HEIDENHAIN

Options and Accessories

For TNC Controls

September 2012

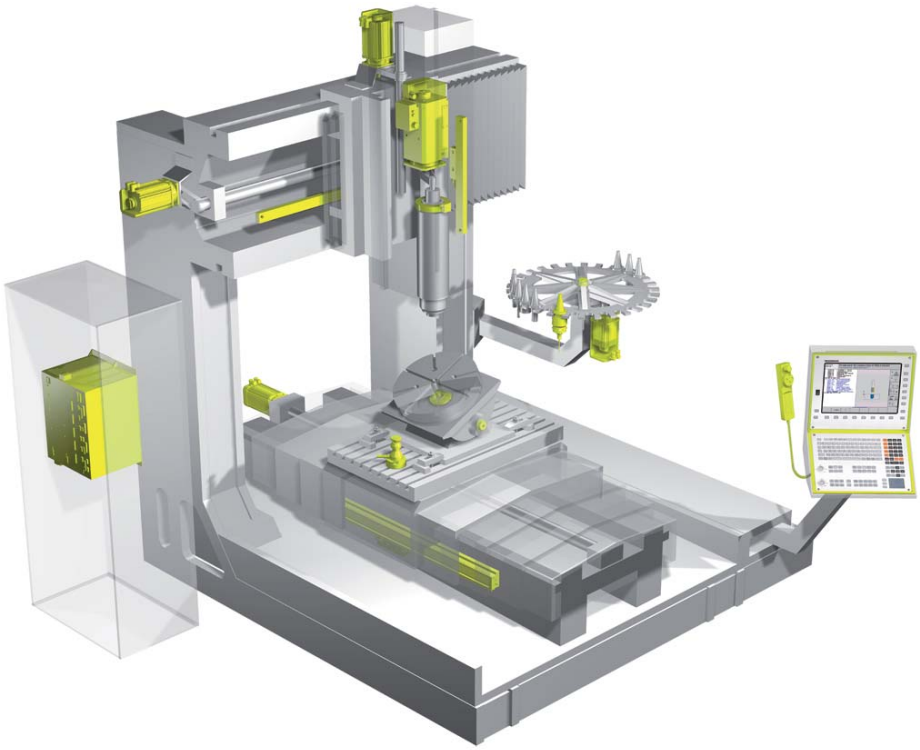
Options and accessories for TNC controls

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								
Additional conversational languages Slovenian Slovak Latvian Norwegian Korean Estonian Turkish Romanian Lithuanian	**41**	530184-01 530184-02 530184-03 530184-04 530184-06 530184-07 530184-08 530184-09 530184-10	•	•	•	•		9
Machining with a rotary table • Programming of cylindrical contours as if in two axes • Feed rate in mm/min or degrees/min	**8**	367591-01	•	•	•	•	X	10
Coordinate transformation Tilting the working plane, PLANE function	**8**	367591-01	•	•	•	•	X	11
Display step to 0.01 µm or 0.000 01°	**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	12
Advanced 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	**19**	628252-01	S	•	S	S		13
Program verification graphics, program-run graphics • Plan view • Projection in three planes • 3-D view	**20**	628253-01	S	•	S	S		14
Pallet management	**22**	628255-01	–	•	S	S	X	15
DXF converter – importing contours and machining positions from DXF files	**42**	526450-01	–	•	•	•		16
Turning functions • Tool management for turning • Tool-tip radius compensation • Switching between milling and turning modes of operation • Lathe-specific contour elements • Turning cycle package	**50**	634 608-01	–	–	•	–		17

• = 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								
Advanced tool management (only with option 9)	**93**	676938-01	–	•	•	•	X	18
Interpolating spindle – interpolation turning	**96**	751653-01	–	–	–	•	X	19
CAD Viewer – opening 3-D CAD data directly on the TNC	**98**	800553-01	–	–	–	•		20
Machine accuracy								
KinematicsOpt – touch probe cycles for automatic measurement of rotary axes	**48**	630916-01	–	•	•	•		21
KinematicsComp – 3-D spatial compensation	**52**	661879-01	–	–	–	•		22
Machining functions								
Interpolation – circular in 3 axes with tilted working plane	**8**	367591-01	•	•	•	•		11
Interpolation – linear in 5 axes	**9**	367590-01	–	•	•	•		–
Spline interpolation – processing third-degree polynomials	**9**	367590-01	–	–	–	•		–
5-axis simultaneous machining • 3-D tool compensation through surface normal vectors • Tool center point management (TCPM): Using the electronic handwheel to change the angle of the swivel head during program run without affecting the position of the tool point • Keeping the tool normal to the contour • Tool radius compensation normal to the tool direction • Manual traverse in the active tool-axis system	**9**	367590-01	–	•	•	•	X	23
Handwheel superimpositioning – superimposing handwheel positioning during program run	**21**	628254-01	–	•	S	S	X	24
Tool compensation – radius-compensated contour LOOK-AHEAD	**21**	628254-01	S	•	S	S		25
DCM – dynamic collision monitoring	**40**	526452-01	–	–	•	•	X	26
Global program settings	**44**	576057-01	–	–	–	•	X	27
AFC – adaptive feed control	**45**	579648-01	–	–	•	•	X	28
3D-ToolComp – 3-D radius compensation depending on the tool's contact angle	**92**	679678-01	–	–	–	•		29
Communication								
HEIDENHAIN-DNC – communication with external PC applications over COM component	**18**	526451-01	•	•	•	•	X	30
Remote Desktop Manager – Display and operation of external computer units (e.g. a Windows PC)	**133**	894423-01	–	–	•	•	X	31

• = 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
Interfacing to the machine									
Additional control loops	Additional axis 1	0	354540-01	•	•	•	•	X	32
	Additional axis 2	1	353904-01	•	•	•	•	X	32
	Additional axis 3	2	353905-01	–	–	•	•	X	32
	Additional axis 4	3	367867-01	–	–	•	•	X	32
	Additional axis 5	4	367868-01	–	–	•	•	X	32
	Additional axis 6	5	370291-01	–	–	•	•	X	32
	Additional axis 7	6	370292-01	–	–	•	•	X	32
	Additional axis 8	7	370293-01	–	–	•	•	X	32
	4 additional control loops	77	634613-01	–	–	•	•	X	32
	8 additional control loops	78	634614-01	–	–	•	•	X	32
Synchronized axes – gantry axes, tandem tables		24	634621-01	•	•	S	S	X	33
Python OEM process – realizing special functions		46	579 650-01	•	•	•	•	X	34
Double speed – short control-loop cycle times for direct drives		49	632223-01	–	•	•	•	X	35
Cross Talk Compensation (CTC) – compensation of position error through axis coupling		141	800542-01	–	•	•	•	X	36
Position Adaptive Control (PAC) – position-dependent adaptation of controller parameters		142	800544-01	–	•	•	•	X	37
Load Adaptive Control (LAC) – load-dependent adjustment of controller parameters		143	800545-01	–	•	•	•	X	38
Motion Adaptive Control (MAC) – motion-dependent adaptation of controller parameters		144	800546-01	–	•	•	•	X	39
Active Chatter Control (ACC) – suppression of tool chatter		145	800547-01	–	•	•	•	X	40
OEM option		101 – 130	579651-01 – 579651-30	–	–	–	•	X	41

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

PC software		TNC 320	TNC 620	TNC 640	iTNC 530	Adaptation by machine tool builder	Page
TNCremo		•	•	•	•		42
TNCremoPlus		•	•	•	•		42
TeleService		•	•	•	•	X	43
RemoTools SDK		•	•	•	•	X	30
virtualTNC		–	–	–	•	X	44
PLCdesign		•	•	•	•	X	45
KinematicsDesign		•	•	•	•	X	46
CycleDesign		•	•	•	•	X	47
TNCscope		•	•	•	•	X	48
DriveDiag		–	•	•	•	X	49
TNCopt		–	•	•	•	X	50
IOconfig		–	•	•	•	X	51
Software Key Generator		•	•	•	•	X	52
BMXDesign		•	•	•	•	X	53
FixtureWizard		–	–	–	•	X	54
Programming station	**Single license trial version**	•	•	•	•		55
	Single license with TNC keyboard	•	•	•	•		55
	Single license with virtual keyboard	•	•	•	•		55
	Network license with virtual keyboard For 14 users	•	•	•	•		55
	Network license with virtual keyboard For 20 users	•	•	•	•		55

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

Overview

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	•	•	•	•		56
	HR 150 11 µAss panel mounted	With detent W/o detent	540940-06 540940-07	•	•	•	•		56
	HR 410 portable handwheel	With detent W/o detent	535220-05 296469-55	•	•	•	•		56
	HR 410 FS Portable handwheel	With detent W/o detent	578114-11 337159-21	•	•	•	•		56
	HR 520 portable handwheel with display	With detent W/o detent	670303-01 670302-01	•	•	•	•		56
	HR 520 FS portable handwheel with display	With detent W/o detent	670305-01 670304-01	•	•	•	•		56
	HR 550 FS portable wireless handwheel w/ display	With detent W/o detent	606622-03 598515-03	•	•	•	•		56
	HRA 551 FS handwheel holder for HR 550 FS		731928-02	•	•	•	•		56
Workpiece touch probe	**TS 220** with cable		293488-xx	•	•	•	•		57
	TS 440 with infrared transmission **TS 444** with infrared transmission		620046-xx 588008-xx	•	•	•	•		57
	TS 640 with infrared transmission		620189-xx	•	•	•	•		57
	TS 740 with infrared transmission		573757-xx	•	•	•	•		57
Tool touch probe	**TT 140** with cable		527797-xx	•	•	•	•		58
	TT 449 with infrared transmission		593007-01	•	•	•	•		58
	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	•	•	•	•		58
Industrial PC	**IPC 6110**		597968-03	–	–	–	•		59
	IPC 6120 with complete TNC keyboard		664939-xx	–	–	–	•		59
	IPC 6341 for Windows interface on TNC		749963-01	–	–	•	•		59

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

Programming and operation
Additional conversational languages

The following conversational languages are available as standard on the TNC:

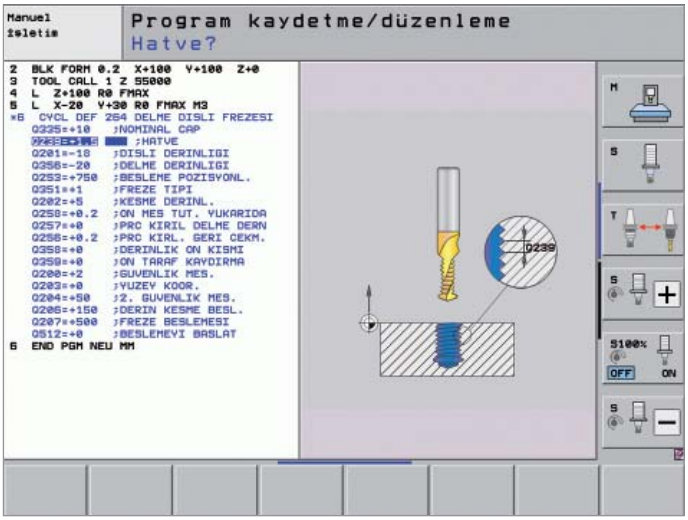
- English
- German
- Czech
- French
- Italian
- Spanish
- Portuguese
- Swedish
- Danish
- Finnish
- Dutch
- Polish
- Hungarian
- Russian
- Chinese (simplified)
- Chinese (traditional)

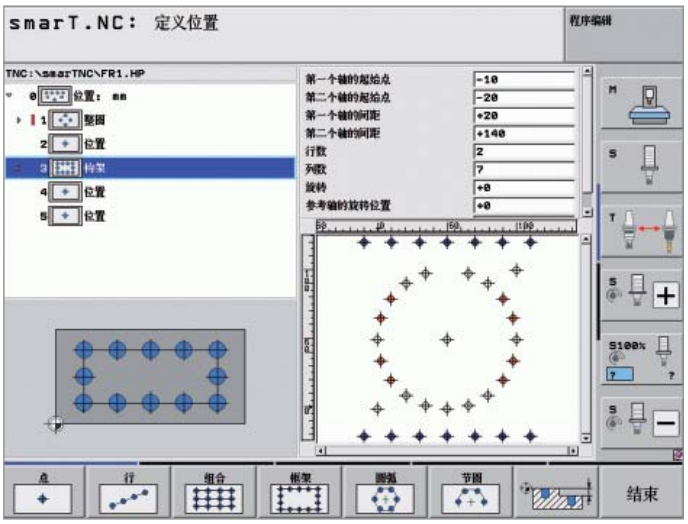
Beyond the above languages, you can also enable one of the following languages with the proper option:

- Slovenian
- Slovak
- Latvian
- Norwegian
- Korean
- Estonian
- Turkish
- Romanian
- Lithuanian

When you switch conversational languages the TNC changes the soft-key texts, dialog questions, fillable-form texts and error messages with description of cause and remedy. The machine tool builder, too, can provide machine-specific dialog texts in the defined conversational language.

The conversational language can be switched through a machine parameter that is accessible to the user. Any machine operator can therefore easily select a preferred language without having to restart the machine.





Additional conversational languages	Option 41	ID
Slovenian		530184-01
Slovak		530184-02
Latvian		530184-03
Norwegian		530184-04
Korean		530184-06
Estonian		530184-07
Turkish[1)]		530184-08
Romanian[1)]		530184-09
Lithuanian[1)]		530184-10
TNC 640 HSCI	As of NC SW 34059x-01	
TNC 620 HSCI	As of NC SW 34056x-02/73498x-01	
TNC 320	As of NC SW 34055x-04	
iTNC 530 HSCI	As of NC SW 60642x-01	
iTNC 530	As of NC SW 34049x-01	
	[1)] As of NC SW 34049x-05	
Installation by the user		
For more information –		

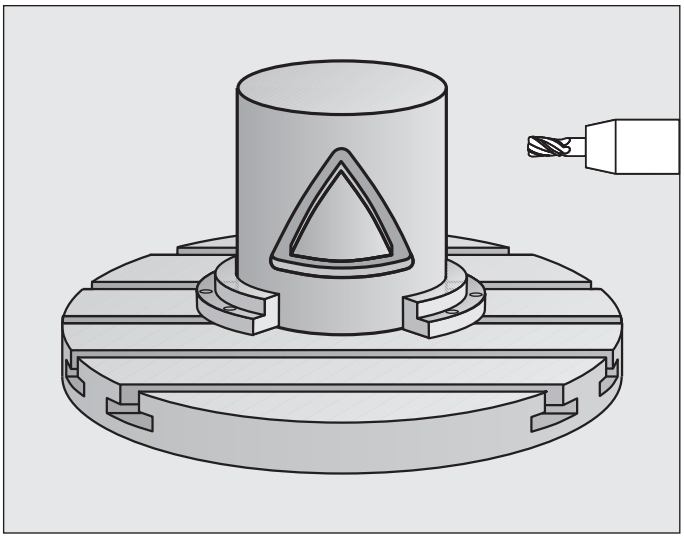
Programming and operation
Machining with a rotary table

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 as if the cylinder surface were unrolled. You enter a contour in two dimensions—as if in a plane—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 (only iTNC 530)

Linear feed rate for rotary tables in mm/min
In the standard version, the feed rate of rotary axes is programmed in degrees/minute. 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.



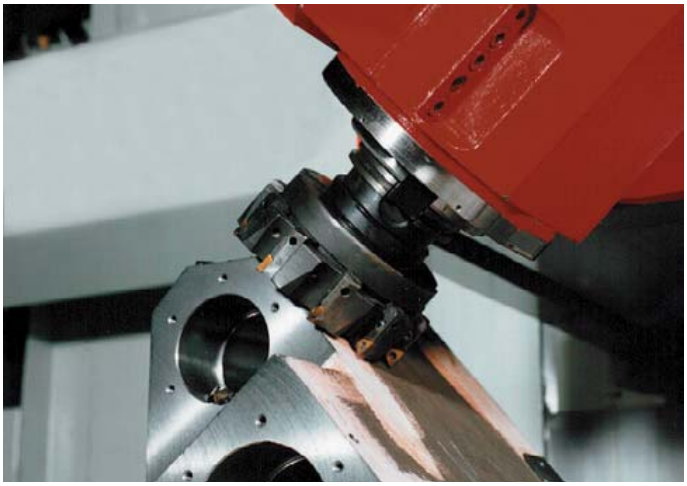
Machining with a rotary table	Option 8	ID 367591-01
TNC 640 HSCI	As of NC SW 34059x-01	
TNC 620 HSCI	As of NC SW 34056x-01/73498x-01	
TNC 320	As of NC SW 34055x-01	
iTNC 530 HSCI	As of NC SW 60642x-01	
iTNC 530	As of NC SW 34049x-01	
Installation by the machine tool builder		
For more information, see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

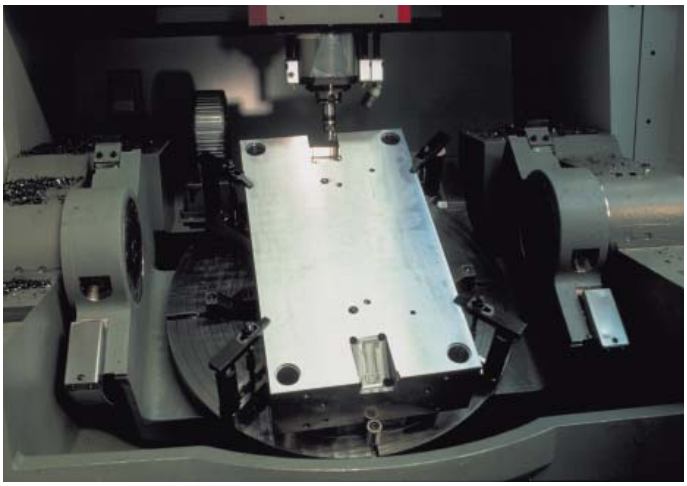
Programming and operation
Coordinate transformation – tilting the working plane, PLANE function

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 part as usual in the working plane (e.g. the X/Y plane), but it is machined in a plane that is rotated in one or more axes about 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 these complex functions as simple as possible, a separate animation is available for each possible plane definition, so that you can view them before selecting the function. Clearly arranged support graphics assist you during input.

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





Coordinate transformation	Option 8	ID 367591-01
TNC 640 HSCI	As of NC SW 34059x-01	
TNC 620 HSCI	As of NC SW 34056x-02/73498x-01	
TNC 320	As of NC SW 34055x-04	
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 the *iTNC 530 Presentation* CD		

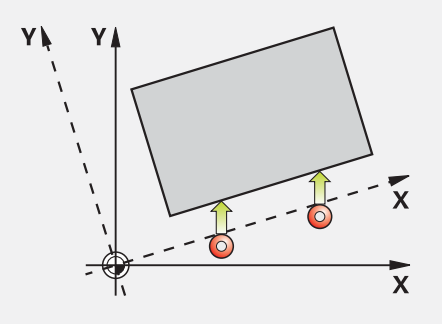
Programming and operation
Touch probe cycles

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.

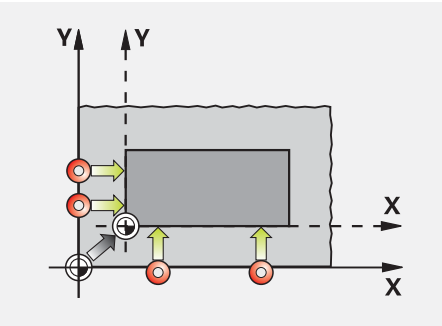
Datum setting
Finding this point quickly and reliably reduces nonproductive time and increases machining accuracy. The TNC features a large number of probing cycles for automatic presetting.

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.

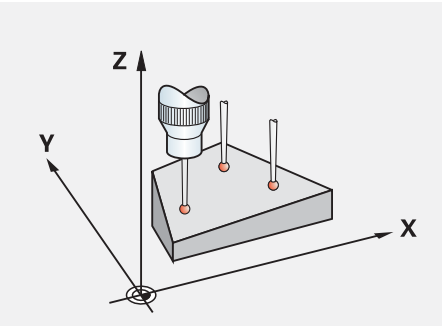
Tool measurement
Together with the TT or TL touch probes for tool measurement, the TNC makes it possible to measure tools automatically while they are in the machine spindle. The TNC saves the ascertained values of tool length and radius in the central tool file. By inspecting the tool during machining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



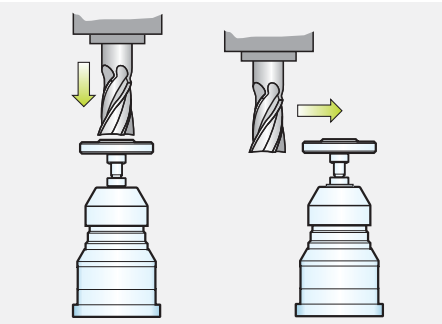
Workpiece alignment Compensate misalignment by rotating the coordinate system or turning the table



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	
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* CD		

Programming and operation

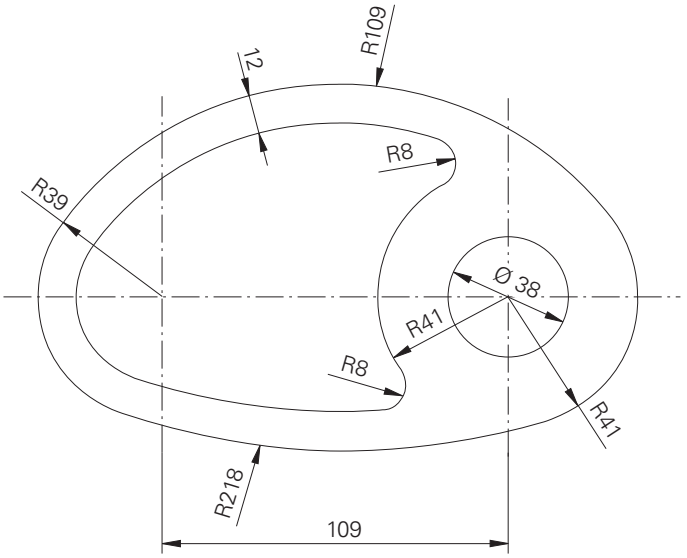
Advanced programming functions – FK free contour programming, fixed cycles

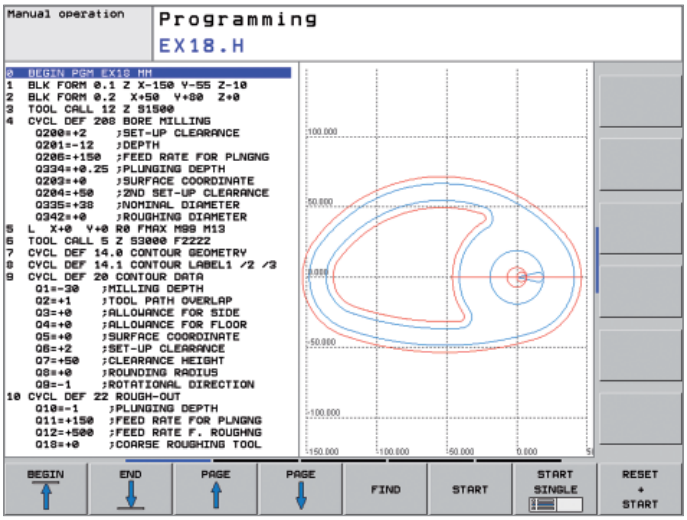
FK free contour programming
Not all workpieces are dimensioned for conventional NC programming. Thanks to FK, the control's free contour programming feature, in such cases you simply type in the data from the drawing—without first having to convert or calculate your data! It does not matter if individual 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
Clearing pockets with combined contours is aided greatly by Subcontour List cycles (SL). 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	
TNC 320	Standard	
iTNC 530 HSCI	Standard	
iTNC 530	Standard	
Installation by the user		
For more information, see the TNC brochures and the *iTNC 530 Presentation* CD		

Programming and operation
Program verification graphics, program-run graphics

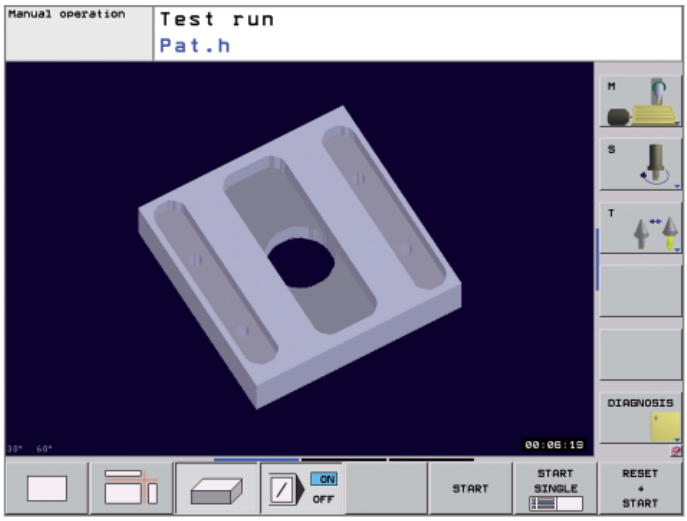
Besides its detailed programming graphics, HEIDENHAIN controls also support you with other graphical functions:

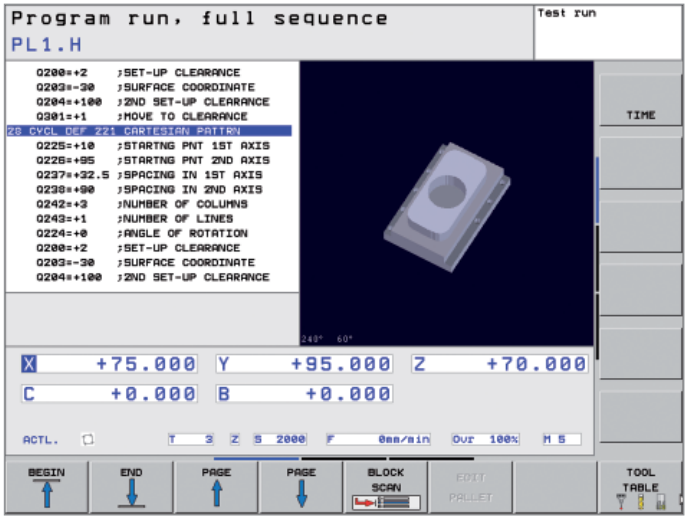
Program verification graphics
To be on the safe side before running a program, the TNC can graphically simulate the machining of the workpiece.

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
On the TNC, you can run the programming graphics or verification graphics even while the workpiece is being machined. Also, it shows a real-time graphic of the machining progress during program run. 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 22	ID 628253-01
TNC 640 HSCI	Standard	
TNC 620 HSCI	As of NC SW 34056x-01/73498x-01	
TNC 320	Standard	
iTNC 530 HSCI	Standard	
iTNC 530	Standard	
Installation by the user		
For more information, see the TNC brochures		

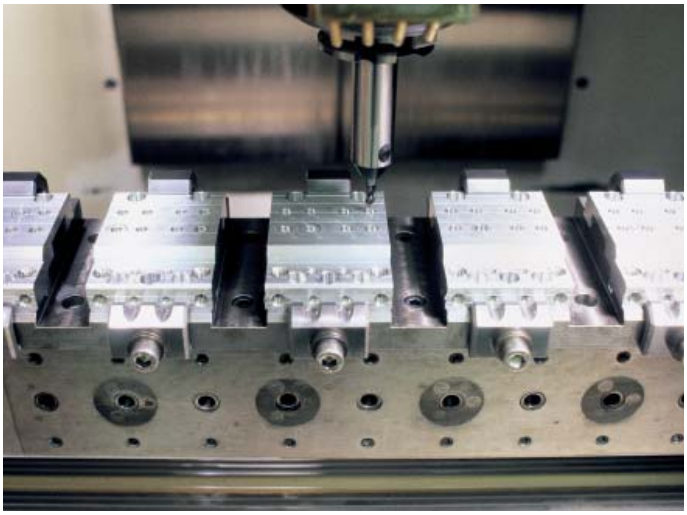
Programming and operation
Pallet management

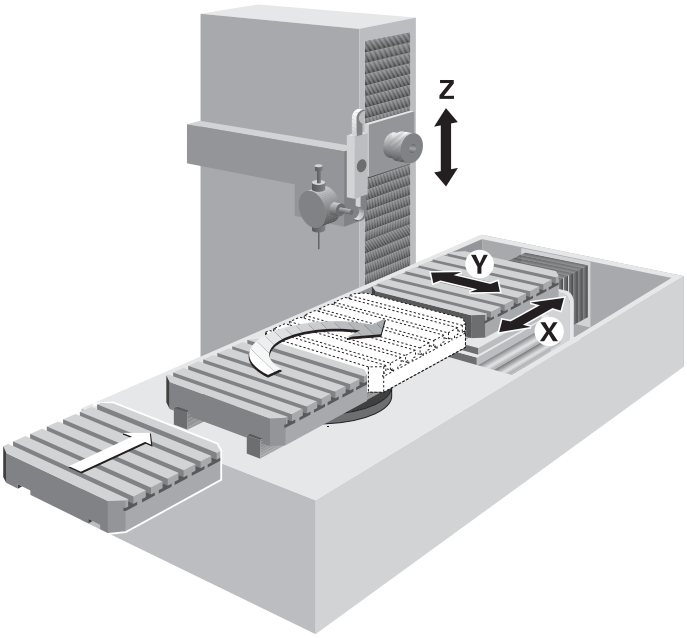
The TNC can assign the appropriate program and datum shift to parts mounted on pallets and brought to the machine in a random sequence.

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 oriented to the workpiece or the tool (only iTNC 530).





Pallet management	Option 22	ID 628253-01
TNC 640 HSCI	Standard	
TNC 620 HSCI	As of NC SW 34056x-01/73498x-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

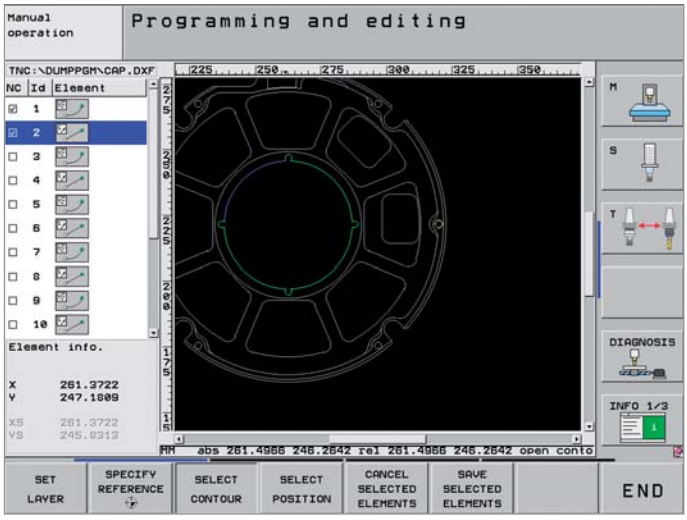
Why program contours when your drawing is already in DXF format anyway? You can open DXF files directly on the TNC in order to extract contours or machining positions from it. Not only does this save time otherwise spent on programming and testing, but you can also be sure that the finished contour is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

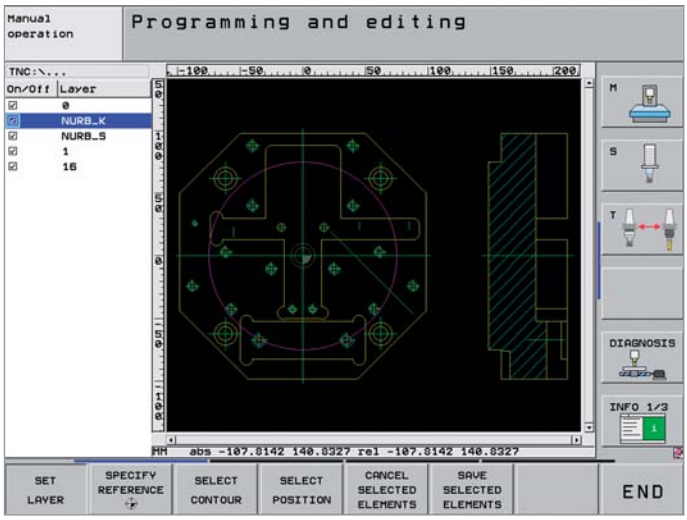
As a rule, DXF files contain multiple layers, with which the designer organizes a drawing. So that as little unnecessary information as possible appears on the screen during selection of the contours, you can hide via mouse click all **excessive layers** contained in the DXF file. This requires a keyboard 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 preset.** 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 34056x-04/73498x-02	
TNC 320	–	
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 the *iTNC 530 Presentation* CD		

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 plain 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 valuable 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 provided the software basis for the turning functions. They enable you to very easily program even complex turning operations at the machine.

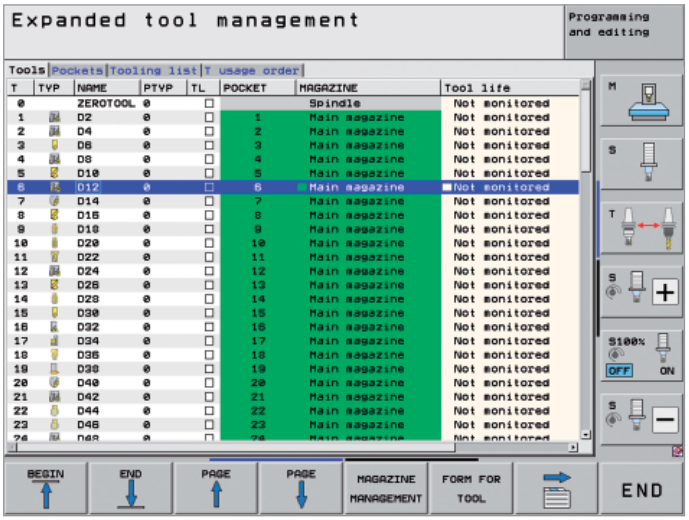
Turning functions	Option 96	ID 751653-01
TNC 640 HSCI	As of NC SW 34059x-01	
TNC 620 HSCI	–	
TNC 320	–	
iTNC 530 HSCI	–	
iTNC 530	–	
Installation by the machine tool builder		
For more information, see the brochure *TNC 640*		

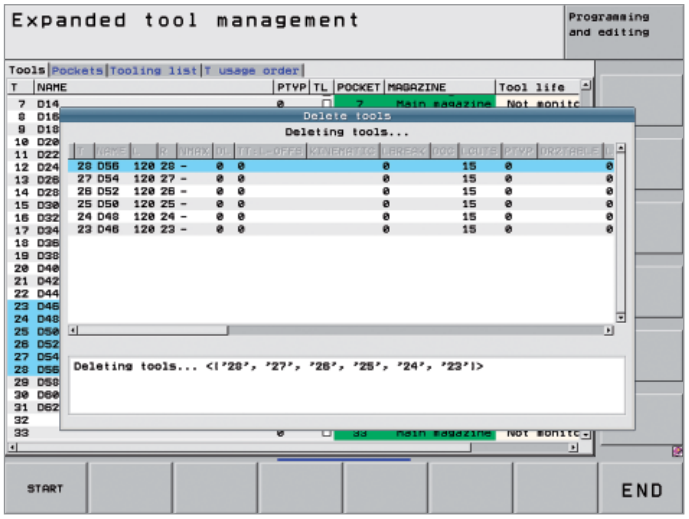
Programming and operation
Expanded tool management

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

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

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





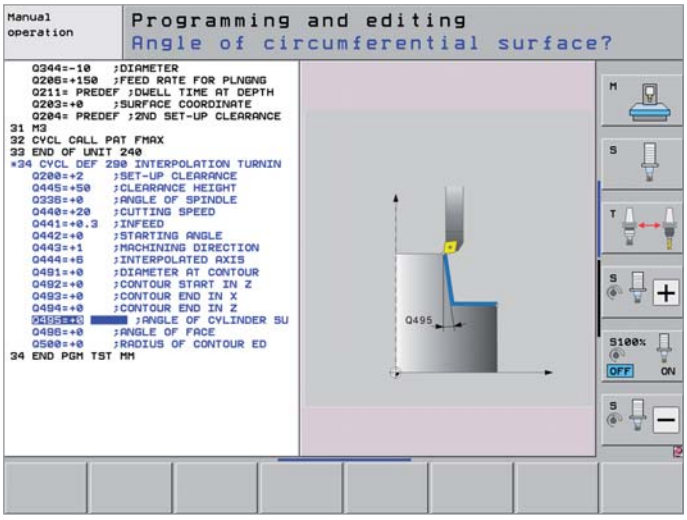
Expanded tool management	Option 93	ID 679938-01
TNC 640 HSCI	As of NC SW 34059x-01	
TNC 620 HSCI	–	
TNC 320	–	
iTNC 530 HSCI	As of NC SW 60642x-01	
iTNC 530	As of NC SW 34049x-05	
Installation by the machine tool builder		
For more information –		

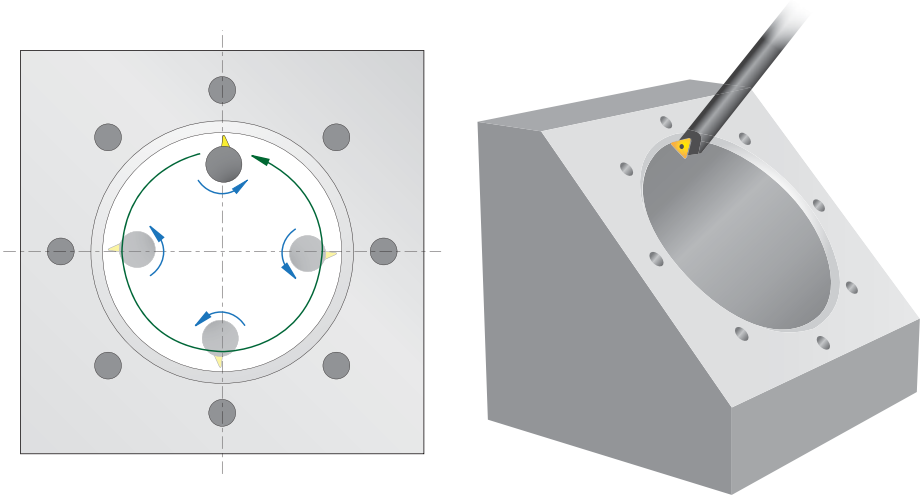
Programming and operation
Interpolating spindle – interpolation turning

In interpolation turning, the cutting edge of the tool moves on a circle, with the cutting edge always oriented to the center of the circle. 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, which is defined by the starting and end point, in the active working plane. 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 steps are not possible. The machining strategy can be chosen flexibly: from the outside in or vice versa, and also from top to bottom or vice versa. This results in four different machining strategies, which are distributed over the four quadrants.





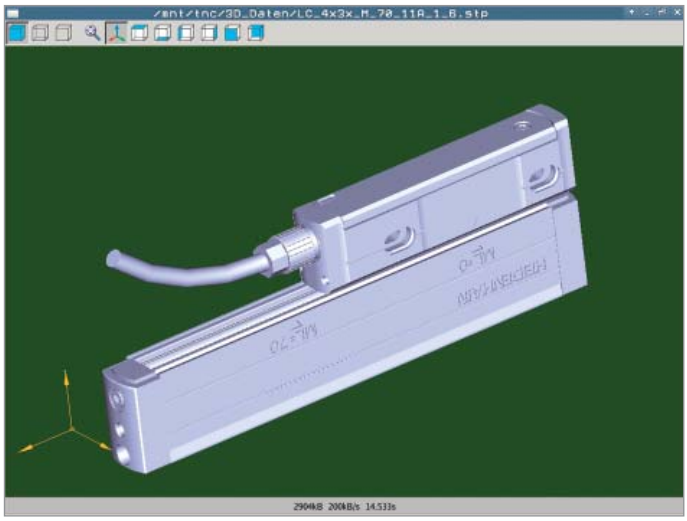
Interpolating spindle	Option 96	ID 751653-01
TNC 640 HSCI	–	
TNC 620 HSCI	–	
TNC 320	–	
iTNC 530 HSCI	As of NC SW 60642x-02	
iTNC 530	As of NC SW 34059x-07	
Installation by the machine tool builder		
For more information, see the brochure *iTNC 530*		

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 the problematic locations can be displayed appropriately.



CAD viewer	Option 98	ID 800553-01
TNC 640 HSCI	–	
TNC 620 HSCI	–	
TNC 320	–	
iTNC 530 HSCI	As of NC SW 60642x-02	
iTNC 530	As of NC SW 34059x-07	
Installation by the user		
For 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 new TNC function **KinematicsOpt** is an important component to help you meet these high requirements: With a HEIDENHAIN touch probe inserted, a 3-D 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.

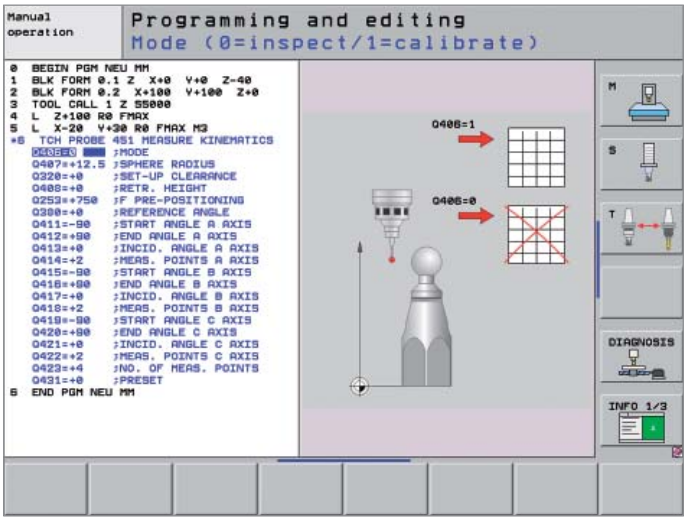
From the measured values, the TNC calculates the static tilting accuracy. The software minimizes the spatial error arising from the tilting movements and, at the end of the measurement process, automatically saves the machine geometry in the respective machine constants of the kinematics table.

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 655475-01
KKH 250	Height 250 mm	ID 655475-02



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	
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 brochure *KinematicsOpt*		

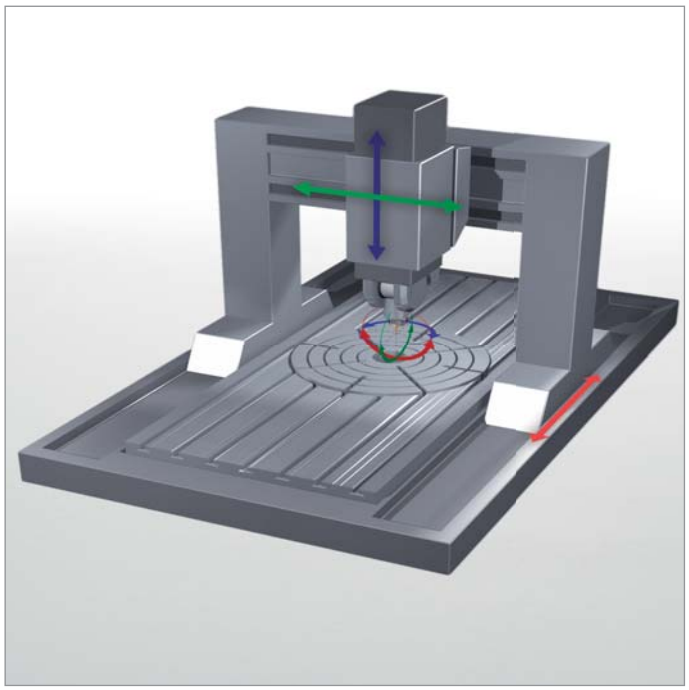
Machine accuracy
KinematicsComp – 3-D spatial compensation

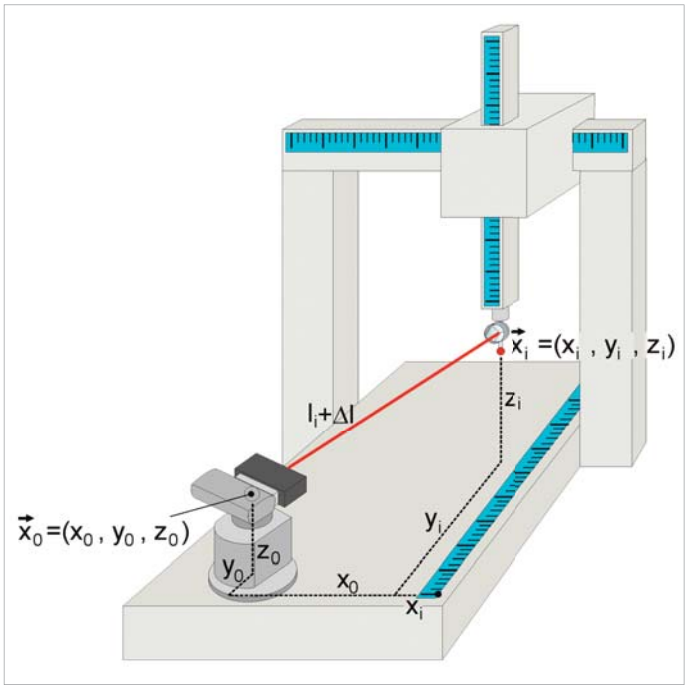
Narrow workpiece tolerances require high machine accuracy. However, machine tools inevitably have errors due to mounting or design.

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 six types of error, and a rotary axis can have eleven. The use of mechanical means to cope with these errors requires considerable effort. These errors become particularly evident on 5-axis machines or very large machines. Thermal expansion that can cause highly complex geometry changes of machine components cannot be disregarded either.

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 in such a way that the tool center point follows exactly the ideal nominal contour. Thermally induced errors are also measured and compensated via sensors and the PLC. For example, the spatial errors of the tool tip are measured with a laser tracer or laser interferometer.

The KinematicsComp option cannot be enabled for the export versions.





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

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

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. When machining with five axes (three linear axes and two tilting axes), the tool can stay perpendicular to the workpiece surface, or if desired, inclined at a predetermined angle.

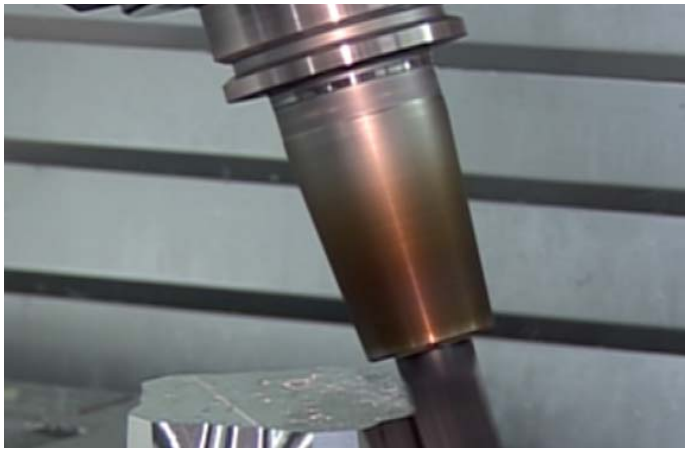
Regardless of what type of 5-axis programs you wish to run, the TNC makes all the compensating movements in the linear axes that result from movements in the tilting axes. The TNC's Tool Center Point Management feature (TCPM)—an improvement upon the proven TNC function M128—provides optimal tool guidance and prevents contour gouging.

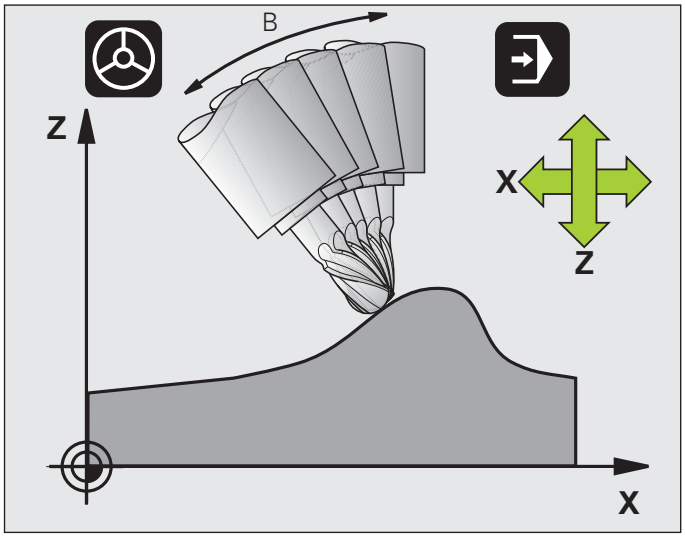
5-axis simultaneous machining	Option 9	ID 367590-01
TNC 640 HSCI	As of NC SW 34059x-01	
TNC 620 HSCI	As of NC SW 34056x-01/73498x-01	
TNC 320	–	
iTNC 530 HSCI	As of NC SW 60642x-01	
iTNC 530	As of NC SW 34049x-01	
Installation by the machine tool builder		
For more information, see the TNC brochures		

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 the change inclination angles of rotary axes in externally written programs that would 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 with without having to change the NC program.

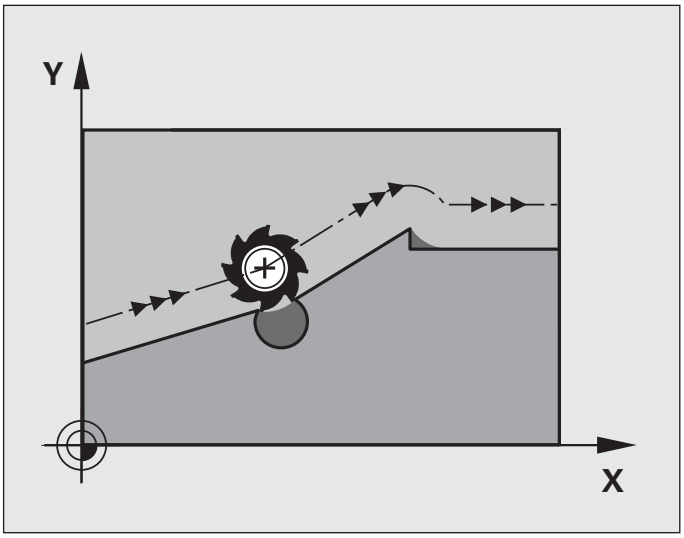




Handwheel Superimposition	Option 21	ID 628254-01
TNC 640 HSCI	Standard	
TNC 620 HSCI	As of NC SW 34056x-01/73498x-01	
TNC 320	–	
iTNC 530 HSCI	Standard	
iTNC 530	Standard	
Installation by the machine tool builder		
For more information, see the TNC brochures		

Machining functions
Tool compensation – radius-compensated contour 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 provide NC programs with tool radius compensation 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	
TNC 320	Standard	
iTNC 530 HSCI	Standard	
iTNC 530	Standard	
Installation by the machine tool builder		
For more information, see the TNC brochures		

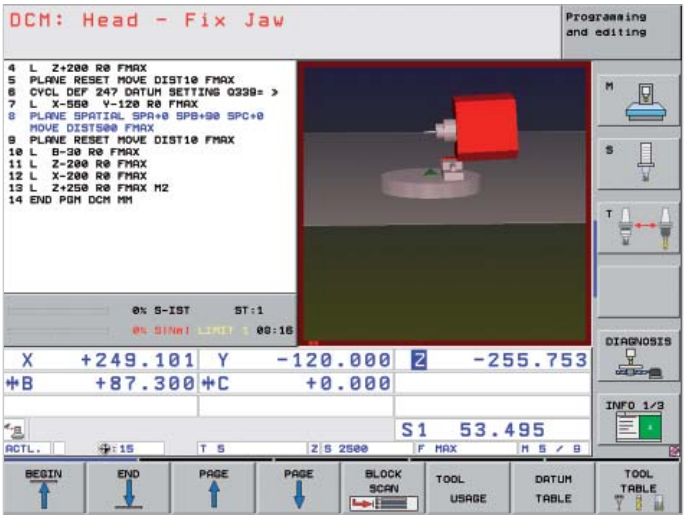
Machining functions
DCM – dynamic collision monitoring

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 **dynamic collision monitoring (DCM)** feature 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 down-times. Unattended shifts become safer and more reliable.

However, DCM works not only in **automatic mode.** It is also active in **manual operation.** 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.



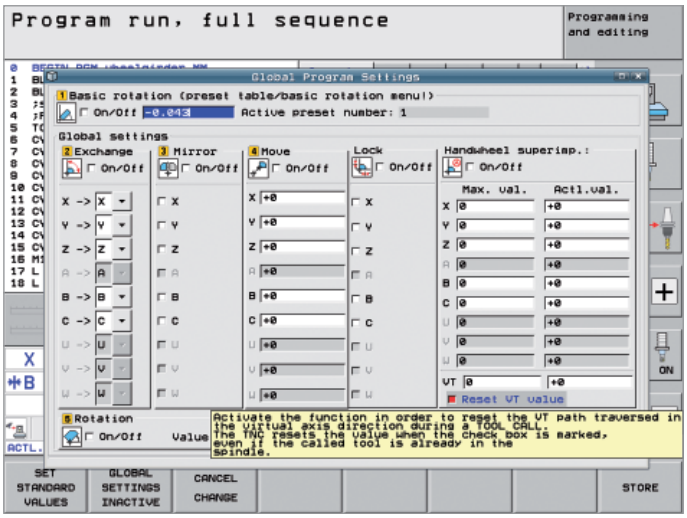
DCM	(option 40)	ID 526452-01
TNC 640 HSCI	As of NC SW 34059x-02	
TNC 620 HSCI	–	
TNC 320	–	
iTNC 530 HSCI	As of NC SW 60642x-01	
iTNC 530	As of NC SW 34049x-02	
Installation by the machine tool builder		
For more information, see the brochure *DCM*		

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. It allows 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:
- Exchanging 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		
For more information –		

Machining functions
AFC – adaptive feed control

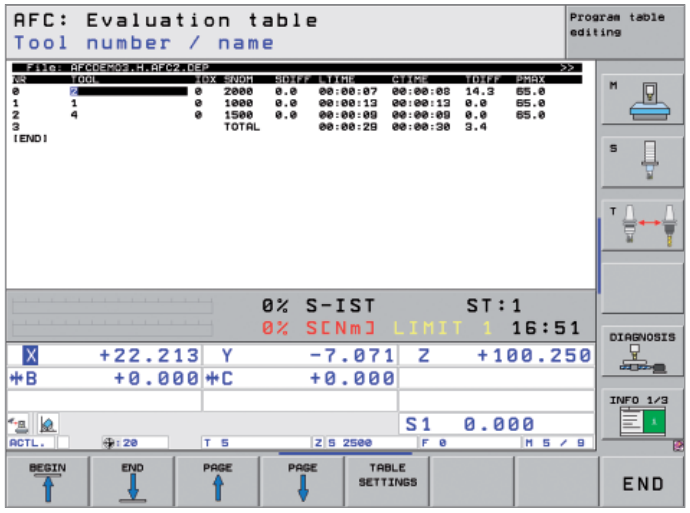
Adaptive feed rate control (AFC) automatically regulates the feed rate of the TNC, taking into consideration the respective spindle power and further process data. In a teach-in cut, the TNC records the maximum spindle power. Then, before actual machining, you define in a table the respective limit values between which the TNC can influence the feed rate in the adaptive control mode 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 rate 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. The total machining time is shortened by an increased feed rate in the machining zones with less stock removal.

Tool monitoring
The adaptive feed rate 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 down to the reference value whenever the learned maximum permissible spindle power is exceeded also reduces the strain and wear on the machine. It effectively protects the spindle from overload.



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 –		

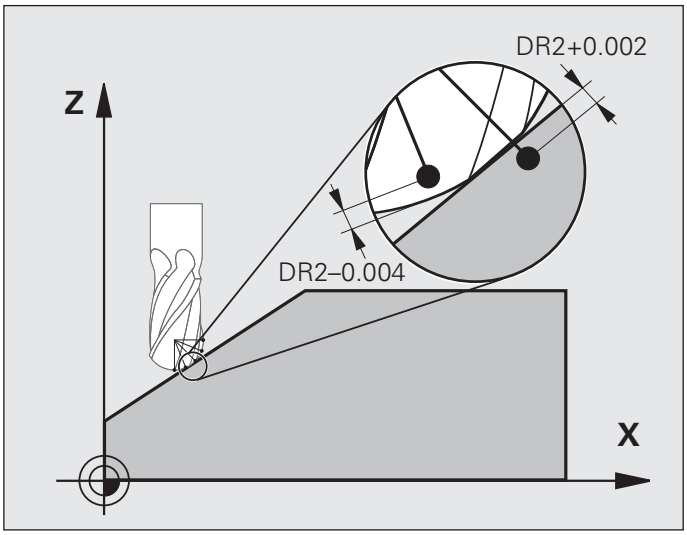
Machining functions

3D-ToolComp – 3-D radius compensation depending on the tool's contact 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 shape (see figure).

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 deviations of the tool used are available as a calibration chart from the tool manufacturer, then you can create the compensation-value table manually.



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

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 increasingly complex 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, for example:

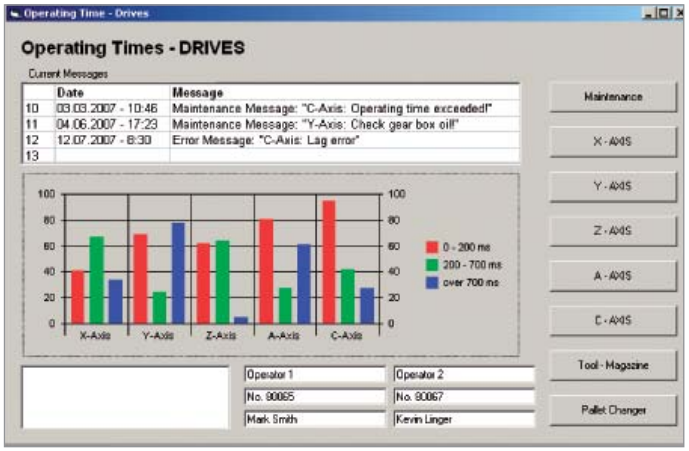
- Error reporting systems that, for example, send the customer a text message to his cell phone reporting problems on the currently running machining process
- Standard or customer-specific PC software that decidedly improves process security and equipment availability
- Software solutions controlling the processes of manufacturing systems
- Information exchange with job management software

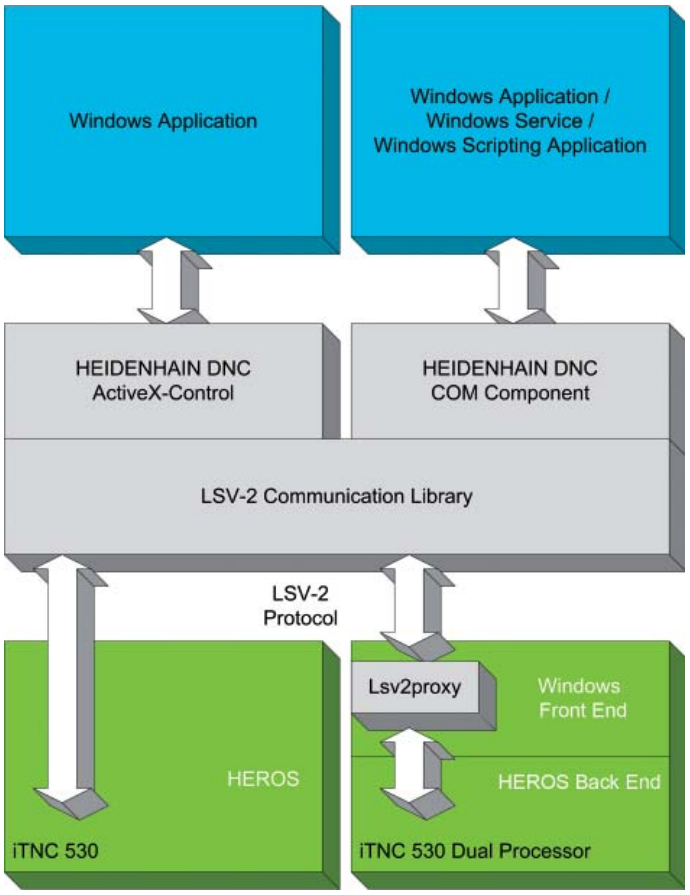
The **HEIDENHAIN DNC** software interface is an attractive communication platform for this purpose. It provides all the data and configuration capabilities needed for these processes so that an external PC application can evaluate data from the control and, if required, influence the manufacturing process.

RemoTools SDK development package
To enable you to use the HEIDENHAIN DNC software interface, installation of the **RemoTools SDK** development package provides COM software components and ActiveX control for development environments on Windows operating systems.

These components are independent of the programming language, and are based on Microsoft's (D)COM technology. Both software components can be used with any modern development environment for Windows XP/Vista/7.

During installation of RemoTools SDK, the software components are registered in the operating system and are thereby available to the development environments.





HEIDENHAIN DNC	Option 18	ID 526451-01
RemoTools SDK	Accessories	ID 340442-xx
TNC 640 HSCI	As of NC SW 34059x-01	
TNC 620 HSCI	As of NC SW 34056x-01/73498x-01	
TNC 320	As of NC SW 34055x-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 brochure *HEIDENHAIN DNC*		

Communication

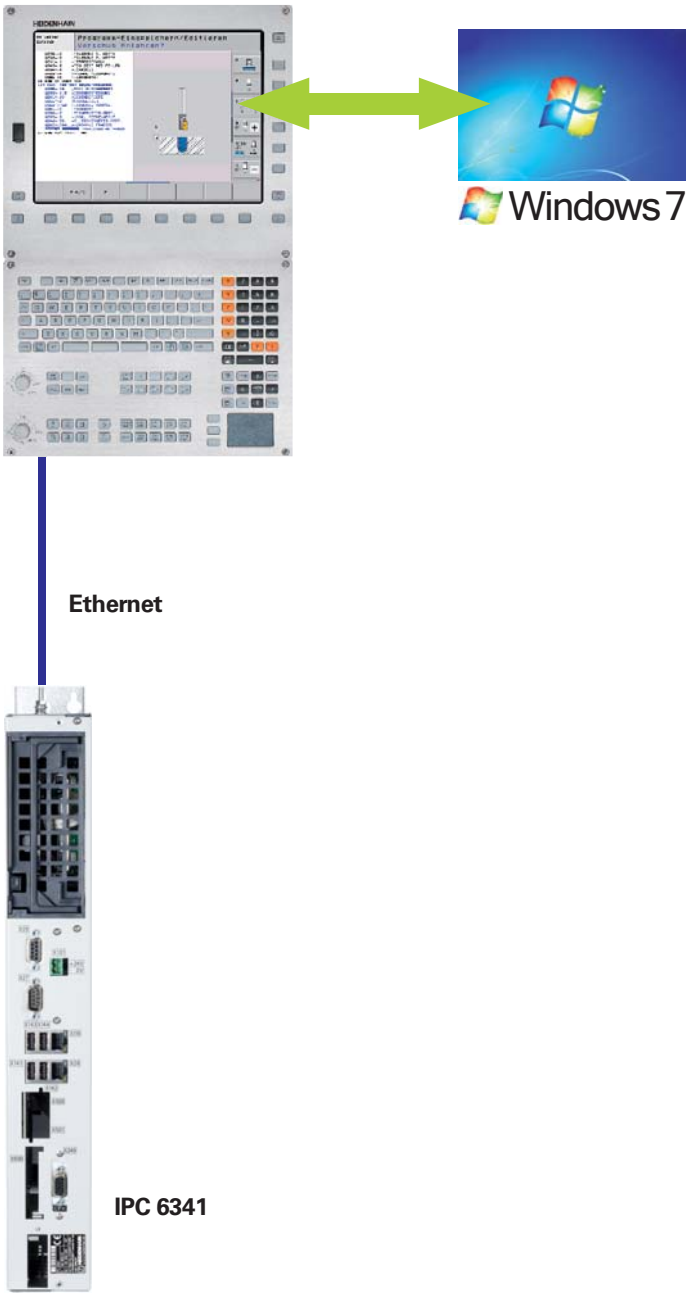
Remote Desktop Manager – display and remote operation of external computer units

In daily operations it can often be necessary to make entries in PC diagnostic or planning and control systems or using Windows-based software. The Remote Desktop Manager option provides the user with the opportunity to operate one or more 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 6341) 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 become 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.



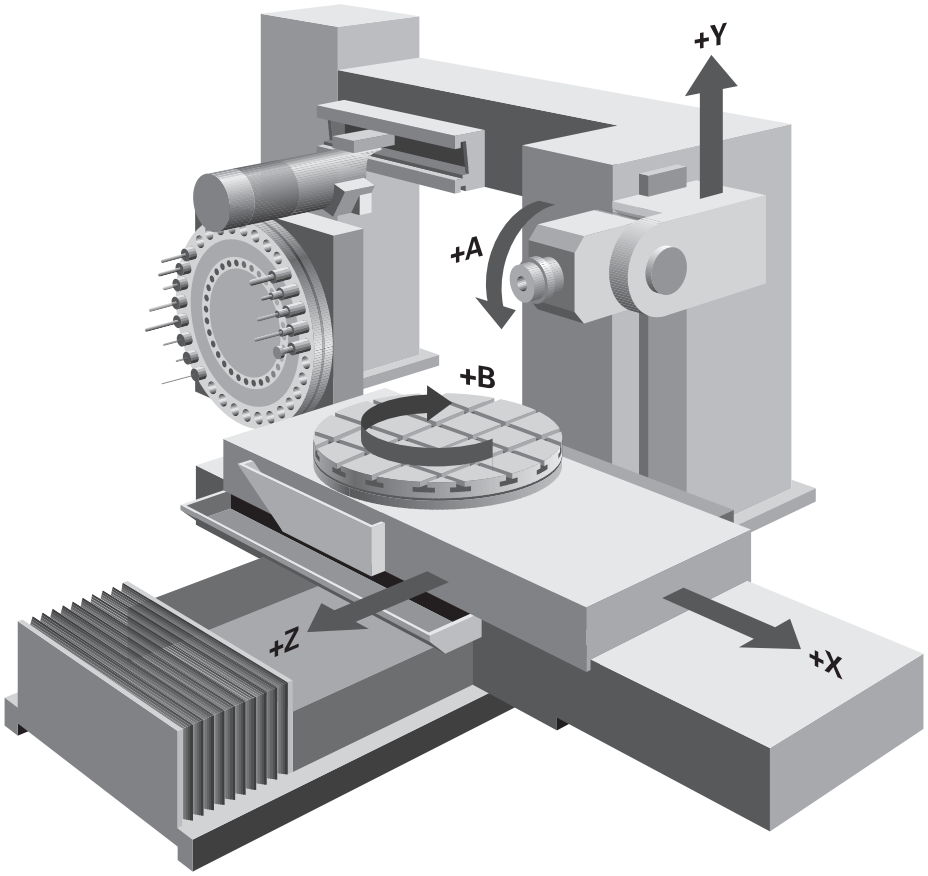
Remote Desktop Manager	Option 133	ID 894423-01
TNC 640 HSCI	As of NC SW 34059x-01	
TNC 620 HSCI	–	
TNC 320	–	
iTNC 530 HSCI	As of NC SW 60642x-02	
iTNC 530	–	
Installation by IT specialists		
For more information, see the Technical Manuals		

Interfacing to the machine
Additional control loops

The number of enabled control loops depends on the inserted SIK, or on additionally enabled control loops that can also be ordered as needed at a later date.

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	
TNC 320	As of NC SW 34055x-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 catalogs *Information for the Machine Tool Builder*

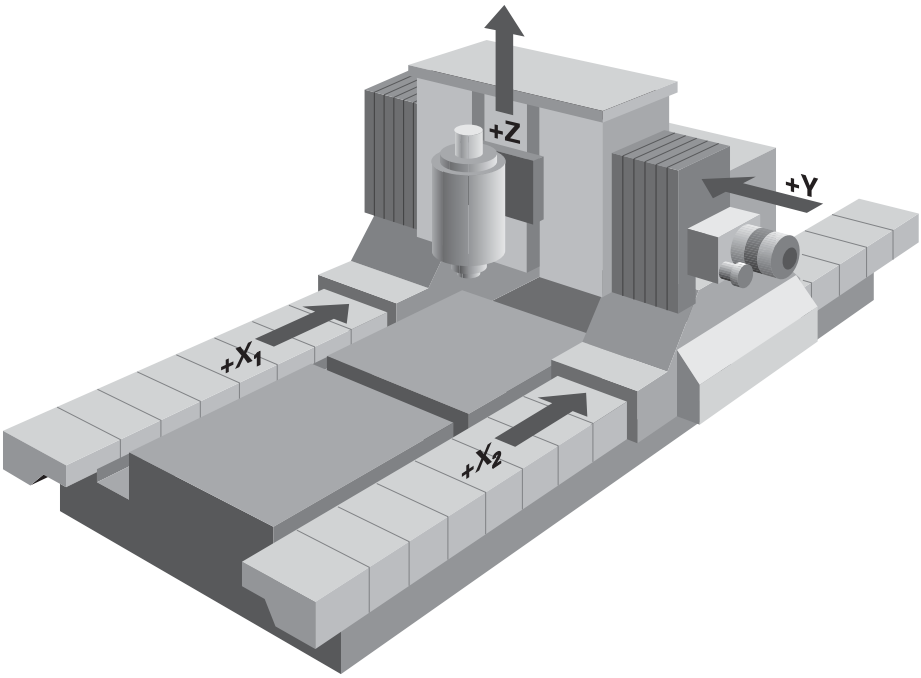
Interfacing to the machine

Synchronized axes – gantry axes, tandem tables

Synchronous 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.



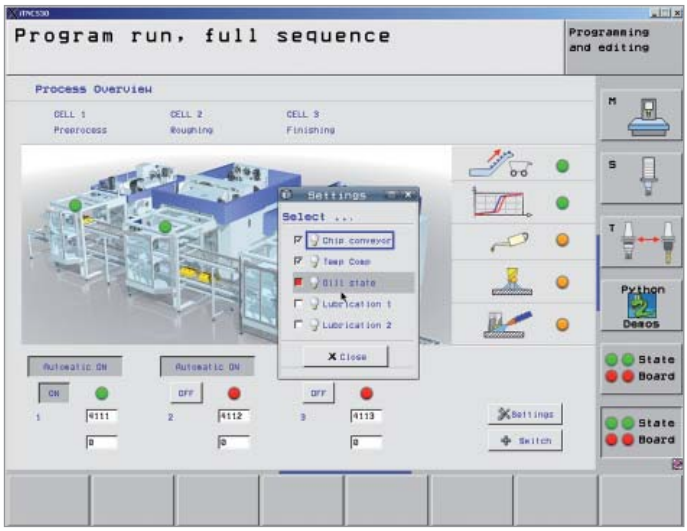
Synchronized axes	Option 24	ID 634621-01
TNC 640 HSCI	Standard	
TNC 620 HSCI	As of NC SW 34056x-01/73498x-01	
TNC 320	As of NC SW 34055x-01	
iTNC 530 HSCI	Standard	
iTNC 530	Standard	
Installation by the machine tool builder		
For more information, see the Technical Manuals		

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 (PLC). Python is an easy-to-learn script language that supports the use of all necessary high-level language elements.

Python OEM Process can be universally used for machine functions and complex calculations, as well as to display special user interfaces. User-specific or machine-specific solutions can be efficiently implemented. A large number of functions are available 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.

The applications created can be included via the PLC in the familiar PLC windows, or they can be displayed in separate windows that are freely integrated in the TNC user interface up to the size of the entire TNC screen.

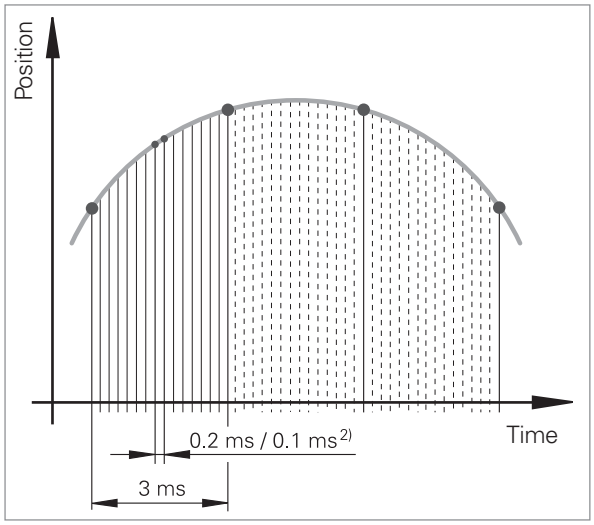


Python OEM process	Option 46	ID 579650-01
TNC 640 HSCI	As of NC SW 34059x-01	
TNC 620 HSCI	As of NC SW 34056x-01/73498x-01	
TNC 320	As of NC SW 34055x-04	
iTNC 530 HSCI	As of NC SW 60642x-01	
iTNC 530	As of NC SW 34049x-04	
Installation by IT specialists		
For more information, see the Technical Manuals		

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}	TINT
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	
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 catalogs *Information for the Machine Tool Builder*		

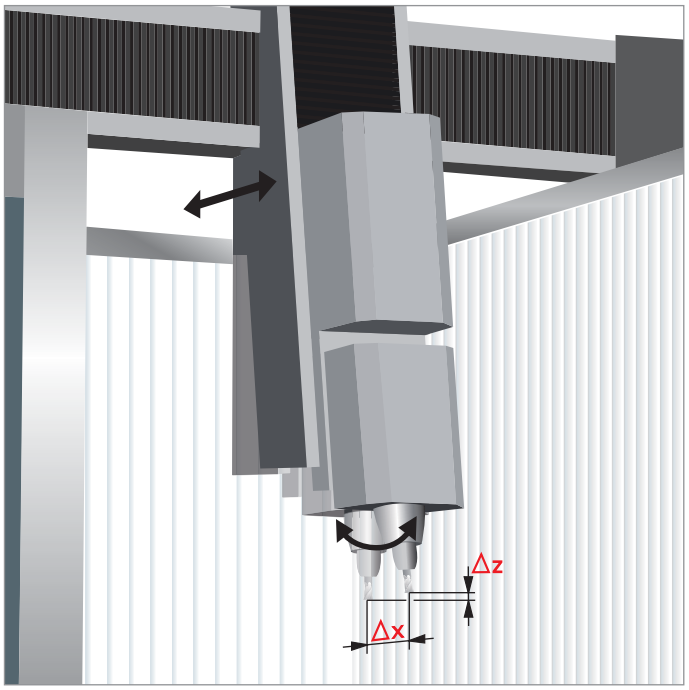
Interfacing to the machine

CTC – compensation of position errors through axis coupling

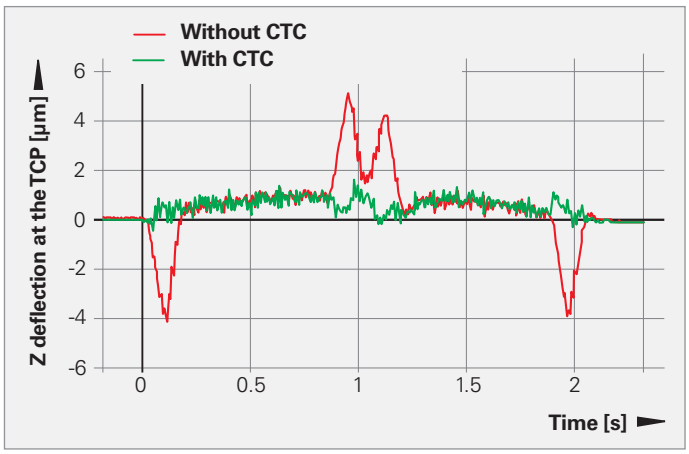
Dynamic acceleration processes introduce forces to the structure of a machine tool. These forces can briefly deform parts of the machine and thereby lead to deviations at the tool center point (TCP). Besides deformation in axis direction, the dynamic acceleration of an axis due to mechanical axis coupling can also result in deformation of axes that are lateral 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 CTC servo-control option (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 fixed with 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 servo control option CTC.



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



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

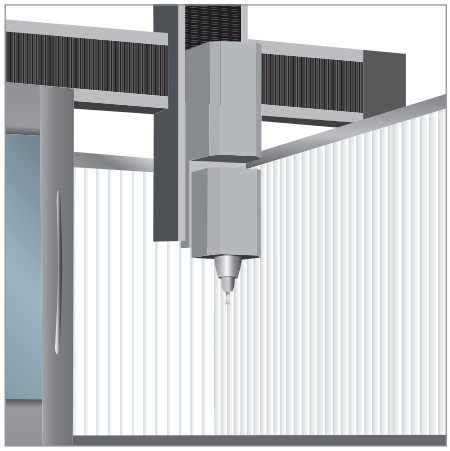
CTC	Option 141	ID 800542-01
TNC 640 HSCI	As of NC SW 34059x-02	
TNC 620 HSCI	As of NC SW 34056x-04/73498x-02	
TNC 320	–	
iTNC 530 HSCI	As of NC SW 60642x-02	
iTNC 530	–	
Installation by the machine tool builder		
For more information –		

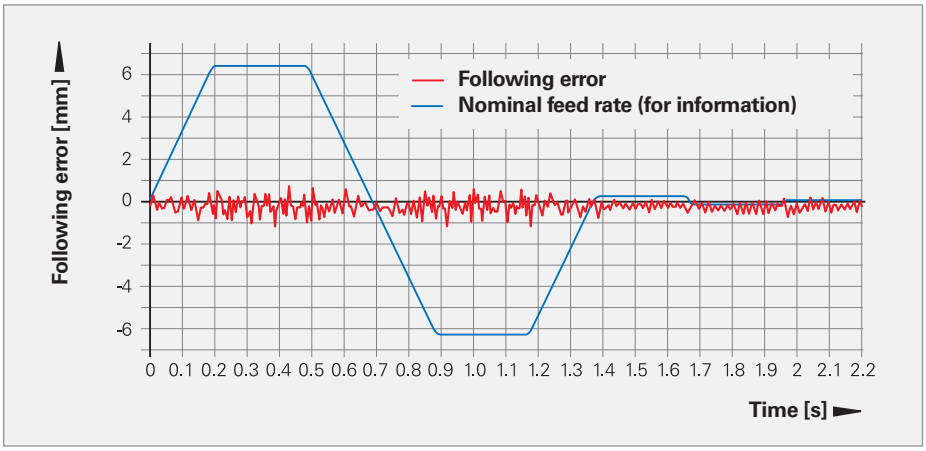
Interfacing to the machine
PAC – position-dependent adaptation of controller parameters

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.

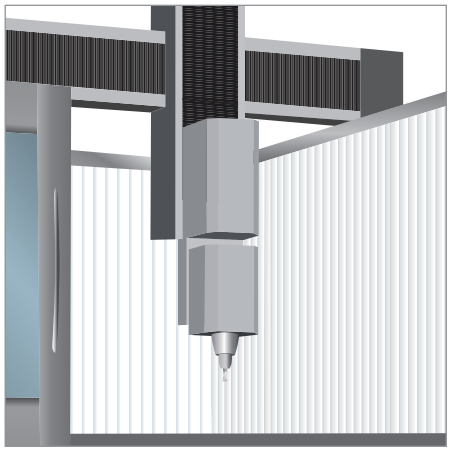
To exploit the machine's dynamic possibilities, you can use the PAC option (Position Adaptive Control) to change machine parameters depending on position.

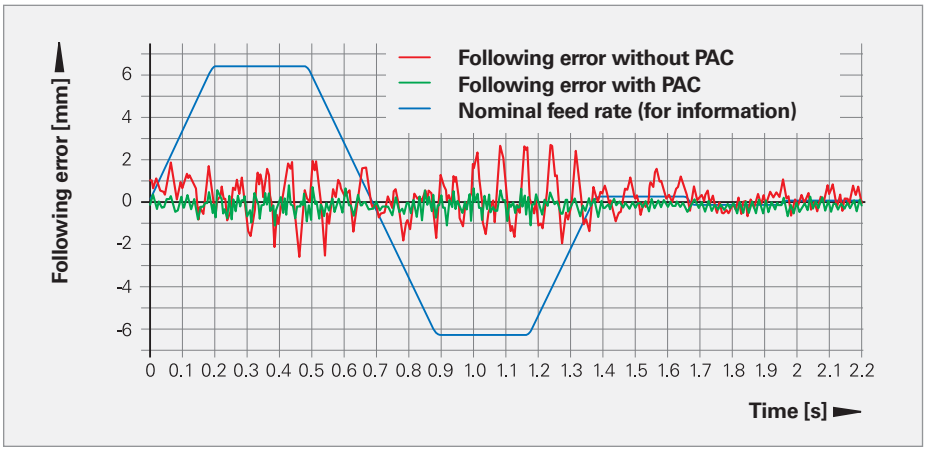
This makes it possible to assign respectively optimal loop gain to defined support points. Additional position-dependent filter parameters can be defined in order to further increase control loop stability.





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





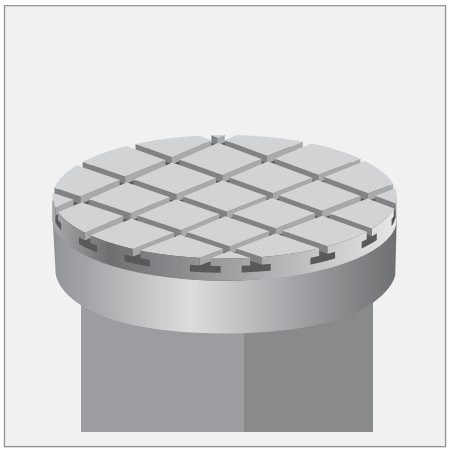
Servo control at Z = -500
- Without PAC: Clearly visible oscillations and following error outside of the tolerance band (± 3 µm)
- With active PAC: Following error stays within the tolerance band (± 1 µm)

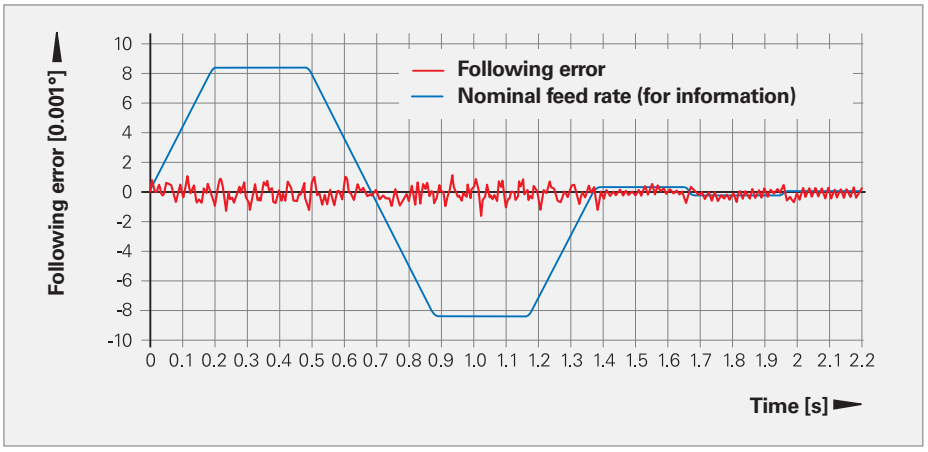
PAC	Option 142	ID 800544-01
TNC 640 HSCI	As of NC SW 34059x-02	
TNC 620 HSCI	As of NC SW 34056x-04/73498x-02	
TNC 320	–	
iTNC 530 HSCI	As of NC SW 60642x-02	
iTNC 530	–	
Installation by the machine tool builder		
For more information –		

Interfacing to the machine
LAC – load-dependent adjustment of controller parameters

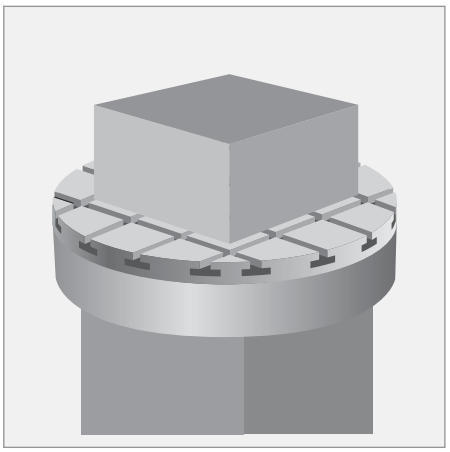
The dynamic behavior of machines with moving tables can vary depending on the mass or mass moment of inertia of the fixed workpiece.

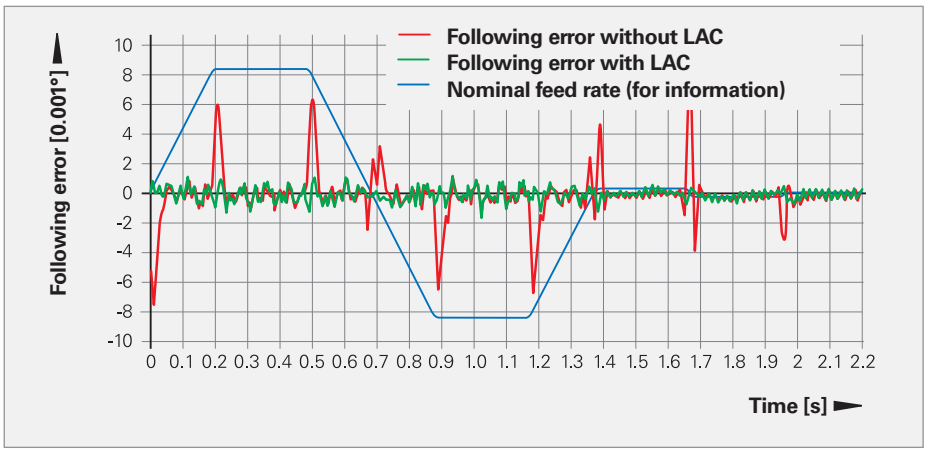
The LAC option (Load Adaptive Control) enables the control to automatically ascertain the workpiece's current mass, mass moment of inertia and the friction forces. In order to optimize changed control behavior at differing loads, adaptive feedforward controls can exploit data on acceleration, holding torque, static friction and friction at high shaft speeds. During workpiece machining, the control can also continuously adjust the parameters of the adaptive feedforward control to the current mass of the workpiece.





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 in active in the feedforward control, the following error is within the tolerance band (± 0.001°)

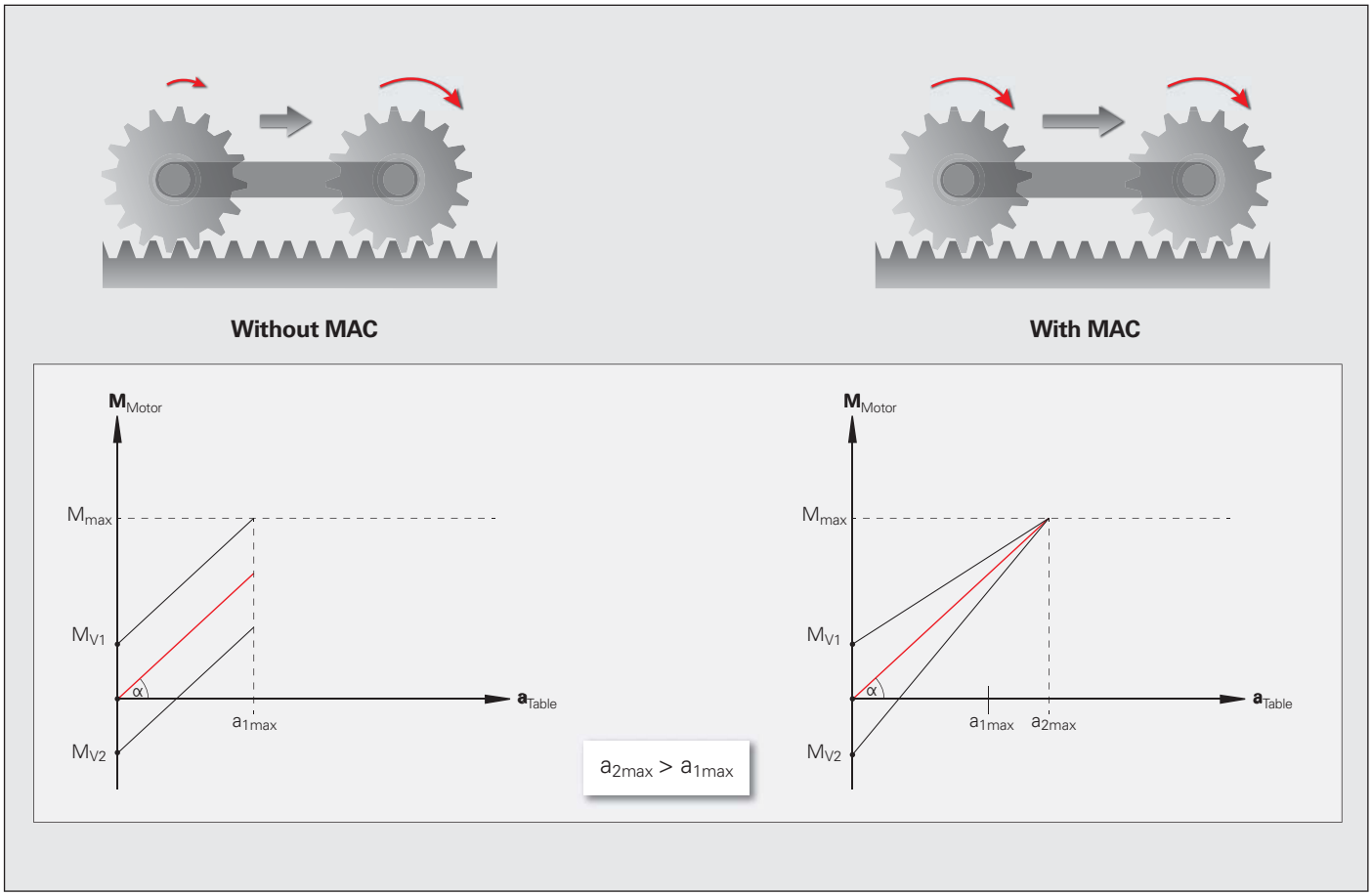
LAC	Option 143	ID 800545-01
TNC 640 HSCI	As of NC SW 34059x-02	
TNC 620 HSCI	As of NC SW 34056x-04/73498x-02	
TNC 320	–	
iTNC 530 HSCI	As of NC SW 60642x-02	
iTNC 530	–	
Installation by the machine tool builder		
For more information –		

Interfacing to the machine
MAC – motion-dependent adaptation of control parameters

In addition to the position-dependent adjustment of machine 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 drive. Through this motion-dependent adaptation of the control parameters it is possible, for example, to realize a velocity-dependent adaptation of the control loop gain 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, a significantly higher maximum acceleration can be attained with this arrangement during rapid traverse, for example through parameterized reduction of the tensioning torque with increasing acceleration.



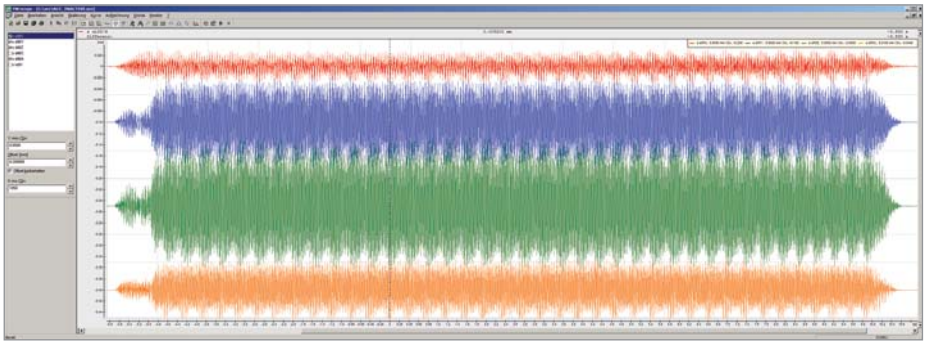
MAC	Option 144	ID 800546-01
TNC 640 HSCI	As of NC SW 34059x-02	
TNC 620 HSCI	As of NC SW 34056x-04/73498x-02	
TNC 320	–	
iTNC 530 HSCI	As of NC SW 60642x-03	
iTNC 530	–	
Installation by the machine tool builder		
For more information –		

Interfacing to the machine
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. This chattering causes ugly marks on the workpiece surface. The tool, too, is subject to heavy and irregular wear from chattering. In extreme cases it can even break the tool during operation.

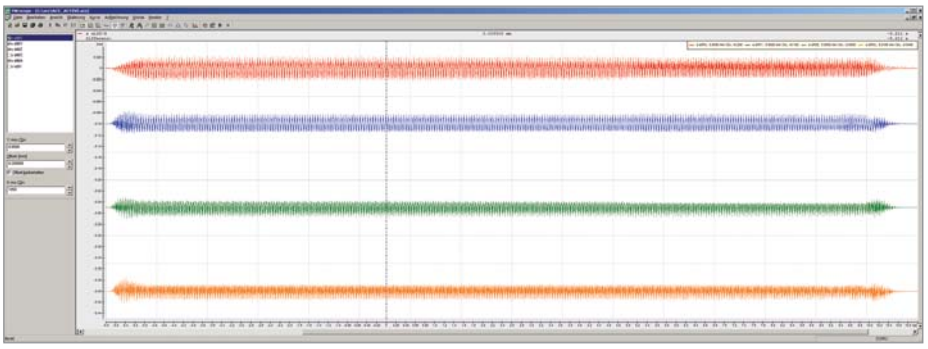
To reduce the inclination to chattering, HEIDENHAIN now offers an effective antidote with the Active Chatter Control (ACC) control function. The use of this control function is particularly advantageous during heavy cutting. ACC makes substantially higher metal removal rates possible. This makes it possible 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

ACC	Option 145	ID 800547-01
TNC 640 HSCI	As of NC SW 34059x-02	
TNC 620 HSCI	As of NC SW 73498x-02	
TNC 320	–	
iTNC 530 HSCI	As of NC SW 60642x-03	
iTNC 530	–	
Installation by the machine tool builder		
For more information –		

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 as the machine tool builder's purposes.

The option range 101 – 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 on-site support by the machine tool builder.

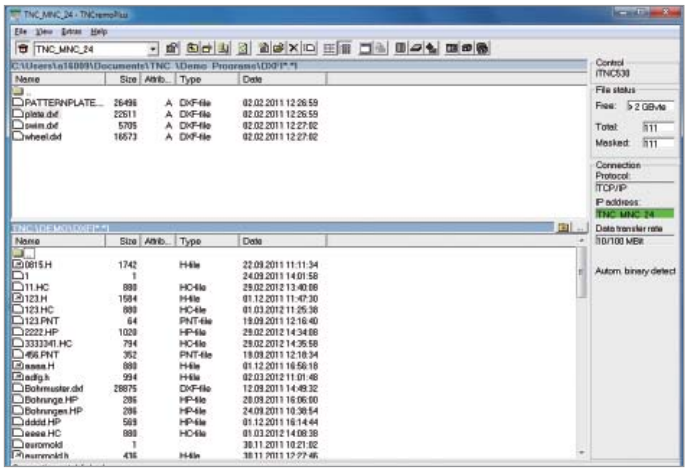
OEM option	Options 101 to 130	ID 579651-01 to ID 579651-30
TNC 640 HSCI	–	
TNC 620 HSCI	–	
TNC 320	–	
iTNC 530 HSCI	As of NC SW 60642x-01	
iTNC 530	As of NC SW 34049x-06	
Installation by the machine tool builder		
For more information –		

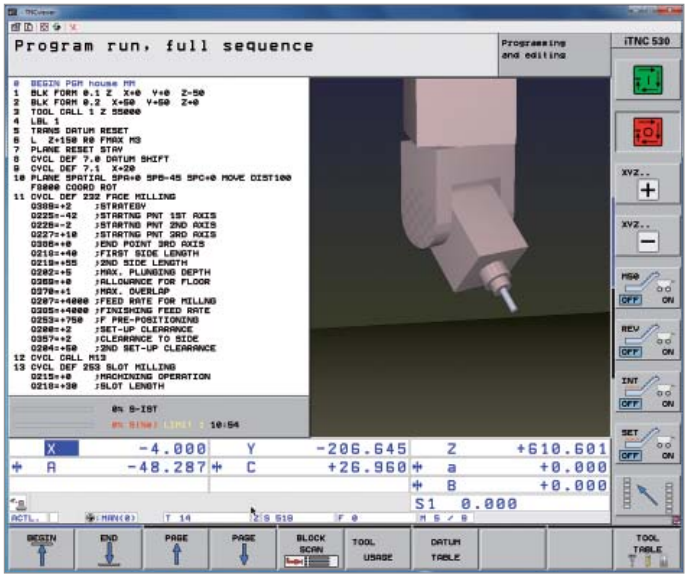
PC software
TNCremo – programs for data transfer

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 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
TNC 320	As of NC SW 34055x-01
iTNC 530 HSCI	As of NC SW 60642x-01
iTNC 530	As of NC SW 34049x-01
Installation by the user	
For more information –	

PC software
TeleService – remote diagnostics for HEIDENHAIN controls

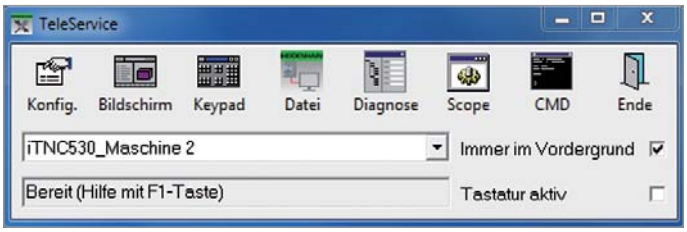
The PC software **TeleService** permits comprehensive remote diagnostics and extensive remote operation of HEIDENHAIN controls. This makes in-depth troubleshooting possible. The service technician communicates online with the control over modem, ISDN or over the Internet, analyzes the control and repairs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

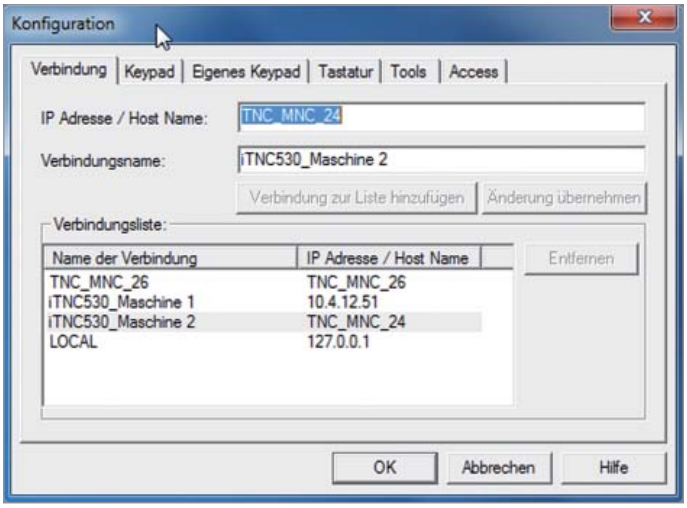
The machine tool builder creates the necessary diagnostic user interface according to its service requirements for checking the desired information. It is operated through the TeleService control panel.

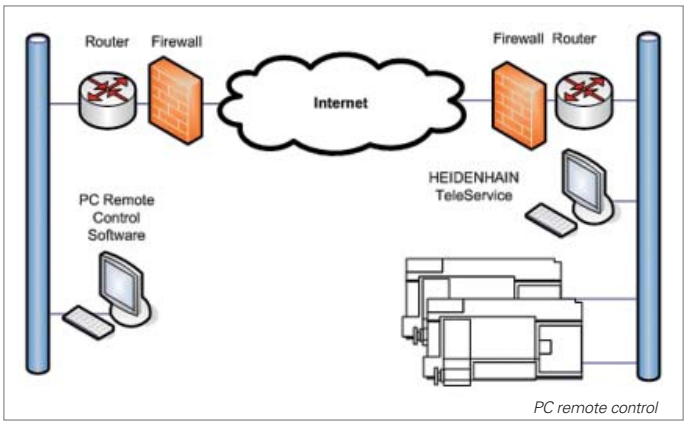
Functions
- Remote operation of the TNC with on-line screen transfer and virtual TNC keyboard
- Transmission of machining and PLC programs, machine parameters, tool and datum tables, etc.
- Display of machine and PLC data through TNCscope or TNCexplorer. The data are adapted to TNCexplorer by the machine 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 function library LSV2-Tool.

TeleService by the machine tool builder
The machine tool builder builds a network of its machines with TNCs at the customer's site and a network of service PCs in its 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.

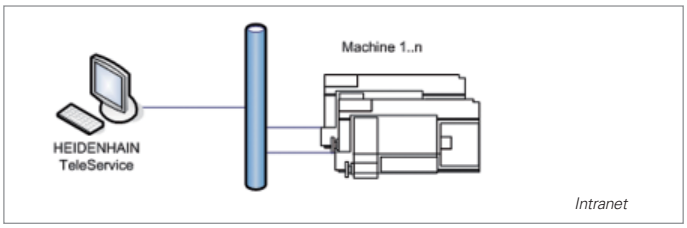
TeleService by the customer
TeleService can also be implemented in the customer's intranet. Here, a PC with TeleService installed is connected directly (without a router) to the network of TNCs. This makes remote operation, remote monitoring and remote diagnostics possible of the machines in the customer's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







Remote connection over PC remote control software



TeleService in the company network

TeleService	Download
TNC 640 HSCI	As of NC SW 34059x-01
TNC 620 HSCI	As of NC SW 34056x-01/73498x-01
TNC 320	As of NC SW 34055x-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 further information, see Product Information *Diagnostics for HEIDENHAIN Controls*	

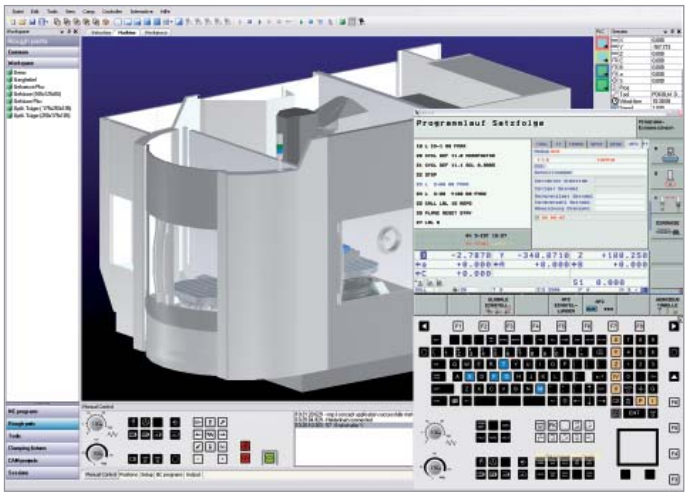
PC software
virtualTNC – control of virtual machines

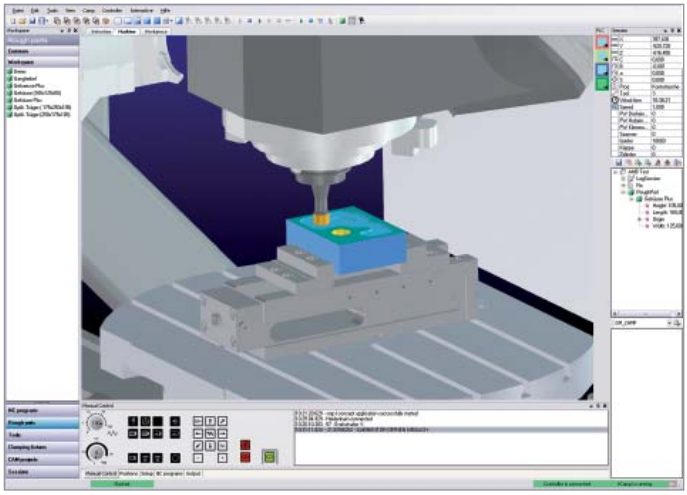
Simulation of NC program run on the control has been a feature of HEIDENHAIN contouring controls for some time now. The PC software "virtualTNC" now makes it possible to use the TNC as control component for machine-simulation applications (virtual machines) on external computer systems.

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 1.2** software development package and its help system.





virtualTNC	ID 584421-01
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 for manufacturers of machine-simulation applications	
For more information, see the brochure *HEIDENHAIN DNC*	

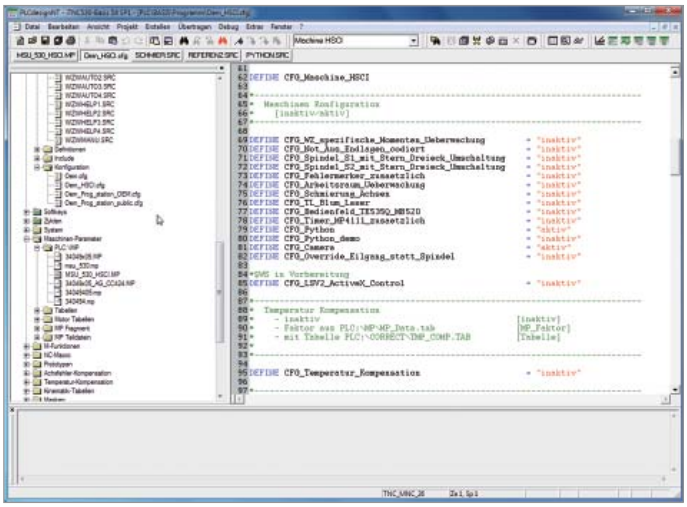
PC software

PLCdesign – software for PLC program development

PLCdesign 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
TNC 320	As of NC SW 34055x-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 catalogs *Information for the Machine Tool Builder*	

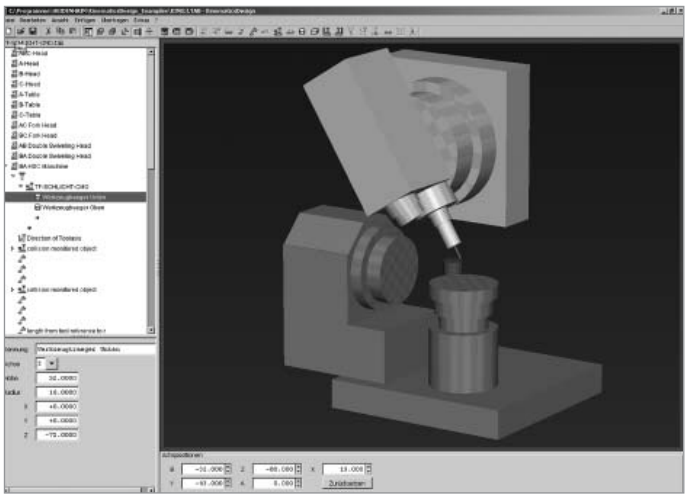
PC software
KinematicsDesign – for creating machine kinematic models

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 or the working space can be simulated when the axes are moved and DCM is active. 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 working envelope.



KinematicsDesign	ID 340448-07
TNC 640 HSCI	As of NC SW 34059x-01
TNC 620 HSCI	As of NC SW 34056x-01/73498x-01
TNC 320	As of NC SW 34055x-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 catalogs *Information for the Machine Tool Builder*	

PC software

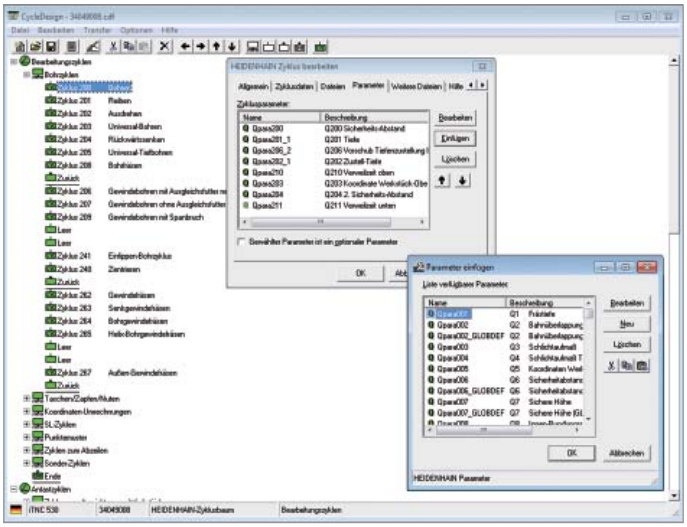
CycleDesign – saving NC subprograms as cycles

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 transfer the cycle data to the control's hard disk with CycleDesign.

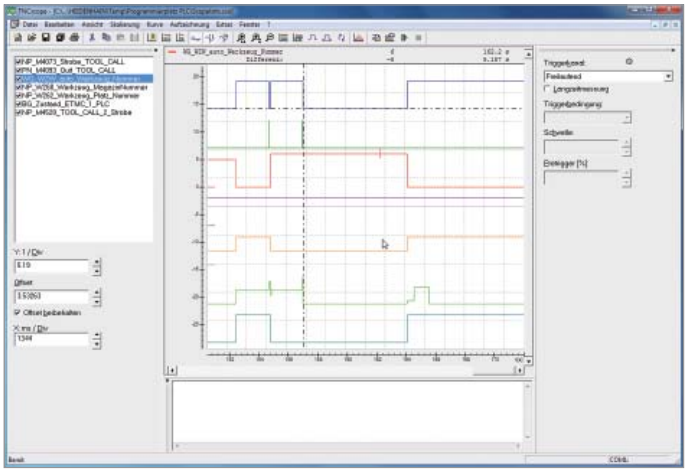


CycleDesign	Free download
TNC 640 HSCI	As of NC SW 34059x-01
TNC 620 HSCI	As of NC SW 34056x-01/73498x-01
TNC 320	As of NC SW 34055x-01
iTNC 530 HSCI	As of NC SW 60642x-01
iTNC 530	As of NC SW 34049x-01
Installation for users and machine manufacturers	
For more information, see the catalogs *Information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

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 graph
- 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)

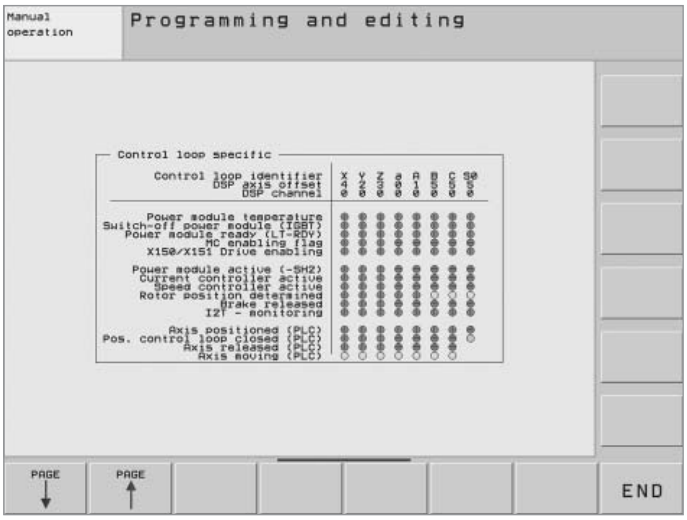
TNCscope	Download
TNC 640 HSCI	As of NC SW 34059x-01
TNC 620 HSCI	As of NC SW 34056x-01/73498x-01
TNC 320	As of NC SW 34055x-01
iTNC 530 HSCI	As of NC SW 60642x-01
iTNC 530	As of NC SW 34049x-01
Installation for machine manufacturers and service	
For more information, see the Technical Manuals, integrated 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 can be called directly from the TNC via the Diagnosis soft key. It is also available for downloading as PC software (accessory) from the HEIDENHAIN Filebase on the Internet. 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
TNC 320	–
iTNC 530 HSCI	As of NC SW 60642x-01
iTNC 530	As of NC SW 34049x-01

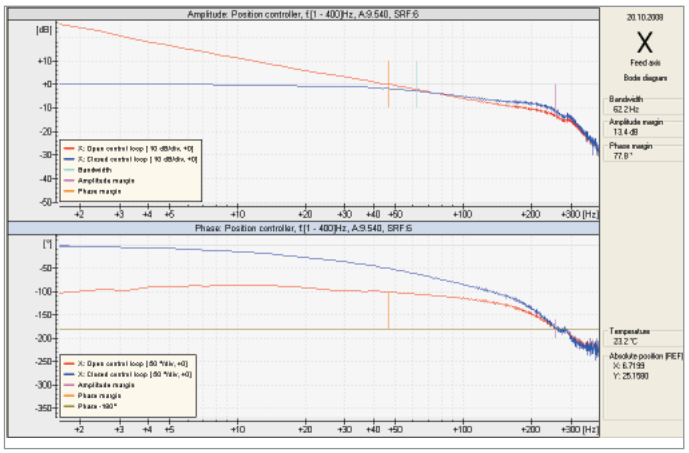
Installation for machine manufacturers and service

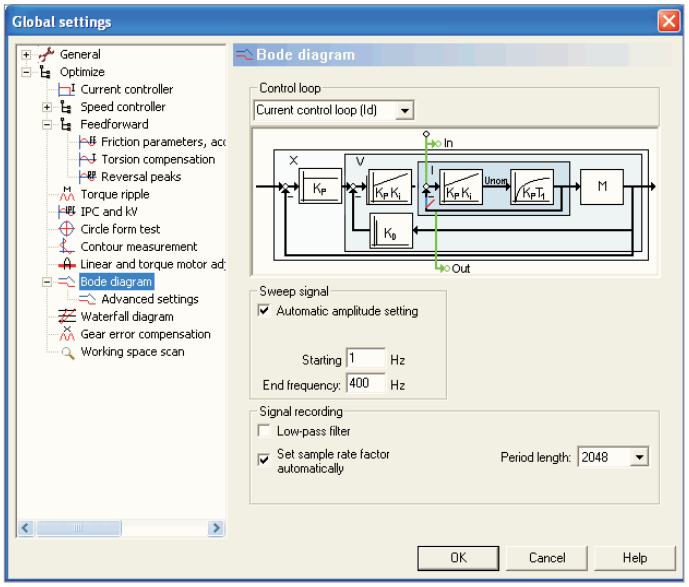
For further information, see Product Information *Diagnostics for HEIDENHAIN Controls*

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 setting in the control's servo loop particularly important. TNCopt helps you to easily maintain an overview and comply with the correct sequence during commissioning of all axes.

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





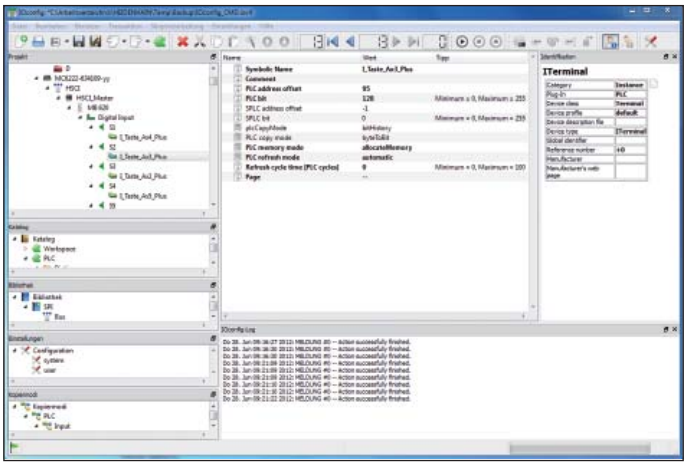
TNCopt	Download
TNC 640 HSCI	As of NC SW 34059x-01
TNC 620 HSCI	As of NC SW 34056x-01/73498x-01
TNC 320	–
iTNC 530 HSCI	As of NC SW 60642x-01
iTNC 530	As of NC SW 34049x-01
Installation for machine manufacturers and service	
For more information, see the catalogs *Information for the Machine Tool Builder*	

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 total project and forms an essential component in the HSCI system in the commissioning of HSCI components and SPI additional modules.

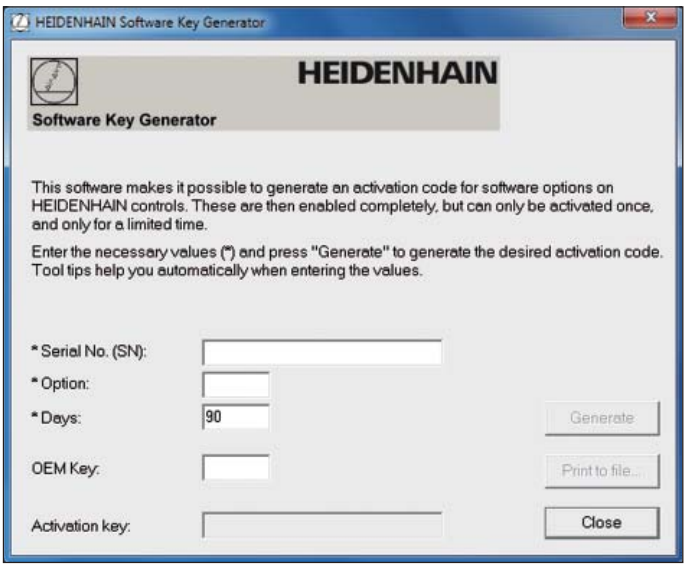


IOconfig	Download
TNC 640 HSCI	As of NC SW 34059x-01
TNC 620 HSCI	As of NC SW 34056x-01/73498x-01
TNC 320	–
iTNC 530 HSCI	As of NC SW 60642x-01
iTNC 530	As of NC SW 34049x-01
Installation for machine manufacturers and service	
For more information, see the Technical Manuals, integrated help system	

PC software
Software key generator – enabling key for software options

TNC controls from HEIDENHAIN offer numerous functions that are specially tailored to specific customer requirements. Temporary use of optional control functions (e.g. DXF converter) can be activated with the PC Software Key Generator in order to support the customer in his selection of available control options. This allows you to use an optional function for up to 90 days without obligation to buy. If the test was successful, you can choose to purchase the option from HEIDENHAIN.

This PC software key generator makes it possible to generate an activation code 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 activation code by entering the SIK number, the option to be enabled, the duration and a manufacturer-specific password.



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
TNC 320	As of NC SW 34055x-05
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 catalogs *Information for the Machine Tool Builder*

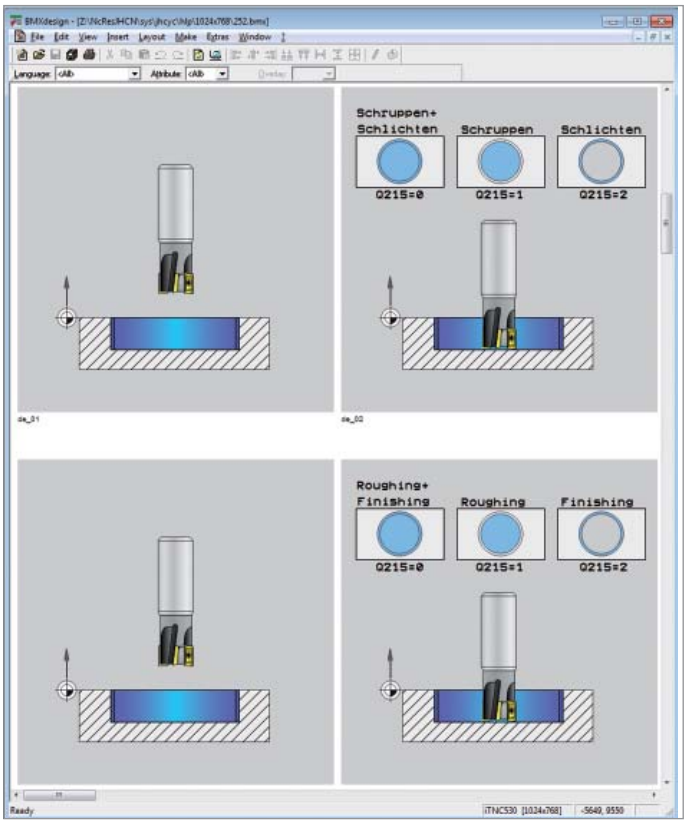
PC software
BMXdesign

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
- Function for exporting as bitmap file (*.BMP)
- Integration of PLCtext for managing BMX files



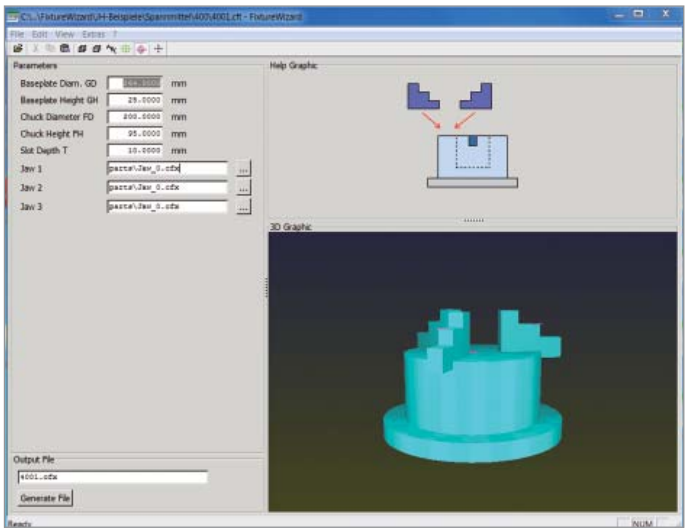
BMXdesign	ID 340443-xx
TNC 640 HSCI	As of NC SW 34059x-01
TNC 620 HSCI	As of NC SW 34056x-01/73498x-01
TNC 320	As of NC SW 34055x-01
iTNC 530 HSCI	As of NC SW 60642x-01
iTNC 530	As of NC SW 34049x-01
Installation for the machine tool builder	
For more information, see the catalogs *Information for the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

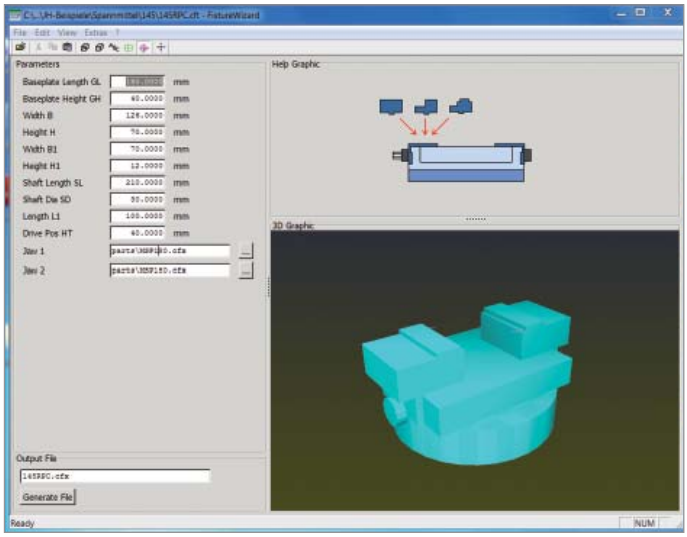
PC software
FixtureWizard

FixtureWizard lets you handily create concrete fixtures or concrete carrier kinematics from fixture templates or tool-carrier kinematic models. You can then integrate the files you create into the collision monitor 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 fits to your concrete 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 monitor.





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	
For more information –	

PC software
Programming station

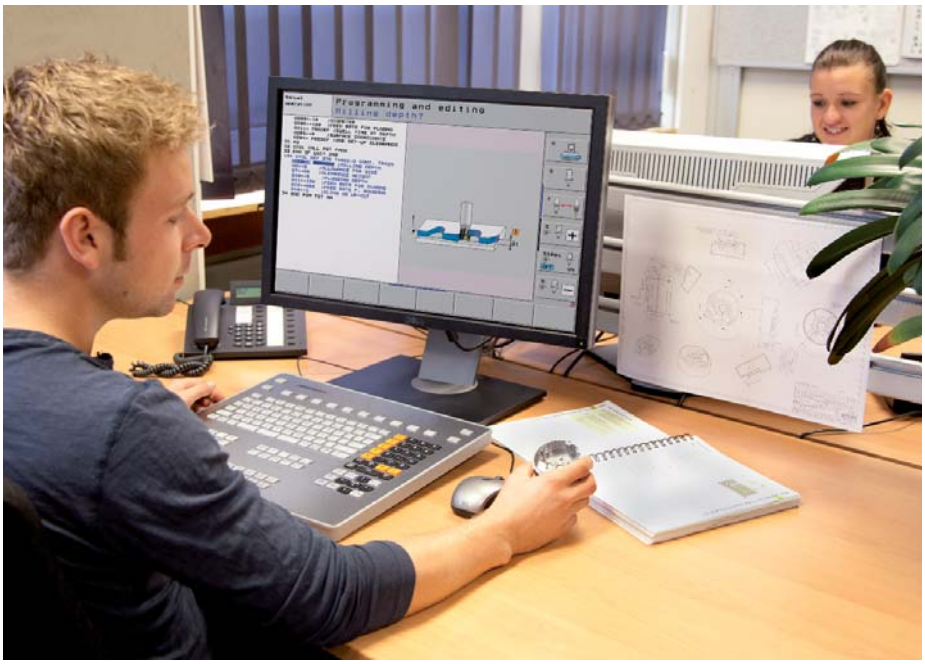
Why a programming station?
Everyone knows that you can quite easily write a part program with the TNC at the machine, even while it's machining another part. Nevertheless, it can often happen that short reloading times and other machining tasks hinder any prolonged or concentrated programming work. With 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 iTNC 530), HEIDENHAIN conversational or DIN/ISO programs for the TNC with the programming station substantially reduces machine idle times. And you need not adjust your way of thinking—every keystroke fits, because 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 even with complex 3-D programs to easily spot contour damage and hidden details.

Training with the programming station
Because the programming station is based on the same software as the lathe control, 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.



Programming station	TNC 640	TNC 620/ TNC 320	iTNC 530
Demo version	825164-xx	741708-xx	535373-xx
Single license with TNC keyboard	801523-01	599609-01	532524-01
Single license with virtual keyboard	825163-01	682781-01	386753-02
Network license with virtual keyboard for 14 users	825166-01	682782-01	643535-01
Network license with virtual keyboard for 20 users	825167-01	682783-01	643539-01
Installation by the machine tool builder			
For more information, see the catalog *Programming Stations*			

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 handwheels.

HR 410, HR 520 and HR 550 portable handwheels
The portable HR 410, HR 520 and HR 550 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. 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).

HR130

HR410

HR 520

HR 550

Electronic handwheel			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
	With detent	Without detent					
HR 130 **HR 150**	ID 540940-01 ID 540940-06	ID 540940-03 ID 540940-07	34059x-01 34059x-01	34056x-01/73498x-01 34056x-01/73498x-01	34055x-01 34055x-01	60642x-01 60642x-01	34049x-01 34049x-01
HR 410 **HR 410 FS** **HR 520** **HR 520 FS** **HR 550 FS**	ID 535220-05 ID 578114-11 ID 670303-01 ID 670305-01 ID 602622-03	ID 296469-56 ID 337159-21 ID 970302-01 ID 670304-01 ID 598515-03	34059x-01 34059x-02 34059x-02 34059x-02 34059x-02	34056x-01/73498x-01 34056x-02/73498x-02 34056x-02/73498x-02 34056x-02/73498x-02 34056x-02/73498x-02	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-02/73498x-02	34055x-06	60642x-01	34049x-07
Installation by the user							
For more information, see the catalogs *Information for the Machine Tool Builder*							

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 automatically perform setup, measuring and inspection functions.

The touch probes for workpiece measurement are inserted in the tool holder either manually or by the tool changer. It can be equipped with various shanks depending on the machine. The ruby ball tips are available in several diameters, and the styli in different lengths. They enable you to use the probing functions offered by your NC control to automatically or manually perform the following functions:
- Workpiece alignment
- Datum setting
- Workpiece measurement

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 or over an infrared beam to the control.

Touch probes with **cable connection for signal transmission** for machines with manual tool change:
TS 220 – compact dimensions, TTL version

Touch probes with **infrared signal transmission** for machines with automatic tool change:
TS 440 – compact dimensions
TS 444 – compact dimensions, battery-free power supply through integrated air turbine generator over central compressed air supply
TS 640 – standard touch probe with wide-range infrared transmission
TS 740 – high probing accuracy and repeatability, low probing force

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 220	ID 293488-xx	34059x-01	34056x-01/73498x-01	34055x-01	60642x-01	34049x-01
TS 440	ID 620046-xx	34059x-01	34056x-01/73498x-01	34055x-01	60642x-01	34049x-01
TS 444	ID 588008-xx	34059x-01	34056x-01/73498x-01	34055x-01	60642x-01	34049x-01
TS 640	ID 620189-xx	34059x-01	34056x-01/73498x-01	34055x-01	60642x-01	34049x-01
TS 740	ID 573757-xx	34059x-01	34056x-01/73498x-01	34055x-01	60642x-01	34049x-01
Installation by the machine tool builder						
For more information, see the catalog *Touch Probes* and the CD *Touch Probes*						

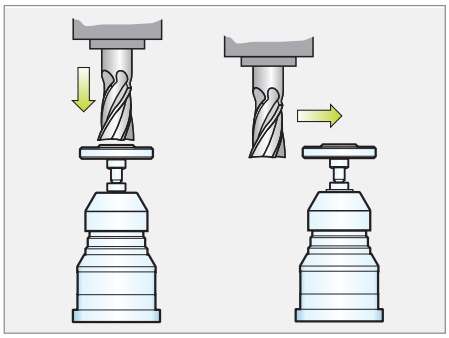
Hardware enhancements
TT, TL – tool touch probes

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

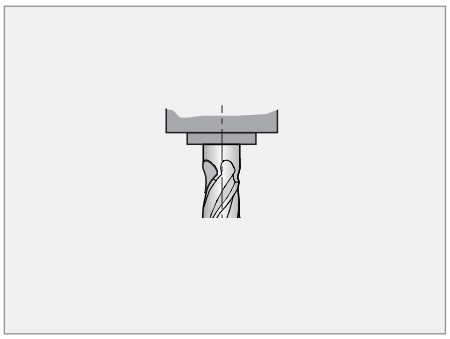
The systems are installed directly in the machine's workspace, where they permit tool measurement either before machining or during interruptions.

The **TT tool touch probes** measure the tool length and radius. When probing the tool, either while rotating or at standstill (such as for measuring individual teeth), the contact plate is deflected and a trigger signal is transmitted to the TNC. The **TT 140** uses signal transmission by cable, whereas the **TT 449** operates with signal transmission over infrared beam and does not require a cable. It is therefore particularly suitable for use on rotary and tilting tables.

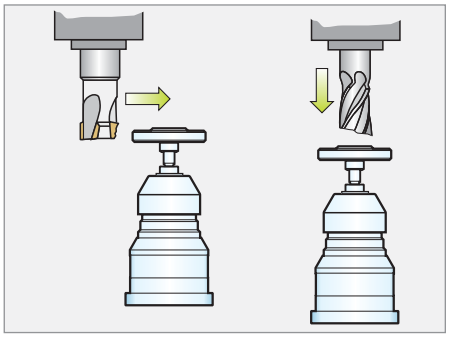
The **TL Nano** and **TL Micro laser systems** are available for various maximum tool diameters. Using a laser beam, they probe the tool without contact, and can detect form deviations of individual teeth along with the tool length and radius.



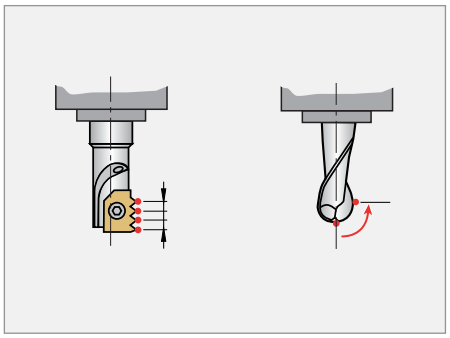
TT tool touch probe
Tool length and radius measurement with stationary or rotating spindle



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 140 **TT 449**	ID 527797-xx ID 593007-01	34059x-01 34059x-02	34056x-01/73498x-01 34056x-04/73498x-02	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 34056x-04/73498x-02 34056x-04/73498x-02 34056x-04/73498x-02	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 catalog *Touch Probes* and the CD *Touch Probes*						

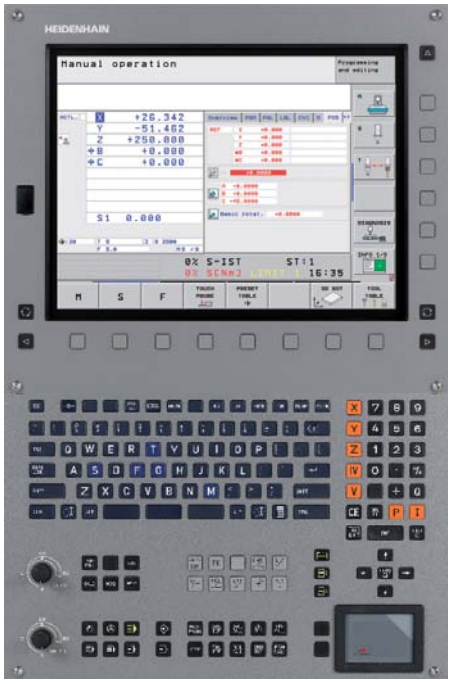
Hardware enhancements
IPC – industrial PC

IPC 6110, IPC 6120
The IPC 6110 and IPC 6120 industrial PCs from HEIDENHAIN are convenient solutions for an additional, remote station for operating the machine or machine units, such as tool-changing stations. The remote operation, which was designed for the TNC, permits very simple connection of the IPC via an Ethernet connection with a cable of up to 100 meters.

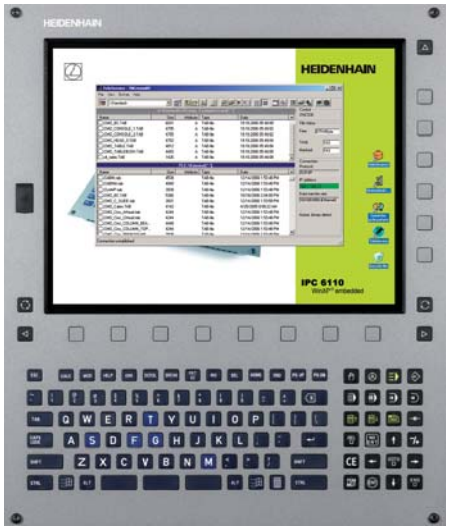
The **IPC 6110** is designed as a compact version and provides the most important function keys of the TNC in addition to the ASCII keyboard. The **IPC 6120** is used in conjunction with the complete TE 630 or TE 620 keyboard unit (without touchpad).

The control automatically detects when remote operation by the IPC is active. The IPC is shut down automatically when the control is switched off. Here the TNCterminal software included with the IPC ensures simple connection and convenient operation of the IPC on the TNC. During remote operation, the control's screen is displayed identically on the IPC, and the most important functions of the control can be operated from the integrated keyboard.

Windows XP Embedded is installed as operating system, and the TNCremo file-transmission software is also installed. This means that the transfer of programs and files from the IPC 6110 to the control is very easy via the USB port located next to the screen. The machine manufacturer also has the option of installing special additional software on the IPC.



IPC 6120



IPC 6110

IPC 6341
With the **IPC 6341** Windows computer 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 Python option 133 is required.

Since Windows runs on a separate computer, it does not influence the NC machining process. The Windows computer 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.


IPC 6341

		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 6110	ID 597968-03	–	–	–	60642x-01	34049x-04
IPC 6120	ID 664939-xx	–	–	–	60642x-01	34049x-04
IPC 6341	ID 749963-01	34059x–01	–	–	60642x-02	–
Installation by the machine tool builder						
For more information	Product Information: *IPC 6110/IPC 6120* Catalogs: *Information for the Machine Tool Builder*					

HEIDENHAIN

DR. JOHANNES HEIDENHAIN GmbH
Dr.-Johannes-Heidenhain-Straße 5
83301 Traunreut, Germany
☏ +49 8669 31-0
FAX +49 8669 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
08468 Heinsdorfergrund, Deutschland
☏ 03765 69544

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

BA Bosnia and Herzegovina → **SL**

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 **Belarus**
GERTNER Service GmbH
50354 Huerth, Germany
www.gertnergroup.com

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**
02770 Espoo, 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

ME Montenegro → **SL**

MK Macedonia → **BG**

MX **HEIDENHAIN CORPORATION MEXICO**
20235 Aguascalientes, Ags., Mexico
E-mail: info@heidenhain.com

MY **ISOSERVE Sdn. Bhd**
56100 Kuala Lumpur, Malaysia
E-mail: isoserve@po.jaring.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**
125315 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 **Posredništvo HEIDENHAIN NAVO d.o.o.**
2000 Maribor, Slovenia
www.heidenhain-hubl.si

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

TR **T&M Mühendislik San. ve Tic. LTD. ŞTİ.**
34728 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.gertnergroup.com

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!*

827222-21 · 10 · 9/2012 · F&W · Printed in Germany