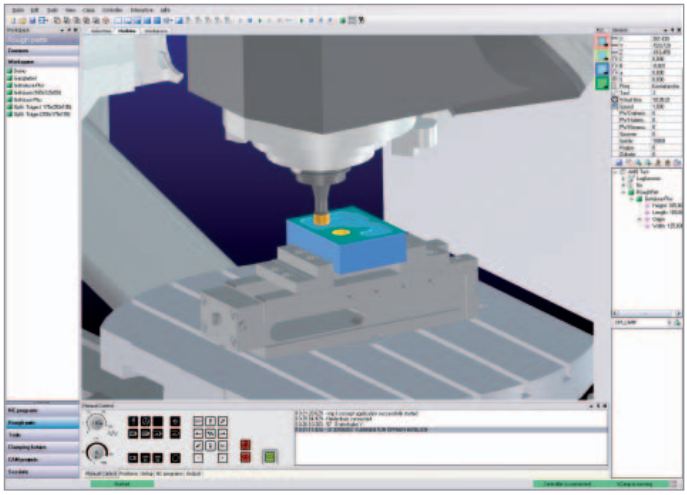
Simulation of NC program runs on the control has been a feature of HEIDENHAIN contouring controls for some time now. The PC software **virtualTNC** now makes it possible to use the TNC as a control component for machine-simulation applications (virtual machine) on external computer systems.

Principle of function of a virtual machine with virtualTNC
Machine-simulation applications (virtual machines) can completely simulate production units in order to optimize production processes in the field beforehand. virtualTNC 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.

virtualTNC 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 **Remo-Tools SDK 3.0** software development package and its help system.



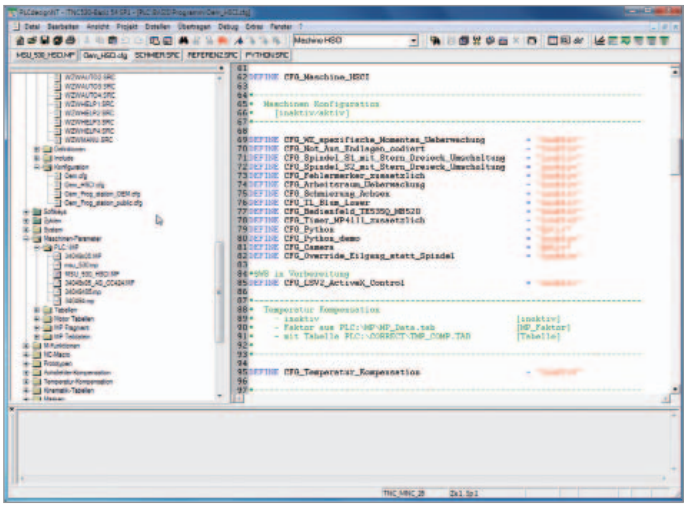


virtualTNC	**TNC 640 HSCI**	**iTNC 530 HSCI** **iTNC 530**
Single station license	ID 1113933-01	
Network license, 14 stations	ID 1113935-01	
Network license, 20 stations	ID 1113936-01	
Installation for manufacturers of machine-simulation applications		
For more information, see the *HEIDENHAIN DNC* brochure		

PC software

PLCdesign – software for PLC program development

The **PLCdesign** software can be used to easily create PLC programs. Comprehensive 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

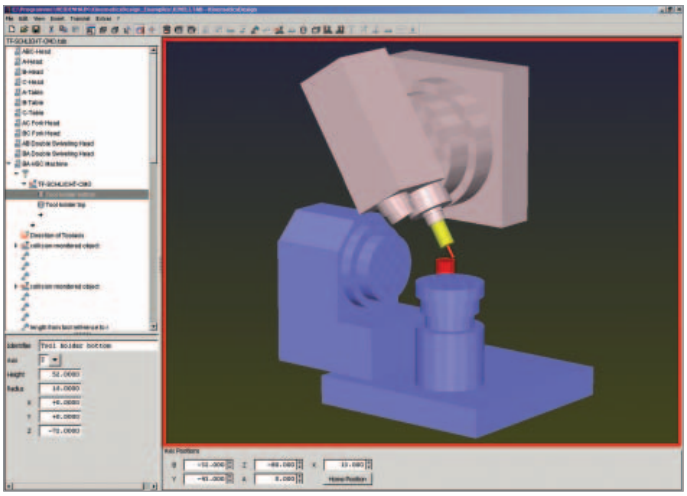
KinematicsDesign is a PC program for creating adaptable kinematic configurations. KinematicsDesign also provides a convenient way to configure the DCM collision monitoring function and to put it into service.

The software makes complete generation possible 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 comprehensive possibilities for displaying range from a pure listing of the transformation chain to a wire model to the depiction of the entire work envelope.

With the TNC 640, you can transfer collision objects out of a CAD file and integrate them into the machine kinematics using the M3D format. The M3D data format from HEIDENHAIN permits an especially finely detailed depiction of high-resolution collision objects. The M3D converter that 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

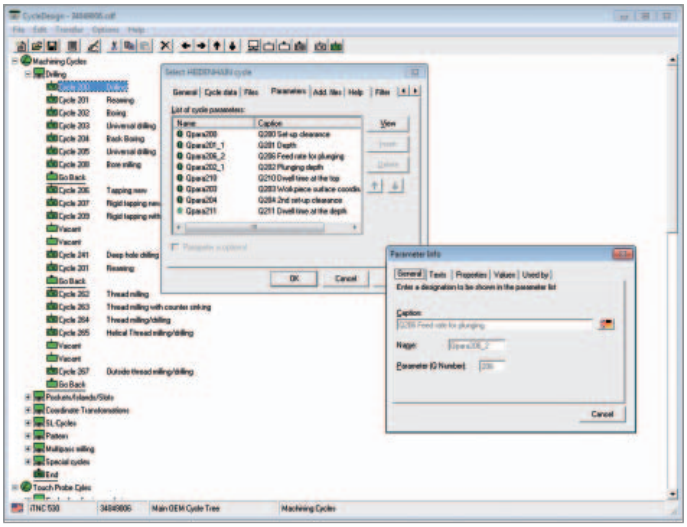
CycleDesign – saving NC subprograms as cycles

For frequently recurring operations, the HEIDENHAIN controls provide you with parameterizable NC subprograms, also referred to as cycles. 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 also use CycleDesign to transfer the cycle data to the memory of the control.

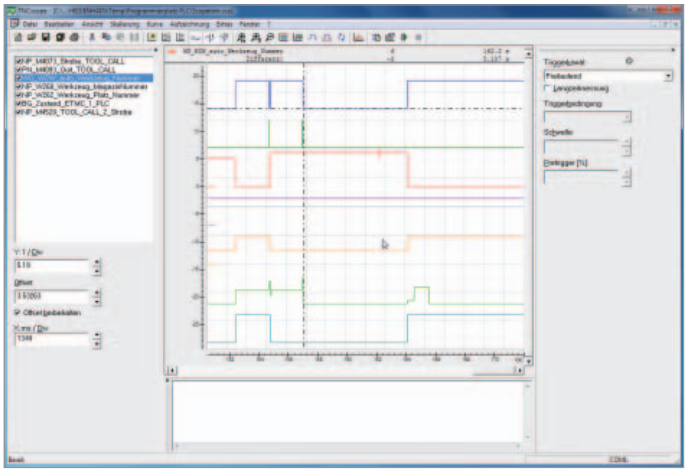


CycleDesign	Free download
TNC 640 HSCI	As of NC SW 34059x-01
TNC 620 HSCI	As of NC SW 34056x-01/73498x-01/81760x-01
TNC 320	As of NC SW 34055x-01/771851-01
iTNC 530 HSCI	As of NC SW 60642x-01
iTNC 530	As of NC SW 34049x-01
Installation for users and machine manufacturers	
For more information, see the *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, but also PLC operands from HEIDENHAIN controls. TNCscope enables you to make a convenient diagnosis from a Windows PC. Through a simple connection over Ethernet, 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 curves to another file
- Printing 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)

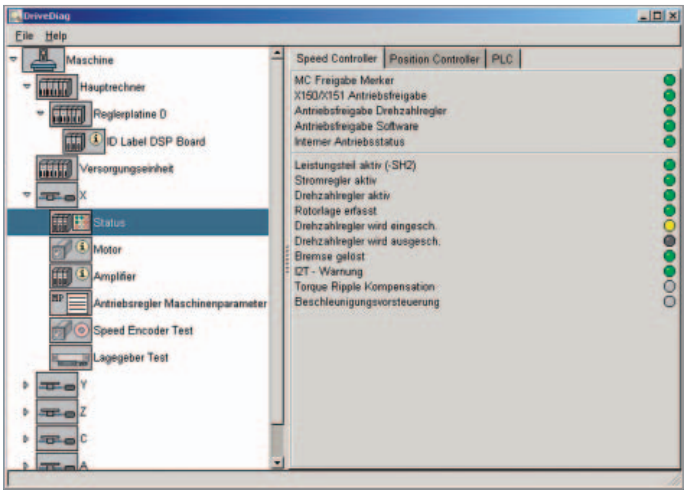
TNCscope	Download
TNC 640 HSCI	As of NC SW 34059x-01
TNC 620 HSCI	As of NC SW 34056x-01/73498x-01/81760x-01
TNC 320	As of NC SW 34055x-01/771851-01
iTNC 530 HSCI	As of NC SW 60642x-01
iTNC 530	As of NC SW 34049x-01
Installation for machine manufacturers and service	
For more information, see the Technical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

PC software
DriveDiag – diagnosis of digital drive systems

The **DriveDiag** software for PCs enables the service technician to make a simple and fast diagnosis of the drives, starting from the motor all the way to the drive control. After a connection is set up between DriveDiag and the control, various signals can be called from the control. In particular with its 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 also available to the controller, such as motor temperature and DC-link voltage
- 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 and inverters, of 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 gives access to comprehensive testing possibilities with DriveDiag.



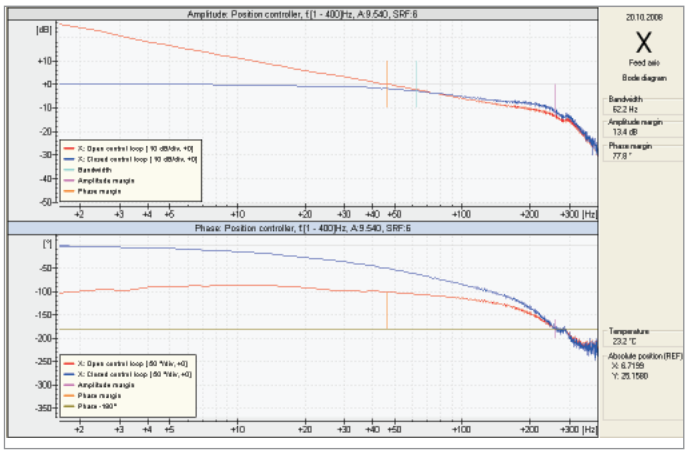
Graphically supported, dynamic display of status signals

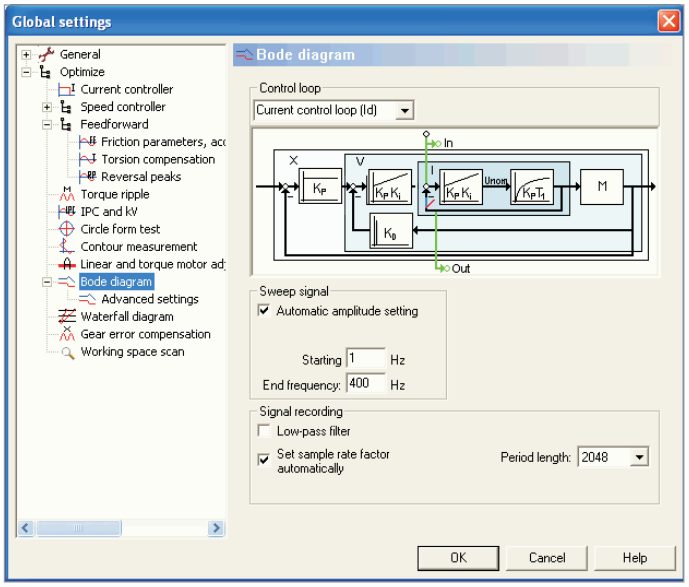
DriveDiag	Download
TNC 640 HSCI	As of NC SW 34059x-01
TNC 620 HSCI	As of NC SW 34056x-01/73498x-01/81760x-01
TNC 320	–
iTNC 530 HSCI	As of NC SW 60642x-01
iTNC 530	As of NC SW 34049x-01
Installation for machine manufacturers and service	
For more information, see the *Diagnostics for HEIDENHAIN Controls*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

PC software
TNCopt – for commissioning digital control loops

High-end machine tools have to operate increasingly faster and more precise. High performance is demanded from the drive system with servo motors and spindles. This is why HEIDENHAIN makes efficient and intelligent control technology a first priority. 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 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





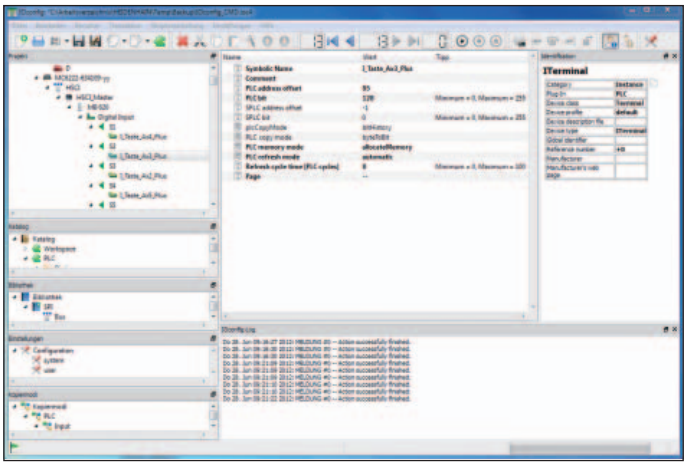
TNCopt	Download
TNC 640 HSCI	As of NC SW 34059x-01
TNC 620 HSCI	As of NC SW 34056x-01/73498x-01/81760x-01
TNC 320	–
iTNC 530 HSCI	As of NC SW 60642x-01
iTNC 530	As of NC SW 34049x-01
Installation for machine manufacturers and service	
For more information, see the *Information for the Machine Tool Builder*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

PC software

IOconfig – configuration of the I/O and HSCI components

Modern machine tools are becoming ever more complex and are shipped with extensive equipment. **IOconfig** supports you in the configuration of HSCI control components and the periphery (e.g. Profibus, ProfiNet) so that you can quickly and conveniently integrate all 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 in the commissioning of HSCI components and SPI additional modules.



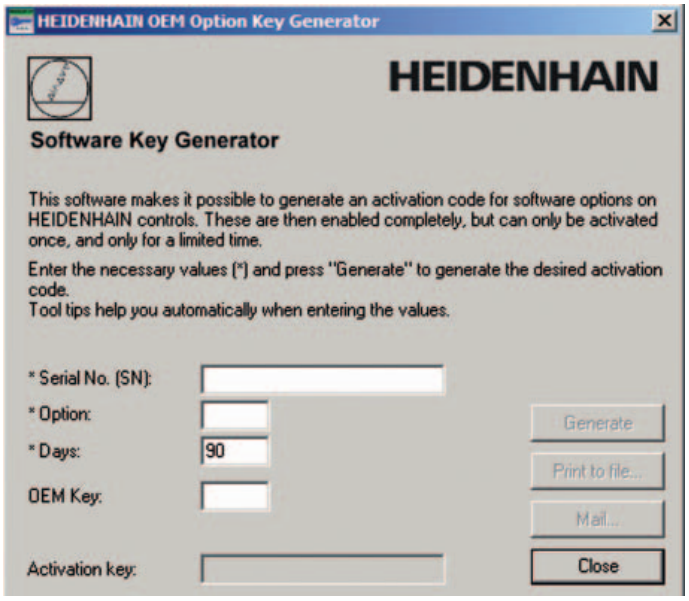
IOconfig	Download
TNC 640 HSCI	As of NC SW 34059x-01
TNC 620 HSCI	As of NC SW 34056x-01/73498x-01/81760x-01
TNC 320	–
iTNC 530 HSCI	As of NC SW 60642x-01
iTNC 530	As of NC SW 34049x-01
Installation for machine manufacturers and service	
For more information, see the Technical Manuals and the integrated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

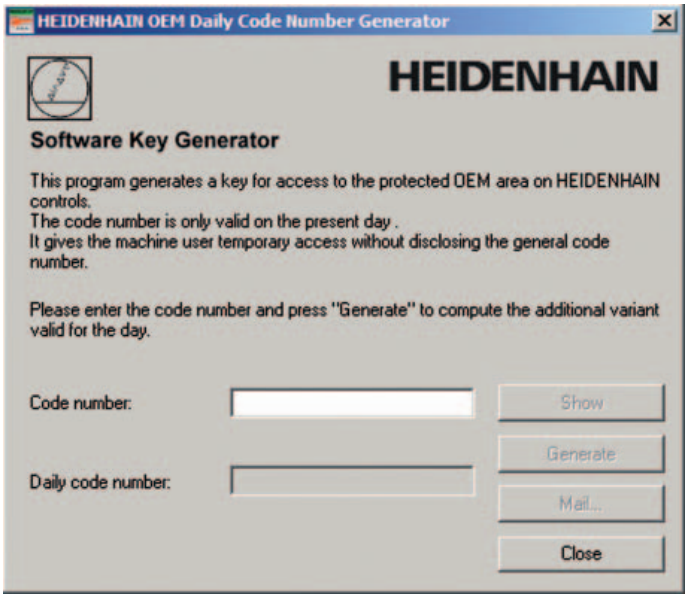
PC software
TNCkeygen – enabling key for software options

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). It can only be enabled 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 enables the customer to extensively test the available options (e.g. the DXF converter) without having to buy them. If the test is successful, he 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 enables the operator to access the area on the day the key was 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	

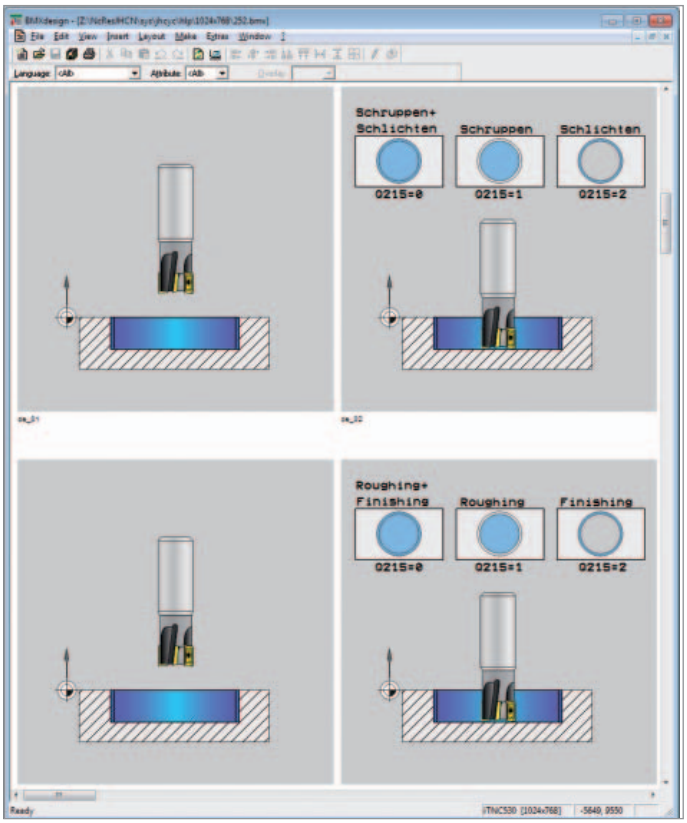
PC software
BMXdesign

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 at 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 variants
- Replacement of database IDs with plain-language texts from multi-lingual files
- Printout with page preview
- Generating BMX files
- Bitmap (*.BMP) export function
- Integration of PLCtext for managing BMX files



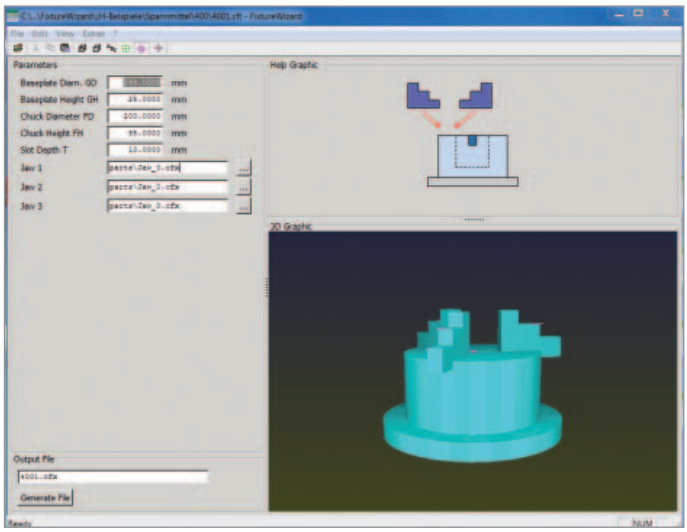
BMXdesign	ID 340443-xx
TNC 640 HSCI	As of NC SW 34059x-01
TNC 620 HSCI	As of NC SW 34056x-01/73498x-01/81760x-01
TNC 320	As of NC SW 34055x-01/771851-01
iTNC 530 HSCI	As of NC SW 60642x-01
iTNC 530	As of NC SW 34049x-01
Installation for the machine tool builder	
For more information, see the *Information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

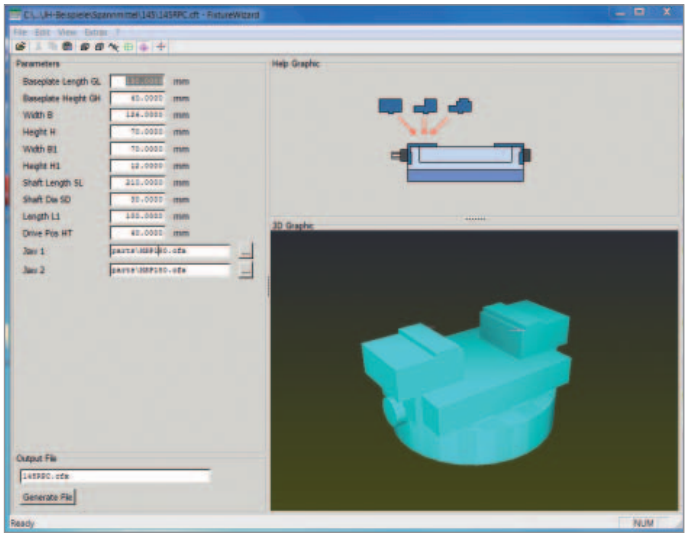
PC software
FixtureWizard

FixtureWizard lets you handily create defined fixtures or carrier kinematics from fixture templates or tool-carrier kinematic models. You can then integrate the files you create into the collision monitoring 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 suits 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 with an appropriate function, transfer it to the TNC and use the fixture management to embed it in the collision monitoring.





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?
Everyone knows that you can quite easily write a part program with the TNC at the machine, even while it's machining another 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 to program just as you do at the machine, but away from the noise and distractions of the shop floor.

Creating programs
Programming, testing and optimizing your smarT.NC (on the iTNC 530), HEIDENHAIN conversational or DIN/ISO programs for the TNC with the programming station substantially reduces machine idle times. You need not adjust your way of thinking—every keystroke fits. On the programming station you program on the same keyboard as at the machine.

Testing of programs created offline
Of course 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 experience needed to enable him to safely operate the machine later. Because it can be programmed with smarT.NC, in HEIDENHAIN conversational language and in DIN/ISO, 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 unchanged except that it now includes the soft keys, which are otherwise integrated in the visual display unit. You connect the TNC keyboard to your PC's USB port. 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 1113924-02		
Network license, 1 station	ID 1125955-02		
Network license, 14 stations	ID 1113926-02		
Network license, 20 stations	ID 1113928-02		
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 handwheels

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 through the feed motors in direct relation to the rotation of the handwheel. For delicate operations you can set the transmission ratio to certain preset distances per handwheel revolution.

HR 130 and HR 150 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. An adapter permits connection of up to three HR 150 electronic panel-mounted handwheels.

HR 510, HR 520 and HR 550 portable handwheels
The HR 510, HR 520 and HR 550 portable handwheels are particularly helpful for when you have to work close to the machine's working space. The axis keys and certain functional 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 informs you directly of the most important operating conditions. As a wireless handwheel, the HR 550 is ideal for use on large machine tools. When you aren't using the handwheel you can simply place it in the HRA 551 FS handwheel holder (transmitter/receiver unit with integrated charging unit).

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 **HR 150**	ID 540940-01/540940-03 ID 540940-06/540940-07	34059x-01 34059x-01	34056x-01/73498x-01/81760x-01 34056x-01/73498x-01/81760x-01	34055x-01 34055x-01	60642x-01 60642x-01	34049x-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 670303-01/970302-01 ID 670305-01/670304-01 ID 602622-03/598515-03	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 731928-02	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 in the workshop and in series production: Together with the TNC, touch probes can manually or automatically perform setup, measuring and inspection functions.
- Workpiece alignment
- Datum setting
- Workpiece measurement

The touch probes for workpiece measurement are inserted in the tool holder either manually or by the tool changer. They 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 either by cable, radio or an infrared beam to the control.

Touch probes with **cable connection for signal transmission** for machines with manual tool change and for grinding machines and lathes:
TS 260 – New generation, axial or radial cable

Touch probe with **radio or infrared signal transmission** for machines with automatic tool change:
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 444 – Battery-free voltage supply through integrated air turbine generator from compressed air, for infrared transmission, with compact dimensions
TS 740 – High probing accuracy and repeatability, low probing force, with infrared transmission

SE 660


TS 260

TS 460

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 737624-xx	34059x-01 SP1	81760x-01 SP1	771851-01	60642x-01	34049x-01
TS 444 **TS 740**	ID 588008-xx ID 573757-xx	34059x-01 34059x-01	34056x-01/73498x-01/81760x-01 34056x-01/73498x-01/81760x-01	34055x-01 34055x-01	60642x-01 60642x-01	34049x-01 34049x-01
TS 230 **TS 440** **TS 640**		34059x-01 34059x-01 34059x-01	34056x-01/73498x-01/81760x-01 34056x-01/73498x-01/81760x-01 34056x-01/73498x-01/81760x-01	34055x-01 34055x-01 34055x-01	60642x-01 60642x-01 60642x-01	34049x-01 34049x-01 34049x-01
Installation by the machine tool builder						
For more information, see the *Touch Probes* brochure						

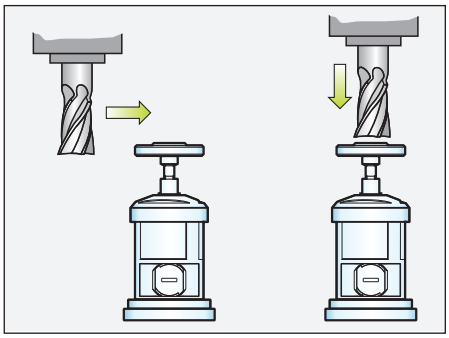
Hardware enhancements
TT, TL – tool touch probes

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 periodic inspection of the tool are necessary for wear and breakage, as well as the shape of each tooth. For tool measurement, HEIDENHAIN offers the **TT triggering tool touch probes** as well as the non-contacting **TL laser systems**.

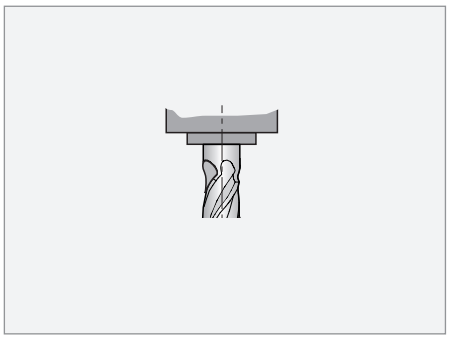
The systems are installed directly in the machine's workspace, where they permit tool measurement either before machining or during interruptions.

The **TT tool touch probes** measure the tool length and radius. When probing the rotating or stationary tool, e.g. during individual tooth measurement, the contact plate is deflected and a trigger signal is transmitted directly to the TNC. The **TT 160** uses signal transmission by cable, whereas the **TT 460** operates with wireless signal transmission over radio or an infrared beam. It is therefore particularly suitable for use on rotary and tilting tables.

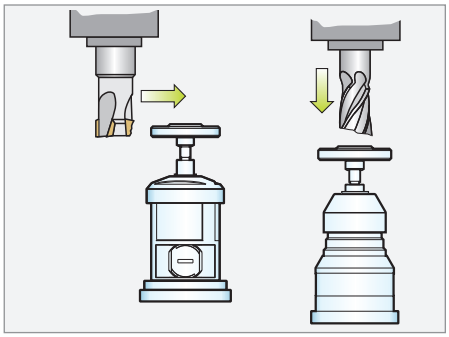
The **TL Nano** and **TL Micro laser systems** are available for various maximum tool diameters. Using a laser beam, they probe the tool without contact, and can detect form errors of individual teeth along with the tool length and radius.



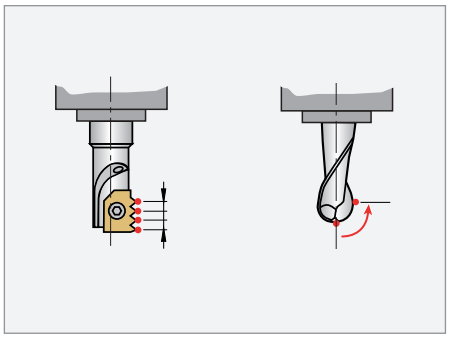
TT tool touch probe
Tool length and radius measurement with stationary or rotating spindle



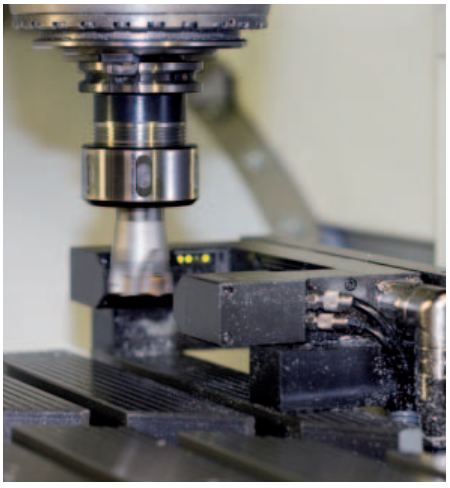
TL laser systems
Tool radius measurement, with detection of tooth breakage



Tool wear measurement and tool breakage monitoring



Inspection of individual tooth shape



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 728346-xx	34059x-05 34059x-04 SP1	81760x-02 81760x-01 SP1	771851-02 771851-01	60642x-01 60642x-01	34049x-01 34049x-05
TT 140 **TT 449**		34059x-01 34059x-02	34056x-01/73498x-01/81760x-01 34056x-04/73498x-02/81760x-01	34055x-03 34055x-06	60642x-01 60642x-01	34049x-01 34049x-05
TL Nano **TL Micro 150** **TL Micro 200** **TL Micro 300**	ID 557690-xx ID 557684-xx ID 557685-xx ID 557686-xx	34059x-02 34059x-02 34059x-02 34059x-02	34056x-04/73498x-02/81760x-01 34056x-04/73498x-02/81760x-01 34056x-04/73498x-02/81760x-01 34056x-04/73498x-02/81760x-01	34055x-06 34055x-06 34055x-06 34055x-06	60642x-01 60642x-01 60642x-01 60642x-01	34049x-01 34049x-01 34049x-01 34049x-01
Installation by the machine tool builder						
For more information, see the *Touch Probes* brochure						

Hardware enhancements
ITC – additional operating station

The additional ITC 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. The remote operation, which was designed for the TNC, permits very simple connection of the ITC via a standard Ethernet connection with a cable of up to 100 meters.

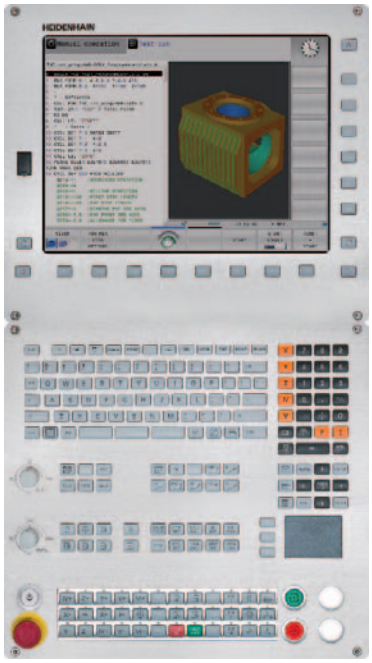
The **ITC 755** is a compact additional operating station for control systems with a 15-inch or 19-inch main screen. It provides the most important function keys of the TNC in addition to an ASCII keyboard and a touch screen. The ITC 755 adjusts its resolution automatically to fit the size of the main screen. The soft keys are operated using the touch screen—keys for the soft keys are therefore not needed.

Together with the TE 73x and the TE 74x respectively, the **ITC 750** (15-inch screen) and **ITC 760** (19-inch screen) each make up one complete second operating station. They are operated in the same way as the TNC.

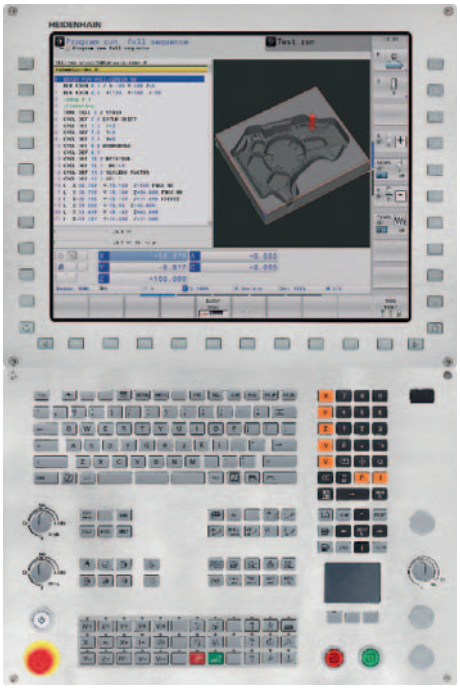
Convenience through plug & play
It is surprisingly simple to connect an ITC: As soon as the TNC identifies an ITC, it provides it with a current operating system. After the ITC has been started, the complete content of the main screen is mirrored to the ITC's screen. Due to this plug & play principle, it is not necessary for the hardware manufacturer to pre-configure the operating station. With a standard X116 Ethernet interface, the TNC automatically integrates the ITC in the system.

You can switch between the TNC and the ITC by directly taking over control, or by using a hand-over method (selectable). The USB interface of the ITC provides a direct connection to your control's file system. The TNC handles the shutting down of the ITC just as reliably as it handles the starting up, thereby ensuring maximum operational reliability.

ITC 755



ITC 750 with TE 735



ITC 760 with TE 745

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 760	ID 827086-01	34059x-04	–	–	60642x-04	–
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. The option 133 is required.

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. No second screen is necessary, 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

Industrial PC		**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
IPC 6641	ID 1039543-01	34059x-04	81760x-01	–	60642x-04	–
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 into the machine's working space. The protective housing features a closing cover and connections for sealing air to prevent the camera optics from being damaged. 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 adapt the VSC inspection procedure optimally to the 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 –					

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 31-1345

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 3026, Australia
E-mail: vicsales@fcrmotion.com

BE **HEIDENHAIN NV/SA**
1760 Roosdaal, Belgium
www.heidenhain.be

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

BR **DIADUR Indústria e Comércio 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

HK **HEIDENHAIN LTD**
Kowloon, Hong Kong
E-mail: sales@heidenhain.com.hk

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.**
Tel Aviv 61570, 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

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**
01133 Kiev, Ukraine
www.heidenhain.ua

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

VE **Maquinaria Diekmann S.A.**
Caracas, 1040-A, Venezuela
E-mail: purchase@diekmann.com.ve

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

Zum Abheften hier falzen! / *Fold here for filing!*

827222-25 · 10 · 9/2016 · F&W · Printed in Germany