HEIDENHAIN

iTNC 530

New Functions with the
NC Software 340 49x-04

September 2007

New Functions with the NC Software 340 49x-04
– The iTNC 530 Makes Setup Even Easier

The **iTNC 530** from HEIDENHAIN has proven itself for years as a versatile contouring control for milling, drilling and boring machines as well as machining centers. Along with HEIDENHAIN's plain-language conversational programming for the shop-floor, the iTNC 530 is characterized by many **useful functions** and **innovative features.** To name just a few, they include:

- Exact tool guidance with five-axis machining
- Simple tilting of the working plane
- Practice-oriented setup functions
- Very high contour accuracy for HSC milling
- Extensive fixed cycles
- Useful programming support through unambiguous function keys, free contour programming and help graphics
- Upwardly compatible part programs
- External programming and fast data transfer

The success story of the iTNC 530 also includes **smarT.NC**—the new operating mode from HEIDENHAIN. It represents another successful step toward a user-friendly interface for shop-floor programming. Well-structured input forms, straightforward graphic support, and comprehensive help texts combine with the easy-to-use pattern generator to form a compelling programming environment.

New functions for the iTNC 530
Of course there is always potential for new development, improvement and simplification. The NC software 340 49x-04 for the iTNC 530 includes a series of new functions for machine manufacturers and users. These functions make it even easier to work with the control, and they also make operation of the machine more safe. The most important are:

- Touch probe cycles for kinematic measurement
- New function in dynamic collision monitoring (DCM)
- Handwheel-superimposed traverse in the active tool-axis direction (virtual axis)

Error fixes, expansion of functions and options
As of NC software 340 49x-02, error fixes were separated from software improvements. An update of NC software will predominantly contain only **error fixes.**

New functions certainly offer added value in user-friendliness and operational reliability. Of course you also have the opportunity to purchase these new functions after a software update: These **improvements in function** will be offered as "feature upgrades," and are enabled via the Feature Content Level option.

If, for example, a control is updated from 340 49x-02 to 340 49x-04, the functions identified with "FCL 04" in the following tables are only available if the **Feature Content Level** is set from 02 to 04. Of course, the current feature content level also includes the upgrade functions of the previous NC software levels.

All of the **options** included in the respective NC software can be purchased, no matter which feature content level you have.

Inspecting and Optimizing Machine Accuracy
– Calibrating Rotary Axes with KinematicsOpt (Option)

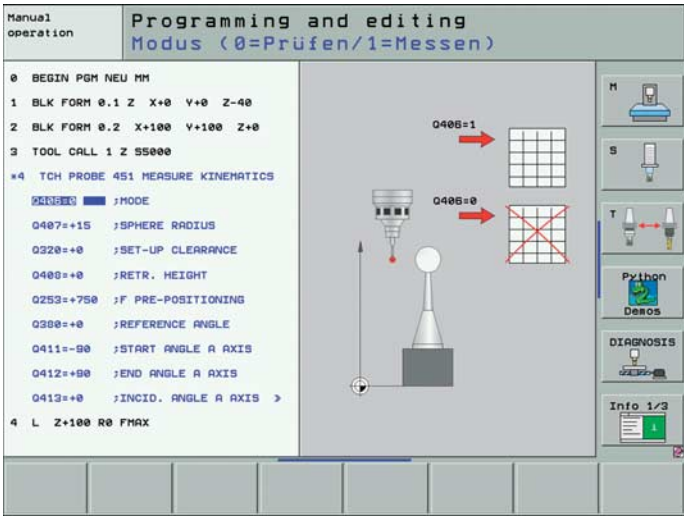
Accuracy requirements are becoming increasingly stringent, particularly in the area of 5-axis machining. Complex parts are required to manufactured with precision and reproducible accuracy even over long periods

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 measurement process is the same regardless of whether the axis is a rotary table, a tilting table or a swivel head.

To measure the rotary axes, a calibration ball 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 area that you want to measure.

From the measured values, the TNC calculates the statistical 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 kinematic table.

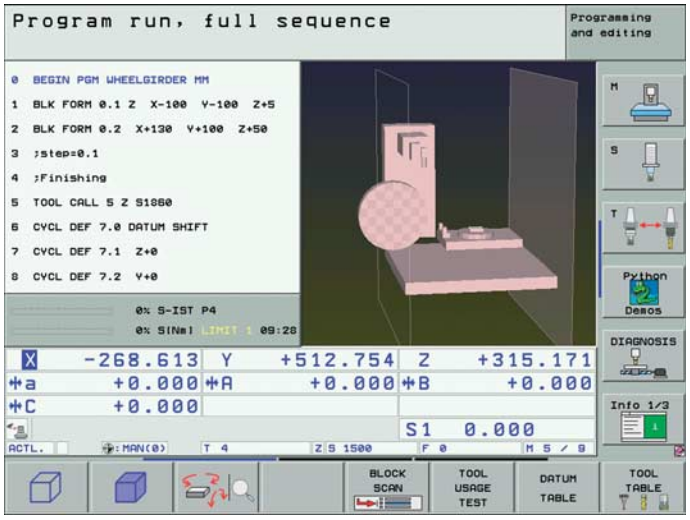
Of course, a comprehensive log file is also saved with the actual measured values and the measured and optimized dispersion (measure for the statistical tilting accuracy) as well as the actual compensation values.



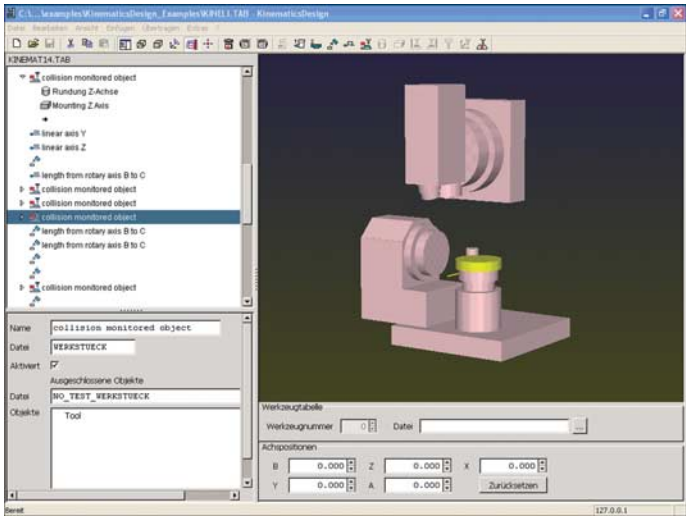
Machining Safely
– Option for Integrated Dynamic Collision Monitoring (DCM)

Since its inception in the fall of 2005, Dynamic Collision Monitoring (DCM) has proven itself countless times. About 1100 machines, or approx. 12 % of all iTNCs shipped since this time have been equipped with this option.

To improve the user's ability to observe the programmed movements within the machine's working space, the screen layout can now be arranged to show the machine kinematics as entered by the machine manufacturer with all defined obstacles at the right next to the program. If desired, the machine kinematics can be shown full screen.



Now the machine manufacturers, too, can develop kinematic tables more quickly. With KinematicsDesign, the user now has a PC tool that gives him graphic support when defining kinematic tables. KinematicsDesign makes it possible to simulate critical axis positions during the conception phase and avoid them by setting limit switches at the right positions on the machine.

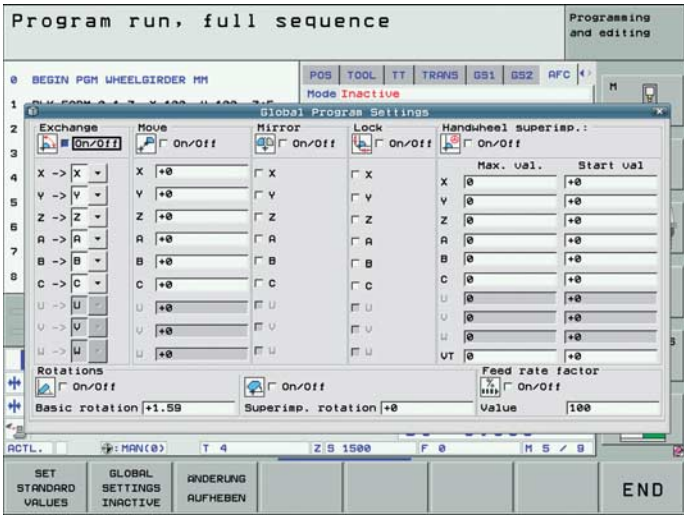


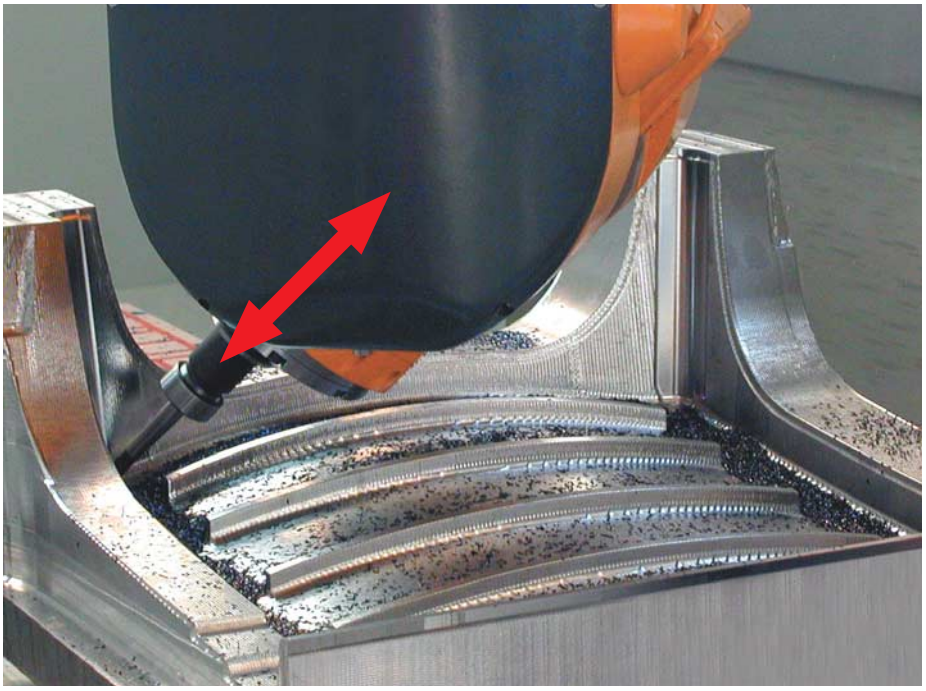
Machining Flexibly
– Option for Global Program Settings (GS)

The **Global Program Settings function (GS)** was already introduced in the NC software 340 49x-03. In the program run modes, this feature enables you to define a wide variety of coordinate transformations and settings that have global effect and are superimposed on the selected part program. Besides datum shifts, rotations and mirror images, now you can also switch axes, lock them, or set handwheel superimpositions—all functions that you can always use especially effectively when you have to modify huge, externally written part programs.

Now you can also use the global program settings (GS) to activate the **virtual axis (VT)** function that is so important for large-scale mold making. With Tool Center Point Management active (TCPM), you can use it to move the tool manually in the currently active tool axis direction, for example in order to run the entire part program with a constant oversize.

When using the HR 420 handwheel, you can select the virtual axis (VT) directly over the handwheel soft keys. At the same time, in the handwheel's display you can see the value of the distance moved in the virtual axis direction. For handwheels without integrated position display, the virtual axis can be assigned to a machine key defined by the machine manufacturer. The TNC shows the distance moved in a separate position display and also in the global program settings form. The value remains stored until you change the tool or switch the function off.

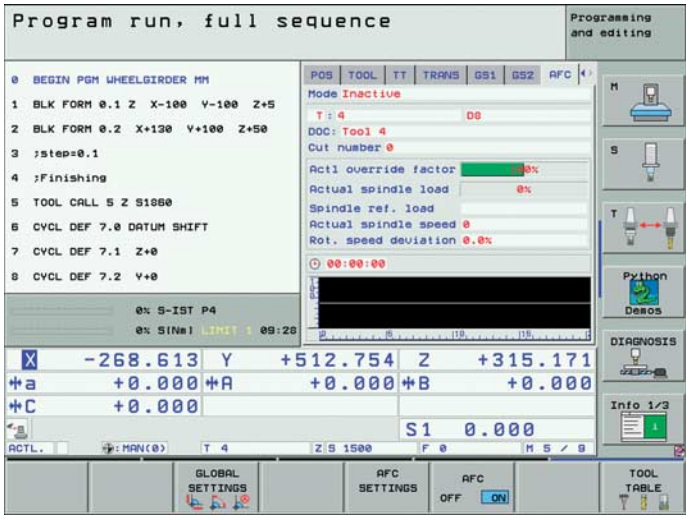




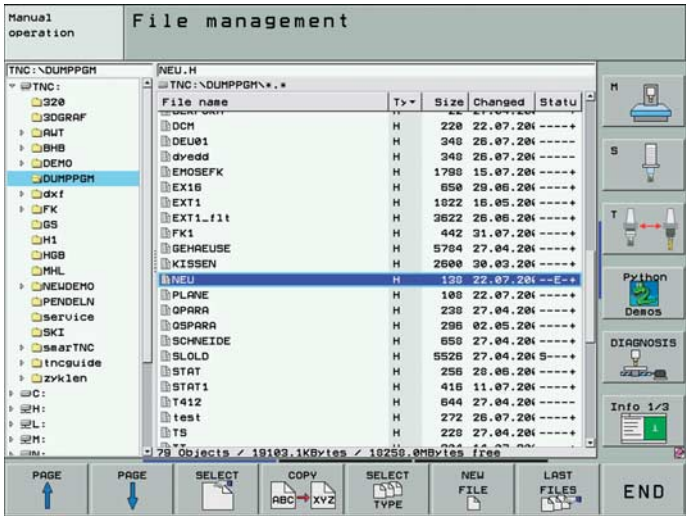
New Programming Functions
– General

3-D basic rotation, machine-specific (upgrade function)
With this function, the TNC can correct any workpiece misalignment in three dimensions (3-D set-up compensation). To use this function, your machine must be equipped with at least two rotary axes and your machine manufacturer must adapt the function to your machine. In one of the upcoming software releases, this function will be integrated in the preset table to make it machine-independent.

Option for Adaptive Feed Control (AFC)
The adaptive feed control feature (AFC) also offers new and improved functions. In the additional status display, a dynamic line chart shows the relationship of the spindle power consumption and the feed rate controlled by the TNC. During the learning phase, the TNC shows the currently saved reference power in a pop-up window. If required, you can use a soft key to reset the reference power measured up to that point and restart the learning process. In addition, your machine manufacturer can define any desired control parameter as input value for adaptive control instead of the current spindle power consumption. This makes it easier to realize special applications that require changing the feed rate when some value defined by your machine manufacturer changes.



File management
The file management was completely revised and adapted to the file management in smarT.NC. The following functions are available:
- The file management can now be operated completely by mouse and soft keys
- Sorting by name, type, size, date of change and status
- Favorites management for very simple directory selection
- Fast file selection through immediate, automatic highlight positioning during file-name entry
- Configurable display of file information
- Configurable date format

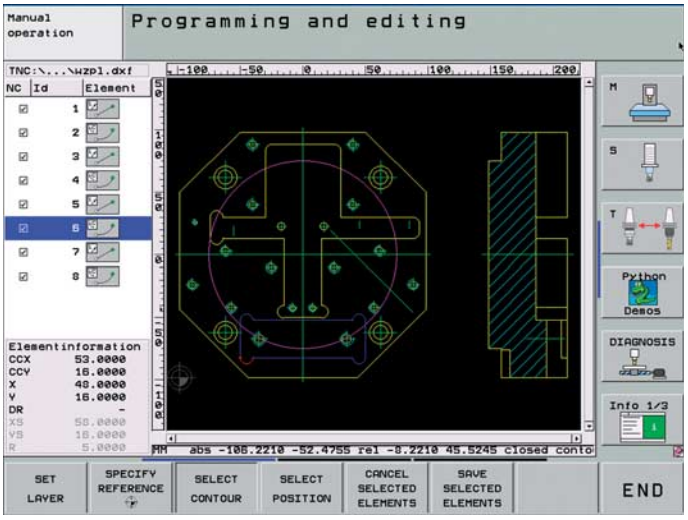


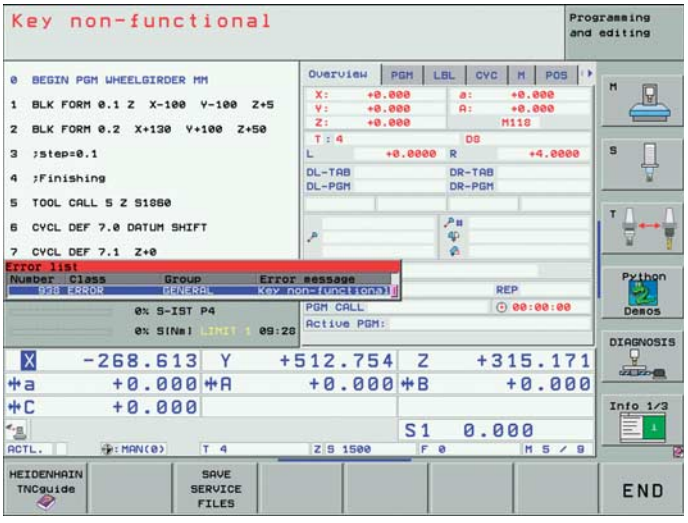
DXF converter (option)
Several improvements were made in the operation of the DXF converter. Now the TNC saves the zoom setting and the reference point of the most recently selected DXF file. The next time you select this file you can use these settings to start work immediately. The use of circle centers as machining positions was also simplified: the TNC now saves the center directly with a simple mouse click. The use of positions on quadrant transitions remains available as an option.

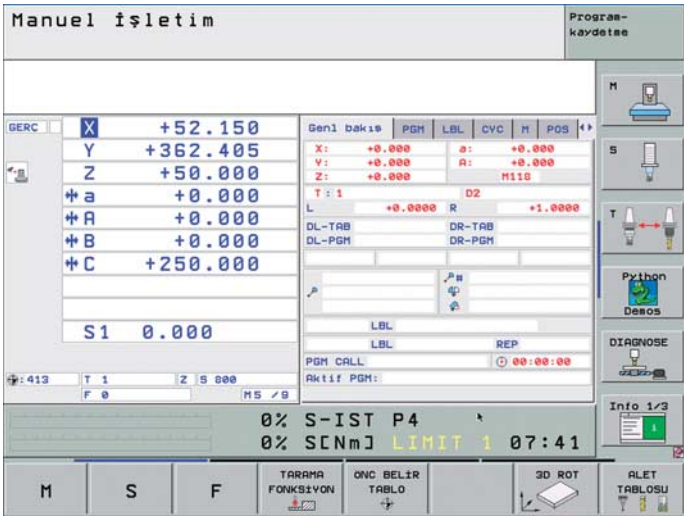
The new info box is particularly informative. It shows all the files of each selected element. You can see the X/Y coordinates for machining positions, the start and end points of contour elements, and also the center and direction of arc rotation.

Creating service files
For better support when problems or questions arise, a new feature collects all important files on the problem at hand and provides them in a .zip file. This .zip file includes the active part program, the tool table TOOL.T, and if required, the active datum table and important system files. You can then send the .zip file by e-mail over the data interface to your machine manufacturer or your HEIDENHAIN service agency.

Additional conversational languages (option)
Turkish and Romanian are now available as options.







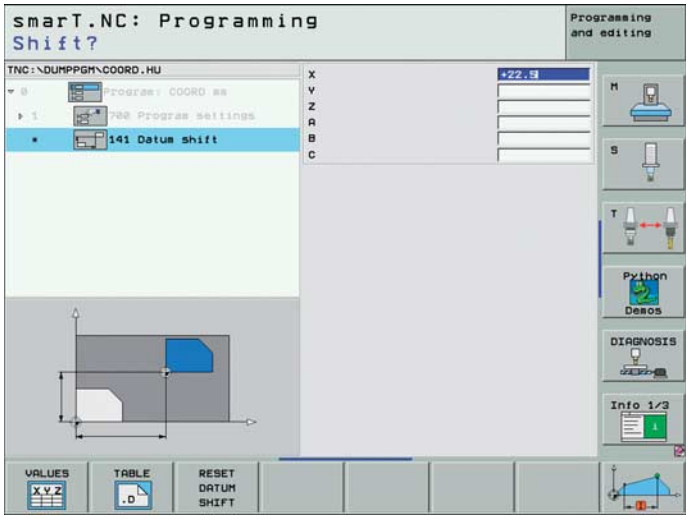
New Programming Functions
– smarT.NC

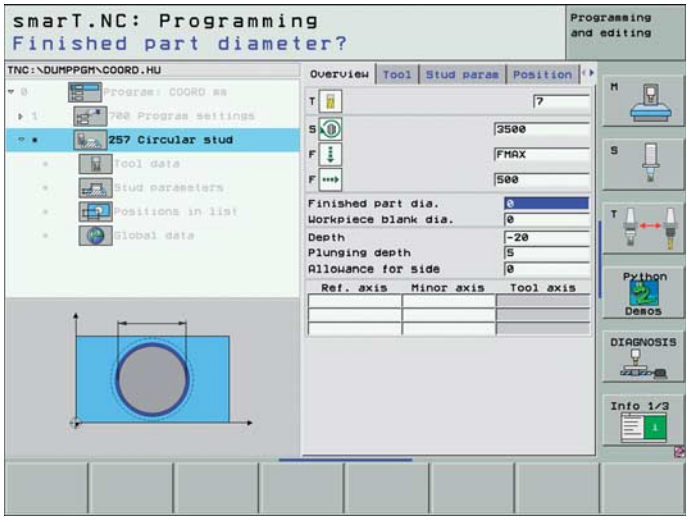
Datum shift
Axis-specific datum shifts can now be defined directly in a form. Until now, datum shifts were possible only through datum tables. In addition, the datum can easily be reset by soft key without having to program the shift value 0 in each axis.

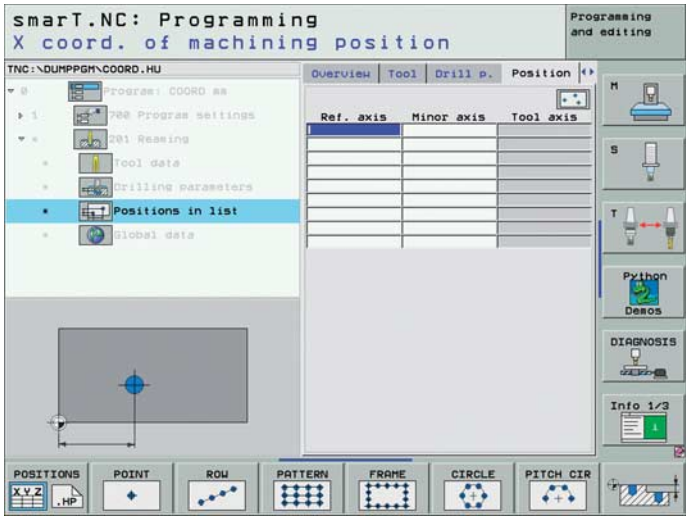
Machining rectangular and circular studs
The new Units 256 and 257 make it possible to easily machine rectangular and circular slots. In their definition and function, these new units are similar to the already available milling units 251 to 254. The constant cut distribution is particularly helpful when the difference between the dimensions of the blank and the finished part is greater than the tool radius. Of course, the distribution of cuts can be modified by an overlap factor.

Program end unit
With the program end unit you can make the following settings at the end of a smarT.NC program:
- Defining M functions, e.g. M5, M30
- Optional approach to a safety position in the tool axis, as selected in the tool or machine coordinate system
- Optional approach to a safety position in the working plane, as selected in the tool or machine coordinate system

Inline pattern definitions
Now machining patterns can also be defined directly in the overview form of a machining unit. Up to now it was necessary to call the pattern generator and define the respective pattern there. The following machining patterns are available:
- Points (up to 9 individual positions)
- Row
- Frame
- Surface
- Circular arc
- Full circle







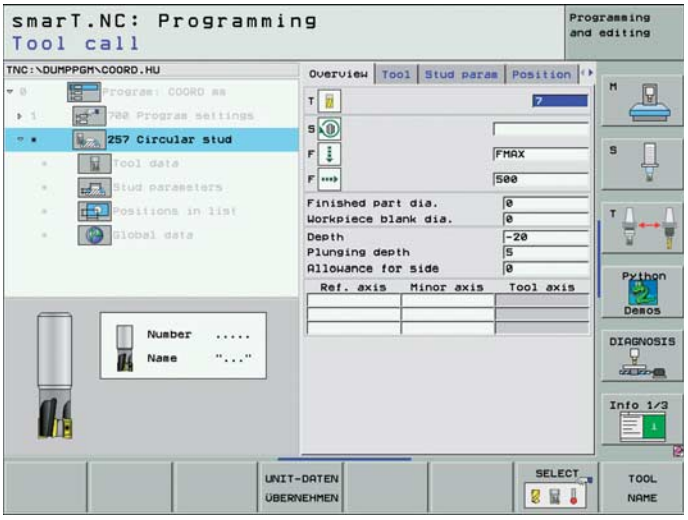
Loading values from previous units
If you want to define a unit that you have already defined in the same smarT.NC program, you can use the values defined in the earlier unit as default values for the new one. Particularly in the milling units, this makes it very easy to realize roughing and finishing definitions with different tools and/or oversizes.

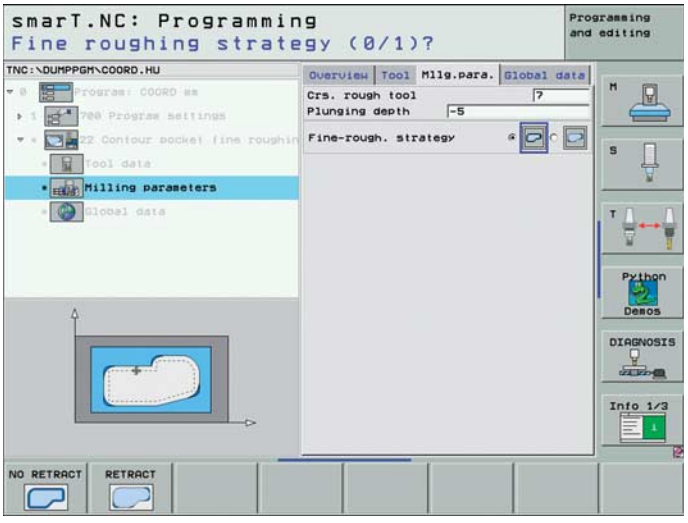
Setting the number of probing points
In touch probe units 412, 413, 421 and 422 you can now choose whether you want to measure circles with 4 (default) or 3 probe points.

Machining strategy for clearing
In Unit 22 you can now choose how the TNC will move the tool during clearing:

- Machining the complete contour:
  The TNC moves at constant height to the areas to be cleared without removing the tool from the finished part contour. This strategy works well when the distance between the areas to be cleared is small and the fine roughing tool is large enough to machine the remaining material in one step.
- Machining individual areas separately:
  After fine roughing each area to be cleared, the TNC moves the tool at rapid traverse to the safety clearance. This strategy is helpful when there is a large distance between the areas.

Fast retraction during tapping
In the tapping unit 209, you can now you can enter a factor by which the TNC increases the traversing speed when retracting the tool. This reduces your machining time.





New Programming Functions
– Conversational Programming

File functions
With the FUNCTION FILE feature you can copy, move and delete files from within the part program. This enables you, for example, to copy and start part programs that you have saved on an external drive.

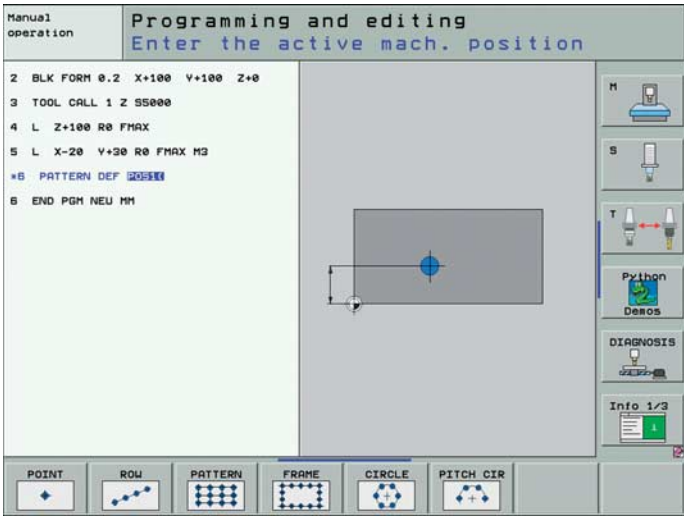
Pattern definitions
With the point pattern generator feature familiar to users of smarT.NC, you can use the PATTERN DEF function to define machining patterns. The following machining patterns are available:
- Points (up to 9 individual positions)
- Row
- Frame
- Surface
- Circular arc
- Full circle

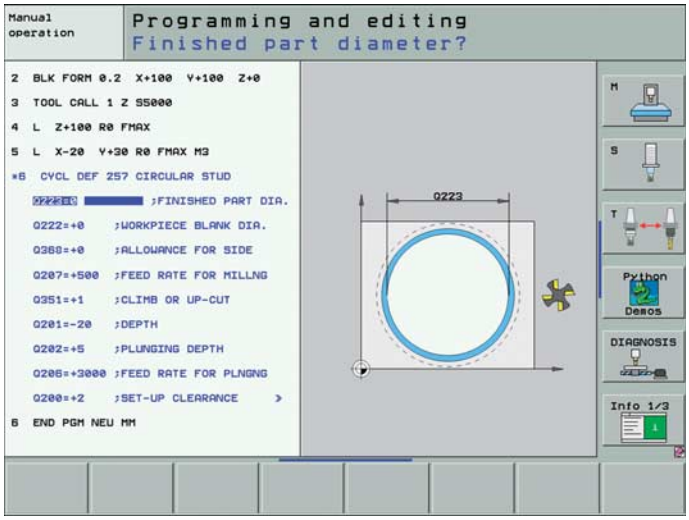
The machining positions defined in this way can be called with the CYCL CALL PATTERN function.

Globally effective cycle parameters
With the new GLOBAL DEF cycles you can define a wide variety of cycle parameters at the beginning of the program with global affect. The GLOBAL DEF cycles are classified in the following groups:
- General cycle parameters such as safety clearance or retraction feed rate
- Drilling-specific cycle parameters, such as dwell times
- Milling-specific cycle parameters, such as the plunging behavior
- Touch-probe-specific cycle parameters, such as clearance height

If you have entered GLOBAL DEF cycles in the program header, when defining a cycle you can use soft keys to apply the values defined there. The TNC then enters the word PREDEF (for predefined) in the cycle definition. Any change in the GLOBAL DEF cycle then affects all cycles that refer to the PREDEF entry in the respective GLOBAL DEF cycle.

Machining rectangular and circular studs
The new Cycles 256 and 257 make it possible to easily machine rectangular and circular slots. In their definition and function, these new cycles are similar to the already available milling cycles 251 to 254. The constant cut distribution is particularly helpful when the difference between the dimensions of the blank and the finished part is greater than the tool radius. Of course, the distribution of cuts can be modified by an overlap factor.





Overview
– All Options in the NC Software 340 49x-04

Option number	Option	As of NC software 340 49x-	ID	Remark
0 **1** **2** **3** **4** **5** **6** **7**	Additional axis	01	354 540-01 353 904-01 353 905-01 367 867-01 367 868-01 370 291-01 370 292-01 370 293-01	Additional control loops 1 to 8
8	Software option 1 (for MC 420)	01	367 591-01	**Machining with a rotary table** • Programming of cylindrical contours as if in two axes • Feed rate in distance per minute **Coordinate transformation** • Tilting the working plane, PLANE function **Interpolation** • Circle in 3 axes with tilted working plane
9	Software option 2 (for MC 420)	01	367 590-01	**3-D machining** • Motion control with minimum jerk • 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 **Interpolation** • Line in 5 axes (subject to export permit) • Spline: execution of splines (3rd degree polynomials) **Block processing time** 0.5 ms
18	HEIDENHAIN DNC	01	526 451-01	Communication with external PC applications over COM components
40	DCM Collision	02	526 452-01	Dynamic Collision Monitoring DCM (only with MC 422 B, MC 422 C)
41	Additional language	 02 03 03 03 03 03 04 04	 530 184-01 530 184-02 530 184-03 530 184-04 530 184-06 530 184-07 530 184-08 530 184-09	Additional conversational language: Slovene Slovak Latvian Norwegian Korean[1)] Estonian Turkish Romanian
42	DXF Converter	02	526 450-01	Load and convert DXF contours
44	Global PGM Settings	03	576 057-01	Global program settings
45	AFC Adaptive Feed Control	03	579 648-01	Adaptive feed control
46	Python OEM Process	04	579 650-01	Python application on the iTNC[2)]
48	KinematicsOpt	04	630 916-01	Touch probe cycles for automatic measurement of rotary axes
53	Feature content level	02	529 969-01	Level of features

[1)] Only with at least 256 MB RAM
[2)] Only with at least 512 MB RAM

Overview
– New Functions with NC Software 340 49x

Operating mode	As of NC software	Standard	FCL	Option	Function
General information	34049x-02			40	DCM: Dynamic Collision Monitoring (only with MC 422 B)
			02		USB support for peripheral memory devices (memory sticks, hard disks, CD-ROMs)
			02		DHCP (Dynamic Host Configuration Protocol) and DNS (Domain Name System) for network settings
		•			Freely definable tables visible also in form view
		•			All soft keys revised
				41	Slovene language
		•			Czech user interface now with native characters
		•			Configurable update procedure for future software updates (e.g. automatic update over USB storage devices)
		•			Additional HR 420 functions: • Selection of the active override possible on the HR 420 • Freely definable soft-key menu for machine functions
		•			Smaller pop-up window when HR 420 is active, to improve legibility of axis positions on screen
		•			Look-ahead can be configured via machine parameters
		•			Calculation of dynamic load for tilting axes
		•			Inclined tool machining with noncontrolled axes
	340 49x-03			44	Global program settings (GS) make it possible to superimpose various coordinate transformations and settings in the Program Run operating modes
				45	AFC: Adaptive feed control adjusts the contouring feed rate to the spindle power
			03		TNCguide: The integrated help system. User information available directly on the iTNC 530 (only with at least 256 MB RAM)
				41	Conversational languages in Slovak, Norwegian, Estonian, Latvian, Korean (Asian languages require at least 256 MB RAM)

Operating mode	As of NC software	Standard	FCL	Option	Function
General information	34049x-04	•			Expanded and completely revised file management
		•			Automatic and manual generation of service files for faster error diagnostics
		•			Tool-change macro for Test Run
			04		Graphic display of machine kinematics in the Program Run modes of operation
			04		3-D basic rotation light: aligning workpieces in three dimensions
				40	Improvements in Dynamic Collision Monitoring (DCM): • Handwheel superimposition possible with active DCM in stopped condition • Automatic cancelation of collision protection for touch probe measurement
				41	Turkish and Romanian languages
				44	Improvements in Global Program Setting (GS): Procedure with handwheel superimposition in the active tool-axis system (virtual axis) with active TCPM
				45	Improvements to Adaptive Feed Control (AFC): • Expanded status display • Resetting the reference power in the learning mode • Use of any value as control parameter over PLC
				46	Python OEM process: Simpler integration of OEM applications in the iTNC
				48	KinematicsOpt: Touch probe cycles for automatic measurement of rotary axes

Operating mode	As of NC software	Standard	FCL	Option	Function
smarT.NC	340 49x-02			42	Direct loading of contours from DXF data and saving as smarT.NC contouring programs
			02		Cycles for coordinate transformation introduced
			02		PLANE function introduced
			02		Contour pocket: Separate depth can be assigned for each subcontour
			02		Mid-program startup with graphical support
		•			Entry of cutting speed as alternative to the spindle shaft speed
		•			Feed rate can also be entered as Fz (feed per tooth) or Fu (feed per revolution)
		•			Tool data can be edited in a pop-up window during tool selection
		•			Axis keys now also position the cursor in the forms. The **I** key (incremental/absolute switchover) and **P** key (polar/rectangular switchover) now also function for contour programming.
		•			CUT/COPY/PASTE of one or more units
		•			Automatic entry of workpiece blank into contour program
		•			Incremental entry of machining positions in forms for machining units
		•			Tooltips displayed when using the mouse
		•			Navigation by axis key through the forms

Operating mode	As of NC software	Standard	FCL	Option	Function
smarT.NC	34049x-03			42	DXF data processing: • Separation of laterally joined contour elements • Generate point files (.HP files) directly from the DXF converter
			03		smarT.NC editor in the Programming and Editing operating mode
		•			Expanded and completely revised file management
		•			Tool table shown as a fillable form
			03		Machining a contour pocket on a point pattern
			03		Individually definable positioning heights in point patterns
			03		Touch probe units 408 and 409 for setting datums in the centerline of a slot or ridge
		•	03		Setting of probing parameters in a separate unit
			03		Automatic feed rate reduction in contour pockets during full tool engagement
		•			Climb milling/up-cut milling for helical finish milling
		•			Retraction speed for tapping with chip breaking
		•			Measured workpiece misalignment can now also be compensated by rotating a C axis
		•			Zoom function in the pattern generator
		•			Entry of stopping angle or angular step in a pitch circle definition
	34049x-04	•			Unit 141, datum shift
		•			Unit 256, machining rectangular studs
		•			Unit 257, machining circular studs
		•			Unit 799, program end unit
		•			Unit 22, fine roughing: Selectable machining strategy
		•			Unit 209, tapping: Definable rotational speed of retraction
		•			Touch probe units 412, 413, 421 and 422: Circles can be measured at either 3 or 4 points
		•			Inline pattern definition with PATTERN DEF
		•			Taking data from a similar, previously define unit
				42	DXF data processing: • Handling improvements • Info box displays data on the selected element
				48	Units 450 and 451, KinematicsOpt: touch probe cycles for automatic measurement of rotary axes

Operating mode	As of NC software	Standard	FCL	Option	Function
Conversational programming	340 49x-02			42	Reading of contours from DXF data and saving them as conversational programs
			02		Cycle for global setting of touch-probe parameters
			02		Point filter for smoothing externally created NC programs
			02		3-D line graphics for verification of programs created offline
			02		Manual traverse in the active tool-axis system
		•			Entry of cutting speed as alternative to the spindle shaft speed
		•			Simplification when working with the preset table, incremental correction of preset values possible, correction of the active preset possible
		•			Contour pockets can now contain significantly more contour elements
		•			Consideration of an active basic rotation in manual probe cycles
		•			Measuring log for probing cycles can now also be displayed on the screen during program interruption
		•			FK transformation selectable as structured plain-language or linearized plain-language
	340 49x-03			42	DXF data processing: • Separation of laterally joined contour elements • Generate point files (.HP files) directly from the DXF converter
			03		Touch probe cycles 408 and 409 for setting datums in the centerline of a slot or ridge
			03		Probing cycle for three-dimensional measurements. Results of measurement shown as desired in the coordinate system of the tool or the machine
			03		Automatic feed rate reduction in contour pockets during full tool engagement
		•			Climb milling/up-cut milling for helical finish milling
		•			Retraction speed for tapping with chip breaking
		•			Workpiece misalignment can now also be compensated by rotating a C axis

Operating mode	As of NC software	Standard	FCL	Option	Function
Conversational programming	340 49x-04	•			Cycle 256, machining rectangular studs
		•			Cycle 257, machining circular studs
		•			Cycle 22, fine roughing: selectable machining strategy
		•			Cycle 209, tapping: definable rotational speed of retraction
		•			Touch probe cycles 412, 413, 421 and 422: circles can be measured at either 3 or 4 points
		•			Special functions of smarT.NC available for conversational programming: • Defining machining patterns with PATTERN DEF • Defining cycle parameters globally with GLOBAL DEF
		•			File management (copying, moving, deleting) from within the NC program
				42	DXF data processing: • Handling improvements • Info box displays data on the selected element
				48	KinematicsOpt: touch probe cycles for automatic measurement of rotary axes
ISO	340 494-02	•			PLANE function also in possible in ISO
Programming station		•			Virtual keyboard can be displayed with new version of the programming station
		•			PLC program provided for optional installation (can be used to move axes)
		•			Access to the PLC with the keyword "PLC"
		•			All options and FCL functions are available
	340 494-04	•			Support for Windows Vista
				•	iTNC programming station available with network license

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

DE **HEIDENHAIN Technisches Büro Nord**
12681 Berlin, Deutschland
☎ (030) 54705-240
E-Mail: tbn@heidenhain.de

HEIDENHAIN Technisches Büro Mitte
08468 Heinsdorfergrund, Deutschland
☎ (03765) 69544
E-Mail: tbm@heidenhain.de

HEIDENHAIN Technisches Büro West
44379 Dortmund, Deutschland
☎ (0231) 618083-0
E-Mail: tbw@heidenhain.de

HEIDENHAIN Technisches Büro Südwest
70771 Leinfelden-Echterdingen, Deutschland
☎ (0711) 993395-0
E-Mail: tbsw@heidenhain.de

HEIDENHAIN Technisches Büro Südost
83301 Traunreut, Deutschland
☎ (08669) 31-1345
E-Mail: tbso@heidenhain.de

AR **NAKASE SRL.**
B1653AOX Villa Ballester, Argentina
☎ +54 (11) 47684242
E-Mail: nakase@nakase.com

AT **HEIDENHAIN Techn. Büro Österreich**
83301 Traunreut, Germany
☎ +49 (8669) 31-1337
E-Mail: tba@heidenhain.de

AU **FCR Motion Technology Pty. Ltd**
Laverton North 3026, Australia
☎ +61 (3) 93626800
E-Mail: vicsales@fcrmotion.com

BE **HEIDENHAIN NV/SA**
1760 Roosdaal, Belgium
☎ +32 (54) 343158
E-Mail: sales@heidenhain.be

BG **ESD Bulgaria Ltd.**
Sofia 1172, Bulgaria
☎ +359 (2) 9632949
E-Mail: info@esd.bg

BR **DIADUR Indústria e Comércio Ltda.**
04763-070 – São Paulo – SP, Brazil
☎ +55 (11) 5696-6777
E-Mail: diadur@diadur.com.br

BY Belarus → **RU**

CA **HEIDENHAIN CORPORATION**
Mississauga, Ontario L5T 2N2, Canada
☎ +1 (905) 670-8900
E-Mail: info@heidenhain.com

CH **HEIDENHAIN (SCHWEIZ) AG**
8603 Schwerzenbach, Switzerland
☎ +41 (44) 8062727
E-Mail: verkauf@heidenhain.ch

CN **DR. JOHANNES HEIDENHAIN (CHINA) Co., Ltd.**
Beijing 101312, China
☎ +86 10-80420000
E-Mail: sales@heidenhain.com.cn

CS Serbia and Montenegro → **BG**

CZ **HEIDENHAIN s.r.o.**
106 00 Praha 10, Czech Republic
☎ +420 272658131
E-Mail: heidenhain@heidenhain.cz

DK **TP TEKNIK A/S**
2670 Greve, Denmark
☎ +45 (70) 100966
E-Mail: tp-gruppen@tp-gruppen.dk

ES **FARRESA ELECTRONICA S.A.**
08028 Barcelona, Spain
☎ +34 934092491
E-Mail: farresa@farresa.es

FI **HEIDENHAIN Scandinavia AB**
02770 Espoo, Finland
☎ +358 (9) 8676476
E-Mail: info@heidenhain.fi

FR **HEIDENHAIN FRANCE sarl**
92310 Sèvres, France
☎ +33 0141143000
E-Mail: info@heidenhain.fr

GB **HEIDENHAIN (G.B.) Limited**
Burgess Hill RH15 9RD, United Kingdom
☎ +44 (1444) 247711
E-Mail: sales@heidenhain.co.uk

GR **MB Milionis Vassilis**
17341 Athens, Greece
☎ +30 (210) 9336607
E-Mail: bmilioni@otenet.gr

HK **HEIDENHAIN LTD**
Kowloon, Hong Kong
☎ +852 27591920
E-Mail: service@heidenhain.com.hk

HR Croatia → **SL**

HU **HEIDENHAIN Kereskedelmi Képviselet**
1239 Budapest, Hungary
☎ +36 (1) 4210952
E-Mail: info@heidenhain.hu

ID **PT Servitama Era Toolsindo**
Jakarta 13930, Indonesia
☎ +62 (21) 46834111
E-Mail: ptset@group.gts.co.id

IL **NEUMO VARGUS MARKETING LTD.**
Tel Aviv 61570, Israel
☎ +972 (3) 5373275
E-Mail: neumo@neumo-vargus.co.il

IN **ASHOK & LAL**
Chennai – 600 030, India
☎ +91 (44) 26151289
E-Mail: ashoklal@satyam.net.in

IT **HEIDENHAIN ITALIANA S.r.l.**
20128 Milano, Italy
☎ +39 02270751
E-Mail: info@heidenhain.it

JP **HEIDENHAIN K.K.**
Tokyo 102-0073, Japan
☎ +81 (3) 3234-7781
E-Mail: sales@heidenhain.co.jp

KR **HEIDENHAIN LTD.**
Suwon, South Korea, 443-810
☎ +82 (31) 2011511
E-Mail: info@heidenhain.co.kr

MK Macedonia → **BG**

MX **HEIDENHAIN CORPORATION MEXICO**
20235 Aguascalientes, Ags., Mexico
☎ +52 (449) 9130870
E-Mail: info@heidenhain.com

MY **ISOSERVE Sdn. Bhd**
56100 Kuala Lumpur, Malaysia
☎ +60 (3) 91320685
E-Mail: isoserve@po.jaring.my

NL **HEIDENHAIN NEDERLAND B.V.**
6716 BM Ede, Netherlands
☎ +31 (318) 581800
E-Mail: verkoop@heidenhain.nl

NO **HEIDENHAIN Scandinavia AB**
7300 Orkanger, Norway
☎ +47 72480048
E-Mail: info@heidenhain.no

PH **Machinebanks` Corporation**
Quezon City, Philippines 1113
☎ +63 (2) 7113751
E-Mail: info@machinebanks.com

PL **APS**
02-489 Warszawa, Poland
☎ +48 228639737
E-Mail: aps@apserwis.com.pl

PT **FARRESA ELECTRÓNICA, LDA.**
4470 - 177 Maia, Portugal
☎ +351 229478140
E-Mail: fep@farresa.pt

RO Romania → **HU**

RU **OOO HEIDENHAIN**
125315 Moscow, Russia
☎ +7 (495) 931-9646
E-Mail: info@heidenhain.ru

SE **HEIDENHAIN Scandinavia AB**
12739 Skärholmen, Sweden
☎ +46 (8) 53193350
E-Mail: sales@heidenhain.se

SG **HEIDENHAIN PACIFIC PTE LTD.**
Singapore 408593,
☎ +65 6749-3238
E-Mail: info@heidenhain.com.sg

SK Slovakia → **CZ**

SL **Posredništvo HEIDENHAIN SAŠO HÜBL s.p.**
2000 Maribor, Slovenia
☎ +386 (2) 4297216
E-Mail: hubl@siol.net

TH **HEIDENHAIN (THAILAND) LTD**
Bangkok 10250, Thailand
☎ +66 (2) 398-4147-8
E-Mail: info@heidenhain.co.th

TR **T&M Mühendislik San. ve Tic. LTD. ŞTİ.**
34738 Erenköy-Istanbul, Turkey
☎ +90 (216) 3022345
E-Mail: info@tmmuhendislik.com.tr

TW **HEIDENHAIN Co., Ltd.**
Taichung 407, Taiwan
☎ +886 (4) 23588977
E-Mail: info@heidenhain.com.tw

UA Ukraine → **RU**

US **HEIDENHAIN CORPORATION**
Schaumburg, IL 60173-5337, USA
☎ +1 (847) 490-1191
E-Mail: info@heidenhain.com

VE **Maquinaria Diekmann S.A.**
Caracas, 1040-A, Venezuela
☎ +58 (212) 6325410
E-Mail: purchase@diekmann.com.ve

VN **AMS Advanced Manufacturing Solutions Pte Ltd**
HCM City, Viêt Nam
☎ +84 (8) 9123658 - 8352490
E-Mail: davidgoh@amsvn.com

ZA **MAFEMA SALES SERVICES C.C.**
Midrand 1685, South Africa
☎ +27 (11) 3144416
E-Mail: mailbox@mafema.co.za

Zum Abheften hier falzen! / Fold here for filing!

Vollständige Adressen siehe www.heidenhain.de
For complete addresses see www.heidenhain.de

636 226-21 · 20 · 5/2008 · H · Printed in Germany · Subject to change without notice