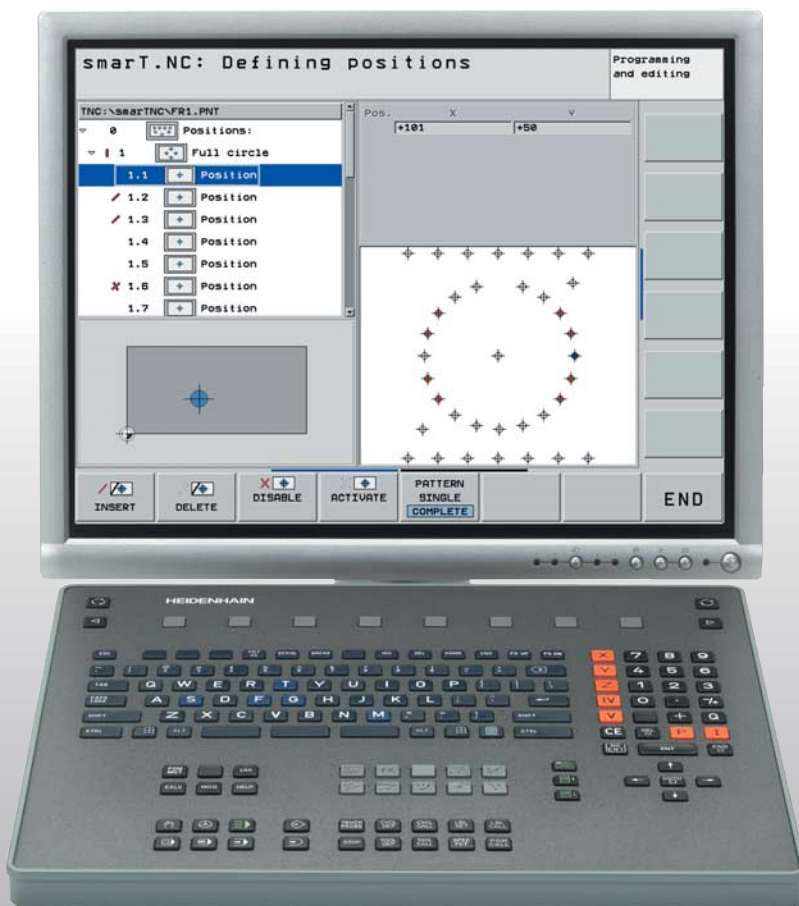




HEIDENHAIN



Operating Instructions
Programming Station

iTNC 530

NC Software
340 494-03

English (en)
8/2006



TNC Model, Software and Features

This manual describes functions and features provided by the TNC programming station as of the following NC software numbers.

TNC model	NC software number
iTNC 530 Programming Station	340 494-03

The programming station software is fully compatible with the TNCs listed below.

TNC model	NC software number
iTNC 530	340 490-03
iTNC 530 (export version)	340 491-03
iTNC 530 with Windows 2000	340 492-03
iTNC 530 with Windows 2000 (export version)	340 493-03

Many machine manufacturers, as well as HEIDENHAIN, offer programming courses for the TNCs. We recommend these courses as an effective way of enhancing your TNC programming skill and sharing information and ideas with other TNC users.



User documentation:

All TNC functions are described in the User's Manuals for the iTNC 530. Please contact HEIDENHAIN if you need a copy of these User's Manuals. Have your NC software number handy. It is shown on the iTNC's MOD screen.

The user documentation is also available as online help which can be called with the HELP key of your Programming Station.

If you have purchased the complete version of the programming station, you will find all the user documentation on the TNCguide DVD included in delivery.



Contents

Important Information about the Software for the Programming Station	1
Items Supplied/Installation	2

1 Important Information about the Software for the Programming Station 9

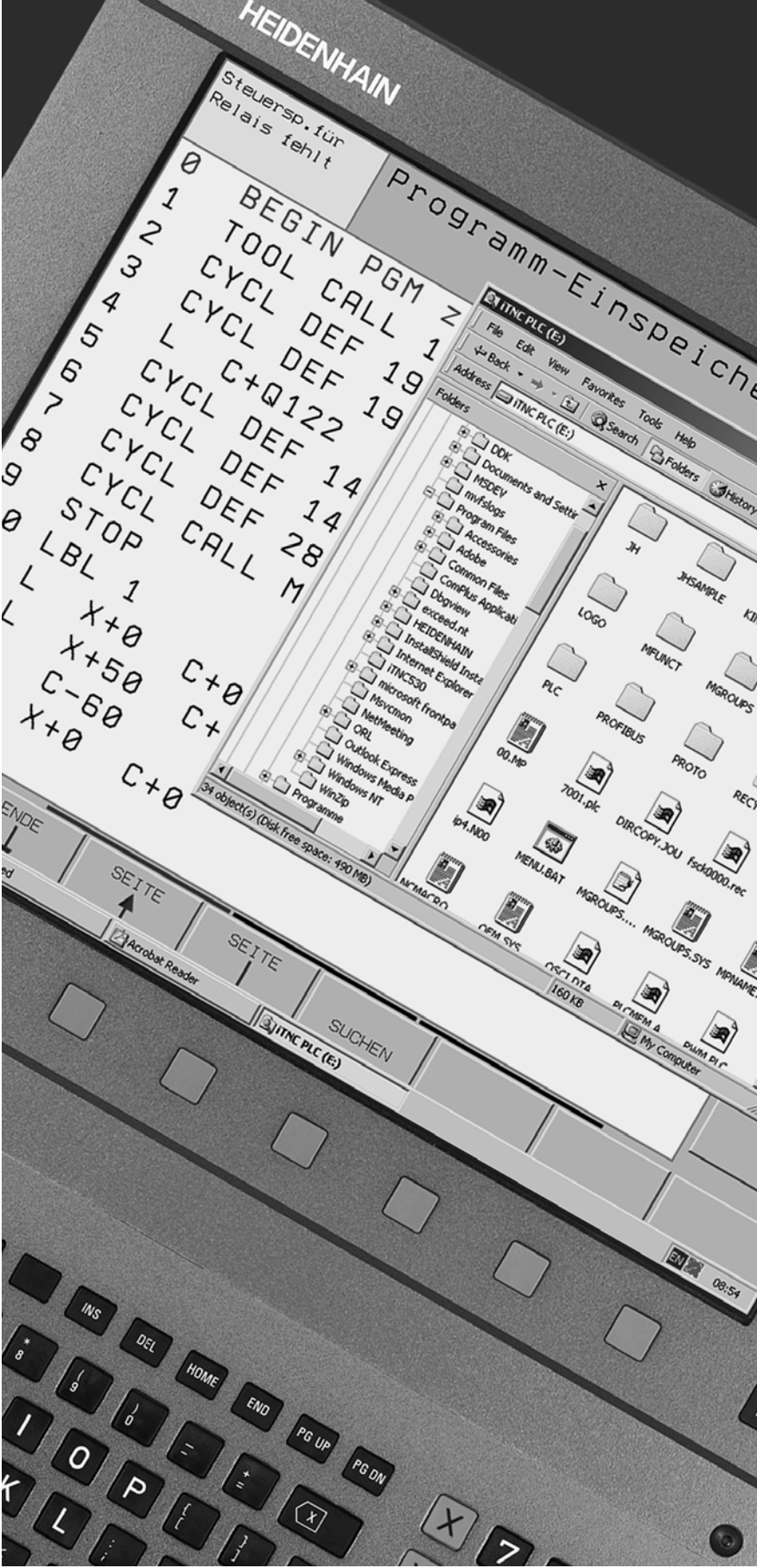
- 1.1 Introduction 10
 - General information 10
 - Options/Feature content level (FCL) 10
 - Compatibility 11
 - Downward compatibility 12
- 1.2 Working with the iTNC 530 Programming Station 27
 - Starting the programming station 27
 - Defining the window mode 27
 - Showing the virtual keyboard 28
 - Exiting the programming station 29
- 1.3 Data Transfer from the Programming Station to the Machine Tool 30
 - Prerequisites 30
 - Preparations at the programming station 30
 - Calling a program from the machine tool 31
 - Connection between the programming station and HEIDENHAIN PC software 31
- 1.4 The Demo Version 32
 - General Information 32
 - Keyboard assignment 33
 - Overview of keyboard assignment 37
 - Key assignment on portable computers 38



2 Items Supplied/Installation 39

- 2.1 Items Supplied / System Requirements 40
 - Items supplied 40
 - System requirements 40
- 2.2 Putting the Programming Station into Operation 41
 - Connect the keyboard for the programming station 41
 - Mounting the cable terminals for USB cable 41
 - Number stickers 42
 - Connecting the USB dongle (only for the version with a virtual keyboard) 42
 - Installing the programming station software 43
 - Set the dialog language 44
 - Working with the HEIDENHAIN basic PLC program 45





Important Information about the Software for the Programming Station



1.1 Introduction

General information



This manual describes the special features of the iTNC 530 programming station. All available NC functions are described in the User's Manual and the Touch Probe Cycles Manual. smarT.NC functions are described in the smarT.NC pilot.

The TNC controls from HEIDENHAIN have always been user-friendly: Simple programming in HEIDENHAIN conversational format, field-proven cycles, unambiguous function keys and clearly structured graphic functions make them extremely popular shop-floor programmable controls.

Now there is also software for running a programming station using Windows. The HEIDENHAIN keyboard included in delivery is simply connected to the USB interface on your PC. For test purposes you can also use the iTNC 530 programming station with your standard PC keyboard (see "The Demo Version" on page 32).

As an alternative, you can also use the virtual keyboard for the programming station (see "Showing the virtual keyboard" on page 28).

Options/Feature content level (FCL)

Some powerful functions are not available as a standard feature on all TNC controls, but must be enabled via a keyword.

All functions are available without surcharge on the iTNC 530 programming station. The free demo version, however, only permits storage of programs up to a certain program length (see "The Demo Version" on page 32).



When creating programs with the iTNC 530 programming station, please ensure that you only use functions actually available on your machine. Otherwise, the program could contain **ERROR** blocks after it has been downloaded to the machine control, or error messages could appear during testing or program run.

Compatibility

The programming modes provide the same features as an iTNC 530 connected to a machine tool (with NC software 340 490-03). You create programs

- in HEIDENHAIN conversational format, according to ISO, or with smarT.NC,
- with graphic support during programming and for program verification,
- and with all other proven iTNC features, such as FK free contour programming.

You work with the original iTNC software—without any compatibility programs. The programs created with the programming station run on machines equipped with an iTNC 530 and the NC software 340 490-03. A requirement for this to work is that the programming station software must have been interfaced to the machine and that the software options you are using on your machine must have been enabled.



If you also use the programming station to create programs for older software versions of the iTNC 530 or older TNC contouring controls, please note the next section **Downward compatibility**.



Downward compatibility

The available functions of the iTNC 530 programming station exactly match those of the TNC software 340 490-03. If your TNC control is running this software, then you can download all programs created with the programming station directly to the control. If you want to use the programming station to create programs for older TNC controls, please note the following:

- Do not use any features of the iTNC 530 that are not available on older TNC controls. The next pages include an overview of which features are available with which software versions.
- For some cycles, parameters that were not available on older controls or in older software versions for the iTNC 530 have been added (see "Differences in machining cycles concerning the TNC 4xx," page 13, or see "Differences in touch probe cycles concerning the TNC 4xx," page 16). These additional parameters are identified internally as optional. Beginning with the last parameter of the respective cycle, you can use the NO ENT key to delete them from the cycle definition to make the program downward compatible.



If there are any uncertainties, compare the program created on the iTNC 530 programming station with the block formatting of your control as it is described in its User's Manual.

Differences in machining cycles concerning the TNC 4xx

Meaning of the symbols used in the table:

- **Parameter not available** in this software version
- ✓ Parameter or cycle available in this software version
- x **Cycle not available** in this software version

Cycle	Additional parameters	TNC 426/ TNC 430, software version					TNC 410
		280 476	280 474	280 472	280 470	280 462	286 060
20, Contour data	–	✓	✓	✓	✓	✓	x
21, Pilot drilling	–	✓	✓	✓	✓	✓	x
22, Rough-out	Q208 Q401	–	–	–	–	–	x
		–	–	–	–	–	x
23, Floor finishing	–	✓	✓	✓	✓	✓	x
24, Side finishing	–	✓	✓	✓	✓	✓	x
25, Contour train	–	✓	✓	✓	✓	✓	x
27, Cylinder surface	–	✓	✓	✓	✓	✓	x
28, Cylinder surface	–	✓	✓	x	x	x	x
30, 3-D data	–	✓	✓	✓	✓	✓	x
32, Tolerance	HSC-MODE TA	–	–	–	x	x	x
		–	–	–			
200, Drilling	Q211	✓	✓	–	–	–	–
202, Boring	Q336	✓	✓	–	–	–	–
203, Universal drilling	Q256	✓	✓	–	–	–	–
204, Back boring	Q336	✓	✓	–	x	x	x
205, Universal pecking	Q379	–	–	x	x	x	–
	Q253	–	–				–
206, Tapping with floating tap holder	–	✓	✓	x	x	x	x
207, Rigid tapping	–	✓	✓	x	x	x	x
208, Bore milling	Q342	✓	–	x	x	x	x
	Q351	–	–	x	x	x	x
209, Tapping with chip breaking	–	✓	✓	x	x	x	x
210, Slot with reciprocating plunge	Q338	✓	✓	–	–	–	–
	Q206	–	–	–	–	–	–



Cycle	Additional parameters	TNC 426/ TNC 430, software version					TNC 410
		280 476	280 474	280 472	280 470	280 462	286 060
211, Circular slot	Q338	✓	✓	✓	✓	✓	–
	Q206	–	–	–	–	–	–
220, Point pattern on circle	Q301	✓	✓	–	–	–	–
	Q365	–	–	–	–	–	–
221, Point pattern on lines	Q301	✓	✓	–	–	–	–
247, Datum setting	–	✓	x	x	x	x	x
251, Rectangular pocket (complete)	–	x	x	x	x	x	x
252, Circular pocket (complete)	–	x	x	x	x	x	x
253, Slot (complete)	–	x	x	x	x	x	x
254, Circular slot (complete)	–	x	x	x	x	x	x
262, Thread milling	–	✓	x	x	x	x	x
263, Thread milling/counter sinking	–	✓	x	x	x	x	x
264, Thread drilling/milling	–	✓	x	x	x	x	x
265, Helical thread drilling/milling	–	✓	x	x	x	x	x
267, Outside thread milling	–	✓	x	x	x	x	x



Differences in machining cycles concerning older software versions of the iTNC 530

Meaning of the symbols used in the table:

- **Parameter not available** in this software version
- ✓ Parameter or cycle available in this software version
- x **Cycle not available** in this software version

Cycle	Additional parameters	iTNC 530, software version	
		340 422	340 420
22, Rough-out	Q208 Q401	✓ –	– –
205, Universal pecking	Q379	✓	–
	Q253	✓	–
208, Bore milling	Q351	–	–
251, Rectangular pocket (complete)	–	✓	x
252, Circular pocket (complete)	–	✓	x
253, Slot (complete)	–	✓	x
254, Circular slot (complete)	–	✓	x



Differences in touch probe cycles concerning the TNC 4xx

Meaning of the symbols used in the table:

- **Parameter not available** in this software version
- ✓ Parameter or cycle available in this software version
- x **Cycle not available** in this software version

Cycle	Additional parameters	TNC 426/ TNC 430, software version					TNC 410
		280 476	280 474	280 472	280 470	280 462	286 060
1, Polar datum plane	–	✓	✓	✓	x	x	x
2, Calibrate TS	–	✓	✓	✓	x	x	x
3, Measuring	MB	–	–	x	x	x	x
	DATUM SYSTEM	–	–	–	–	–	
4, Measuring in 3-D	–	x	x	x	x	x	x
9, Calibrate TS length	–	x	x	x	x	x	x
400, Basic rotation	Q305	–	–	–	x	x	x
401, Basic rot. from two holes	Q305	–	–	–	x	x	x
	Q402	–	–	–	x	x	x
	Q337	–	–	–	x	x	
402, Basic rot. from two studs	Q305	–	–	–	x	x	x
	Q402	–	–	–	x	x	x
	Q337	–	–	–	x	x	
403, Basic rot. via rotary axis	Q337	–	–	–	x	x	x
	Q305	–	–	–	x	x	
	Q303	–	–	–	x	x	
	Q380	–	–	–	x	x	
404, Set basic rotation	–	✓	✓	x	x	x	x
405, Rotation of the C axis	–	✓	✓	x	x	x	x
408, Slot center ref pt	–	x	x	x	x	x	x
409, Ridge center ref pt	–	x	x	x	x	x	x

Cycle	Additional parameters	TNC 426/ TNC 430, software version					TNC 410
		280 476	280 474	280 472	280 470	280 462	286 060
410, Datum from inside of rectangle	Q303	–	–	–	X	X	X
	Q381	–	–	–	X	X	
	Q382	–	–	–	X	X	
	Q383	–	–	–	X	X	
	Q384	–	–	–	X	X	
	Q333	–	–	–	X	X	
411, Datum from outside of rectangle	Q303	–	–	–	X	X	X
	Q381	–	–	–	X	X	
	Q382	–	–	–	X	X	
	Q383	–	–	–	X	X	
	Q384	–	–	–	X	X	
	Q333	–	–	–	X	X	
412, Datum from inside of circle	Q303	–	–	–	X	X	X
	Q381	–	–	–	X	X	
	Q382	–	–	–	X	X	
	Q383	–	–	–	X	X	
	Q384	–	–	–	X	X	
	Q333	–	–	–	X	X	
413, Datum from outside of circle	Q303	–	–	–	X	X	X
	Q381	–	–	–	X	X	
	Q382	–	–	–	X	X	
	Q383	–	–	–	X	X	
	Q384	–	–	–	X	X	
	Q333	–	–	–	X	X	
414, Datum in inside corner	Q303	–	–	–	X	X	X
	Q381	–	–	–	X	X	
	Q382	–	–	–	X	X	
	Q383	–	–	–	X	X	
	Q384	–	–	–	X	X	
	Q333	–	–	–	X	X	



Cycle	Additional parameters	TNC 426/ TNC 430, software version					TNC 410
		280 476	280 474	280 472	280 470	280 462	286 060
415, Datum in outside corner	Q303	–	–	–	X	X	X
	Q381	–	–	–	X	X	
	Q382	–	–	–	X	X	
	Q383	–	–	–	X	X	
	Q384	–	–	–	X	X	
	Q333	–	–	–	X	X	
416, Datum in center of bolt hole circle	Q303	–	–	–	X	X	X
	Q381	–	–	–	X	X	
	Q382	–	–	–	X	X	
	Q383	–	–	–	X	X	
	Q384	–	–	–	X	X	
	Q333	–	–	–	X	X	
417, Datum in TS axis	Q303	–	–	–	X	X	X
418, Datum using 4 holes	Q303	–	–	–	X	X	X
	Q381	–	–	–	X	X	
	Q382	–	–	–	X	X	
	Q383	–	–	–	X	X	
	Q384	–	–	–	X	X	
	Q333	–	–	–	X	X	
419, Datum in one axis	–	X	X	X	X	X	X
420, Measure angle	–	✓	✓	✓	X	X	X
421, Measure hole	–	✓	✓	✓	X	X	X
422, Measure circle outside	–	✓	✓	✓	X	X	X
423, Measure rectangle inside	–	✓	✓	✓	X	X	X
424, Measure rectangle outside	–	✓	✓	✓	X	X	X
425, Measure inside width	–	✓	✓	✓	X	X	X
426, Measure ridge outside	–	✓	✓	✓	X	X	X
427, Measure coordinate	–	✓	✓	✓	X	X	X
430, Measure bolt hole circle	–	✓	✓	✓	X	X	X
431, Measure plane	–	✓	✓	✓	X	X	X
440, Measure axis shift	–	✓	X	X	X	X	X

Cycle	Additional parameters	TNC 426/ TNC 430, software version					TNC 410
		280 476	280 474	280 472	280 470	280 462	286 060
441, Fast probing	–	x	x	x	x	x	x
480, Calibrate TT	–	✓	x	x	x	x	x
481, Measure tool length	–	✓	x	x	x	x	x
482, Measure tool radius	–	✓	x	x	x	x	x
483, Measure length and radius	–	✓	x	x	x	x	x



Differences in touch probe cycles concerning older software versions of the iTNC 530

Meaning of the symbols used in the table:

- **Parameter not available** in this software version
- ✓ Parameter or cycle available in this software version
- x **Cycle not available** in this software version

Cycle	Additional parameters	iTNC 530, software version	
		340 422	340 420
4, Measuring in 3-D	–	x	x
9, Calibrate TS length	–	✓	x
400, Basic rotation	Q305	✓	–
401, Basic rot. from two holes	Q305	✓	–
	Q402	–	–
	Q337	–	–
402, Basic rot. from two studs	Q305	✓	–
	Q402	–	–
	Q337	–	–
403, Basic rot. via rotary axis	Q337	✓	–
	Q305	✓	–
	Q380	✓	–
408, Slot center ref pt	–	x	x
409, Ridge center ref pt	–	x	x
410, Datum from inside of rectangle	Q303	✓	–
	Q381	✓	–
	Q382	✓	–
	Q383	✓	–
	Q384	✓	–
	Q333	✓	–
411, Datum from outside of rectangle	Q303	✓	–
	Q381	✓	–
	Q382	✓	–
	Q383	✓	–
	Q384	✓	–
	Q333	✓	–

Cycle	Additional parameters	iTNC 530, software version	
		340 422	340 420
412, Datum from inside of circle	Q303	✓	–
	Q381	✓	–
	Q382	✓	–
	Q383	✓	–
	Q384	✓	–
	Q333	✓	–
413, Datum from outside of circle	Q303	✓	–
	Q381	✓	–
	Q382	✓	–
	Q383	✓	–
	Q384	✓	–
	Q333	✓	–
414, Datum in inside corner	Q303	✓	–
	Q381	✓	–
	Q382	✓	–
	Q383	✓	–
	Q384	✓	–
	Q333	✓	–
415, Datum in outside corner	Q303	✓	–
	Q381	✓	–
	Q382	✓	–
	Q383	✓	–
	Q384	✓	–
	Q333	✓	–
416, Datum in center of bolt hole circle	Q303	✓	–
	Q381	✓	–
	Q382	✓	–
	Q383	✓	–
	Q384	✓	–
	Q333	✓	–
417, Datum in TS axis	Q303	✓	–



Cycle	Additional parameters	iTNC 530, software version	
		340 422	340 420
418, Datum using 4 holes	Q303	✓	–
	Q381	✓	–
	Q382	✓	–
	Q383	✓	–
	Q384	✓	–
	Q333	✓	–
419, Datum in one axis	–	✓	x
441, Fast probing	–	x	x



Differences in miscellaneous functions M concerning the TNC 4xx

Meaning of the symbols used in the table:

x **Function not available** in this software version

✓ **Function available** in this software version

M Function	TNC 426/ TNC 430, software version					TNC 410 286 060
	280 476	280 474	280 472	280 470	280 462	
Two miscellaneous functions can be programmed in one NC block	✓	x	x	x	x	✓
M01	✓	x	x	x	x	✓
M04	✓	✓	x	x	x	x
M28	✓	✓	✓	x	x	x
M29	✓	✓	✓	x	x	x
M30	✓	✓	✓	✓	x	x
M34	✓	✓	✓	x	x	x
M35	✓	✓	✓	x	x	x
M36	✓	✓	x	x	x	x
M37	✓	✓	x	x	x	x
M38	✓	✓	x	x	x	x
M40	✓	x	x	x	x	x
M41	✓	x	x	x	x	x
M42	✓	x	x	x	x	x
M43	✓	x	x	x	x	x
M44	✓	x	x	x	x	x
M45	✓	x	x	x	x	x

Differences in miscellaneous functions M concerning older software versions of the iTNC 530

Meaning of the symbols used in the table:

x **Function not available** in this software version

✓ **Function available** in this software version

M Function	iTNC 530, software version	
	340 422	340 420
M50	x	x



Differences in Q parameter programming concerning the TNC 4xx

Meaning of the symbols used in the table:

- x **Function not available** in this software version
- ✓ **Function available** in this software version

Function	TNC 426/ TNC 430, software version					TNC 410 286 060
	280 476	280 474	280 472	280 470	280 462	
Formula entry: SGN	x	x	x	x	x	✓
Formula entry: %	x	x	x	x	x	x
FN16	✓	✓	✓	✓	✓	x
FN20	✓	✓	✓	x	x	x
FN23	✓	✓	✓	x	x	x
FN24	✓	✓	✓	x	x	x
FN25	✓	✓	✓	x	x	x
FN26	✓	✓	x	x	x	x
FN27	✓	✓	x	x	x	x
FN28	✓	✓	x	x	x	x
Contour formula QC...	x	x	x	x	x	x
String formula QS...	x	x	x	x	x	x

Differences in Q parameter programming concerning older software versions of the iTNC 530

Meaning of the symbols used in the table:

- x **Function not available** in this software version
- ✓ **Function available** in this software version

Function	iTNC 530, software version	
	340 422	340 420
String formula QS...	x	x



Differences in other functions concerning the TNC 4xx

Meaning of the symbols used in the table:

x **Function not available** in this software version

✓ **Function available** in this software version

Function	TNC 426/ TNC 430, software version					TNC 410 286 060
	280 476	280 474	280 472	280 470	280 462	
CYCL CALL PAT	✓	x	x	x	x	✓
CYCL CALL POS	x	x	x	x	x	x
Cutting data tables (WMAT block)	✓	✓	✓	x	x	x
Feed rate F in the CHF block	✓	✓	✓	x	x	✓
Contour definition DECLARE CONTOUR	x	x	x	x	x	x
Tilting the working plane: PLANE	x	x	x	x	x	x
Positioning behavior of rotary axes TCPM FUNCTION	x	x	x	x	x	x
FK programming: FL P2X/FLT P2X	✓	✓	✓	✓	✓	x
FK programming: FL P2Y/FLT P2Y	✓	✓	✓	✓	✓	x
FK programming: FL RX/FLT RX	✓	✓	✓	✓	✓	x
FK programming: FL RY/FLT RY	✓	✓	✓	✓	✓	x
FK programming: FL RPR/FLT RPR	✓	✓	✓	✓	✓	x
FK programming: FL RPA/FLT RPA	✓	✓	✓	✓	✓	x
FK programming: FL RAN/FLT RAN	✓	✓	✓	✓	✓	x
FK programming: FC LEN/FCT LEN	✓	✓	✓	✓	✓	x
FK programming: FC AN/FCT AN	✓	✓	✓	✓	✓	x
FK programming: FC P2X/FCT P2X	✓	✓	✓	✓	✓	x
FK programming: FC P2Y/FCT P2Y	✓	✓	✓	✓	✓	x
FK programming: FC P3X/FCT P3X	✓	✓	✓	✓	✓	x
FK programming: FC P3Y/FCT P3Y	✓	✓	✓	✓	✓	x
FK programming: FC RX/FCT RX	✓	✓	✓	✓	✓	x
FK programming: FC RY/FCT RY	✓	✓	✓	✓	✓	x
FK programming: FC RPR/FCT RPR	✓	✓	✓	✓	✓	x
FK programming: FC RPA/FCT RPA	✓	✓	✓	✓	✓	x
FK programming: FC RAN/FCT RAN	✓	✓	✓	✓	✓	x



Function	TNC 426/ TNC 430, software version					TNC 410 286 060
	280 476	280 474	280 472	280 470	280 462	
FK programming: FC RCCX/FCT RCCX	✓	✓	✓	✓	✓	x
FK programming: FC RCCY/FCT RCCY	✓	✓	✓	✓	✓	x
FK programming: FC RCCPR/FCT RCCPR	✓	✓	✓	✓	✓	x
FK programming: FC RCCPA/FCT RCCPA	✓	✓	✓	✓	✓	x

Differences in other functions concerning older software versions of the iTNC 530

Meaning of the symbols used in the table:

- x **Function not available** in this software version
- ✓ **Function available** in this software version

Function	iTNC 530, software version	
	340 422	340 420
Tilting the working plane: PLANE	✓	x
Tilting the working plane: PLANE AXIS	x	x
Positioning behavior of rotary axes TCPM FUNCTION	✓	x



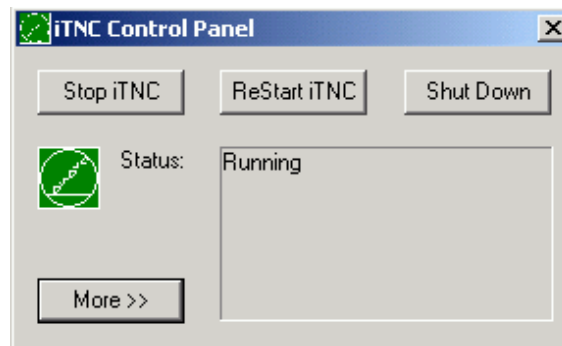
1.2 Working with the iTNC 530 Programming Station

Starting the programming station

Start the iTNC 530 programming station as you would any other Windows application. Either double-click the HEIDENHAIN icon on your desktop or use the Start menu. The iTNC Control Panel appears (see figure at upper right). As soon as the programming station software is started, the iTNC Control Panel automatically minimizes.



The TNC functions (e.g. **L**, **C**, **CC** etc.) are sent internally over certain key combinations (accelerators) to the programming station software. Remember that such key combinations are also used by other Windows programs, which can result in undesired effects.



Defining the window mode

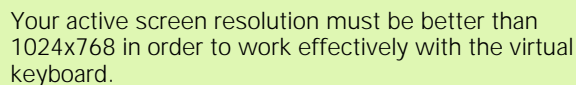
In the standard setting, the programming station software starts in full screen mode. If your PC permits a higher resolution than 1024x768, you can also start the programming station in the window mode so that the programming station behaves like any other Windows application.

To select the window mode, proceed as follows:

- ▶ Start the programming station software.
- ▶ Double-click the green HEIDENHAIN symbol to the lower right in the Task Bar for the iTNC Control Panel to appear (see figure).
- ▶ Click the **More >>** button: Additional settings are displayed
- ▶ Click the **Settings** button: Display options are displayed
- ▶ Select the **1024 x 768 window** option. Confirm with the **OK** button.



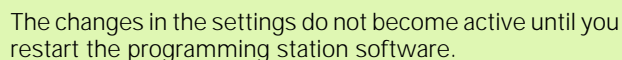
The changes in the settings do not become active until you restart the programming station software.



- Scroll the soft-key row to the left: F9
- Scroll the soft-key row to the right: F10
- Set the screen layout: F11
- Switch between the foreground and background operating mode: F12

- ▶ Start the programming station software.
- ▶ Double-click the green HEIDENHAIN symbol to the lower right in the Task Bar for the iTNC Control Panel to appear.
- ▶ Click the **More >>** button: Additional settings are displayed
- ▶ Click the **Keypad** button: The virtual keyboard is shown

- ▶ Start the programming station software.
- ▶ Double-click the green HEIDENHAIN symbol to the lower right in the Task Bar for the iTNC Control Panel to appear.
- ▶ Click the **More >>** button: Additional settings are displayed
- ▶ Click the **Settings** button: Display options are displayed
- ▶ Select the **Launch keypad at startup** option. Confirm with the **OK** button.



Exiting the programming station

Fundamentals

In order to avoid losing data when exiting the program, you must quit the programming station correctly. The following sections describe the two possibilities for doing this.



Inappropriate exiting of the programming station can lead to data loss.

Exiting the programming station completely

- ▶ Select the Manual Operation mode.
- ▶ Shift the soft-key row until the soft key for shutting down the system appears.



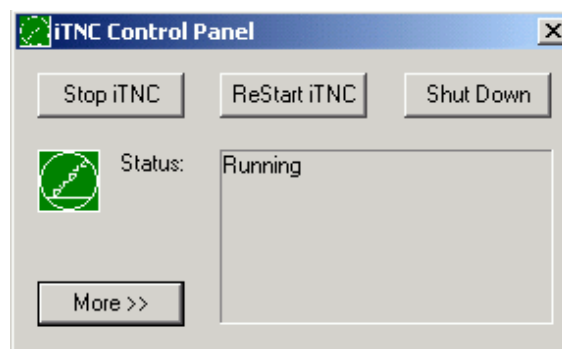
- ▶ Select the function for shutting down and confirm the following dialog prompt again with the YES soft key.

Exiting the programming station temporarily

- ▶ Press the Windows key on the ASCII keyboard for the screen to display the last active application and the Task Bar.
- ▶ Double-click the green HEIDENHAIN symbol to the lower right in the Task Bar for the iTNC Control Panel to appear (see figure at right).



- ▶ Select the function for terminating the iTNC 530 application: Press the **Stop iTNC** button. The iTNC Control Panel remains active. To restart the programming station software, press the **Restart iTNC** button.



1.3 Data Transfer from the Programming Station to the Machine Tool

Prerequisites



Before you can transfer the data, you must connect the machine tool to the PC on which the iTNC programming station has been installed. Refer to the section in the User's Manual about the Ethernet interface.

You can start the data transfer only from the machine tool.

Preparations at the programming station

In order to transfer programs created with the programming station to the machine tool, you must create a transfer directory on the PC, in which you temporarily store the programs to be transferred. Proceed as follows:

- ▶ If such a directory does not yet exist, create either on your PC or on any network drive a new directory (e.g. <c:\pgmtransfer>) using Windows Explorer.
- ▶ Return to the programming station software.
- ▶ Select the Programming and Editing operating mode.
- ▶ To call the file manager, press the PGM MGT key.
- ▶ Select the program you want to transfer to the machine.
- ▶ Select the split-screen view.
- ▶ In the right-hand window select the <c:\pgmtransfer> path.
- ▶ Copy the program to the <pgmtransfer> directory.



Refer to the TNC User's Manual for more information about copying files.

Calling a program from the machine tool

- ▶ Select the Programming and Editing operating mode.
- ▶ To call the file manager, press the PGM MGT key.
- ▶ Select the target directory to which you want to copy the program created with the programming station.
- ▶ Select the split-screen view.
- ▶ In the right-hand window, select the <c:\pgmtransfer> directory on the programming station PC.
- ▶ Select the program to be transferred, and transfer it to the machine tool.



Refer to the TNC User's Manual for more information about copying files.

Connection between the programming station and HEIDENHAIN PC software

To be able to set up a connection between HEIDENHAIN PC software (e.g. TNCremo NT or CycleDesign) and the programming station, the programming station has a special IP address, the **loop-back IP**. This loop-back IP is **127.0.0.1** and must be entered in the appropriate configuration menu of the HEIDENHAIN PC software. Then you can set up a connection with the programming station, as if you were connected with a machine.



1.4 The Demo Version

General Information

After you have installed the iTNC 530 programming station on your PC, you can call the TNC features with your standard keyboard for test purposes. All features of the iTNC 530 are available. As an alternative, you can also use the virtual keyboard for the demo version (see "Showing the virtual keyboard" on page 28).



The demo version of the iTNC 530 programming station permits you to save up to 100 NC blocks per NC program.

When extracting contours from DXF files, the TNC saves no more than 50 NC blocks in the corresponding contour program. The same constraint applies to extracting machining positions into a point table.

Keyboard assignment

















The assignment of the TNC's special keys (axis keys and dialog keys) and the TNC number pad are shown in the figure at upper right. The assignment of the screen keys is shown in the **Function of the screen keys** table below. All other TNC functions are called with shortcut commands, which are listed in the **TNC functions** table on the next page.

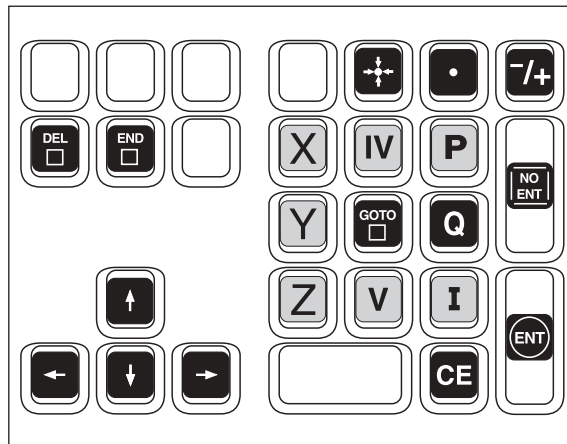
Keys on the TNC screen















Functions for vertical soft keys are available only if you use a special PLC program for the programming station.

To press a vertical soft key, you must also press and hold the **CTRL+ALT+SHIFT** keys.


Function of the screen keys	TNC key	PC key
Horizontal soft key 1		
Horizontal soft key 2		
Horizontal soft key 3		
Horizontal soft key 4		
Horizontal soft key 5		
Horizontal soft key 6		
Horizontal soft key 7		
Horizontal soft key 8		
Shift soft-key row left		
Shift soft-key row right		
Split screen layout		

































Function of the screen keys	TNC key	PC key
Toggle display between machining and programming modes		
Vertical soft key 1		
Vertical soft key 2		
Vertical soft key 3		
Vertical soft key 4		
Vertical soft key 5		





































Keys for TNC functions

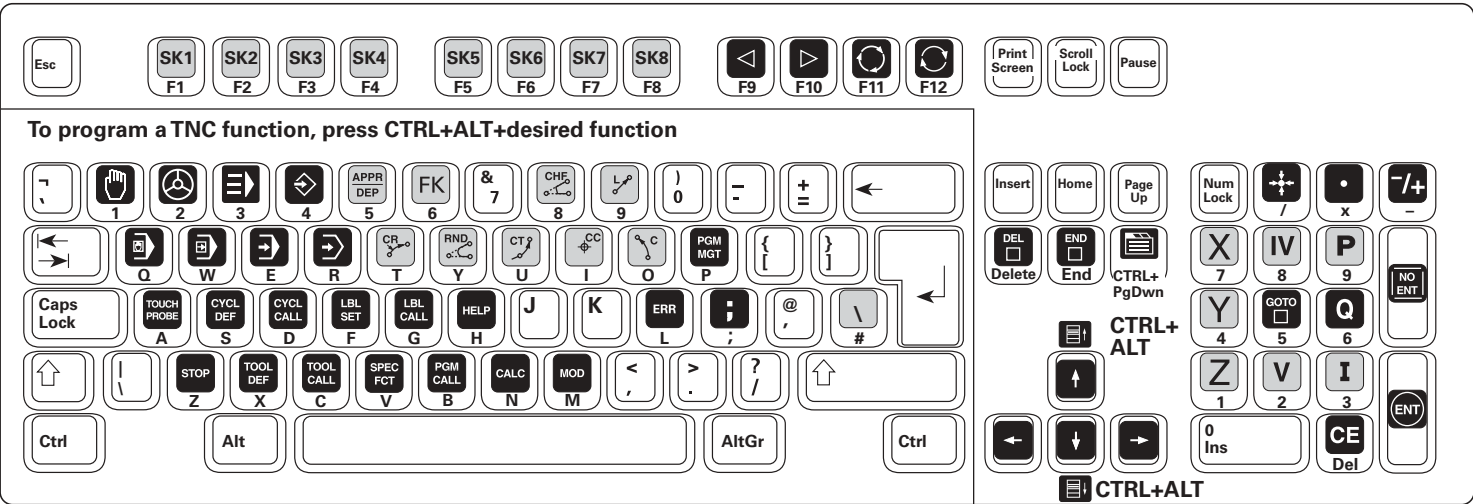


In order to call the TNC functions below with the PC keyboard, you must also press and hold the **CTRL+ALT** keys.

TNC function	TNC key	PC key
Select or delete programs and files, external data transfer		
Pocket calculator		
Select the MOD function.		
Display help text for NC error messages		
Display all current error messages		
Approach/depart contour		
FK free contour programming		
Program a chamfer		
Program a straight line segment		
Program a circular arc with radius		
Program a corner rounding		
Program a circular arc with tangential connection		
Program the circle center/pole for polar coordinates		
Program a circular arc with center		
Select the Manual Operation mode.		

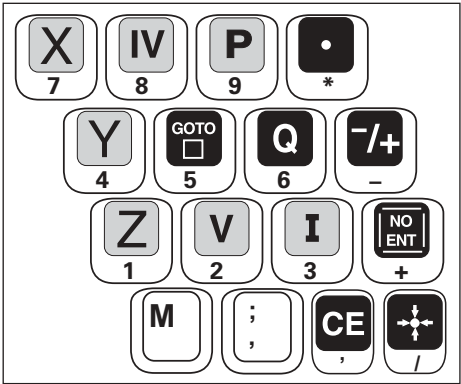


TNC function	TNC key	PC key
Select the smarT.NC operating mode.		
Select the Handwheel operating mode		
Select the Positioning with MDI operating mode		
Select the Program Run, Single Block operating mode		
Select the Program Run, Full Sequence operating mode		
Select the Programming and Editing operating mode.		
Select the Test Run operating mode		
Define touch probe cycles		
Define fixed cycles		
Call fixed cycles		
Define subprograms and program section repeats		
Call subprograms and program section repeats		
Enter a program stop		
Define tool in the program		
Call the tool		
Call the soft-key menu for special functions		
Enter program call		

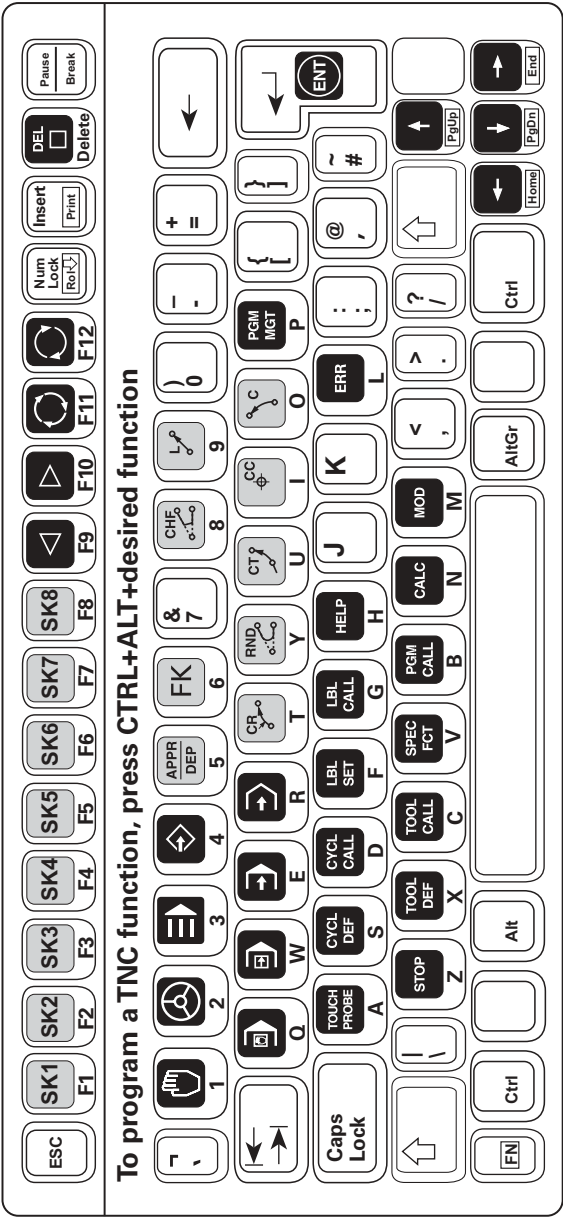


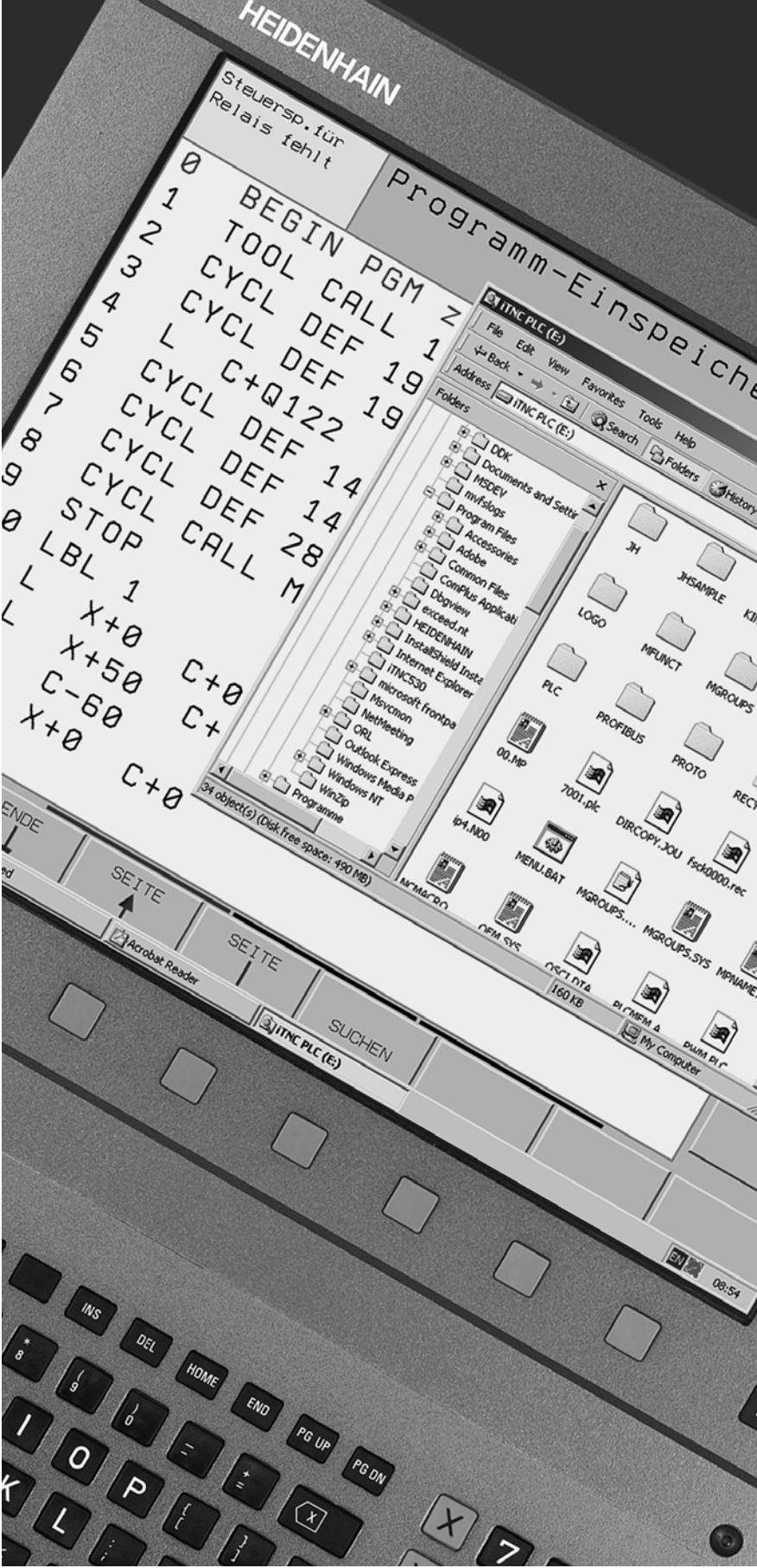
Key assignment on portable computers

On portable computers (laptops, notebooks), the number of available keys is less than on a desktop PC (see figure at right). With the NUM key you activate the numeric keypad, which usually has its own color on your keyboard (see illustration below). Refer to the operating instructions of your laptop/notebook. In addition, some functions such as DEL (delete) or the cursor keys can be activated with an FN key. Refer also to the operating instructions of your laptop/notebook.



On many portable computers there is no ENT key. In most cases you can then continue the dialog with the rightward cursor key.





2

Items Supplied/Installation



2.1 Items Supplied / System Requirements

Items supplied

Two versions of the iTNC programming station are available.

Programming station with TNC keyboard

Included in delivery are:

- The programming station software on CD
- The TE 530 L keyboard for connection to your PC via the USB interface
- 1 USB cable, 2 m
- 4 cable terminals for mounting on the programming station housing
- Self-adhesive stickers for soft-key labels
- User documentation on DVD

Programming station with virtual keyboard

Included in delivery are:

- The programming station software on CD
- USB dongle for surpassing the memory restrictions
- User documentation on DVD

System requirements

The programming station software runs on PCs that meet the following requirements:

- Standard PC with Windows 2000 or Windows XP
- USB interface
- At least 128 MB RAM
- Graphic card: Color depth at least 16 bits
- At least 350 MB available memory on the hard disk
- At least 17-inch screen with a resolution of 1280 x 1024 pixels.



2.2 Putting the Programming Station into Operation

Connect the keyboard for the programming station

Connect the programming station keyboard to an available USB port on your computer. The programming station software automatically recognizes the connected programming station keyboard.

The rear of the programming station keyboard has a USB port, to which you can attach another USB device.



The additional USB device attached to the programming station keyboard may not draw more than 200 mA of current.

Mounting the cable terminals for USB cable

The rear panel of the programming station keyboard has five pockets (see figure at right) in which you can glue the provided cable terminals. The cable terminals serve to relieve the strain from the USB connector.



HEIDENHAIN recommends using these cable terminals to prevent loosening of the USB connector and damage to the connector on the PCB.

To fasten the cable terminals, proceed as follows:

- ▶ Lay the programming station keyboard with the keys downward on a flat surface.
- ▶ Remove the protective film on the bottom of the cable terminal.
- ▶ Place the cable terminal into the desired pocket on the programming station keyboard and press:
 - Use pockets 1, 2 or 5 if the cable is to exit to one side.
 - Use pockets 3 and 5 if the cable is to exit straight out.
- ▶ Connect the USB cable and insert it in the cable terminal.
- ▶ Place the fastener from above on the terminal and press downward until the USB cable is fastened.



Number stickers

The self-adhesive number stickers included with the programming station keyboard enable the user to recognize which manually pressed soft key belongs to which soft key displayed on the screen.

You can apply the contiguous sticker to the programming station keyboard and the individual numbers to the PC screen.

Connecting the USB dongle (only for the version with a virtual keyboard)

Connect the USB dongle to an available USB port on your computer. The programming station software detects the connected dongle automatically.



Installing the programming station software



Note that the following files are normally overwritten during installation of an update:

- Configuration file **OEM.SYS**
- Machine parameter file **DEMO.MP**
- All example NC programs in the **TNC:\DEMO** directory

Should you have made changes to any of these files, make a backup copy before installation, or rename the files.

- ▶ Place the programming station CD in the CD-ROM drive.
- ▶ The CD browser starts automatically if your CD-ROM drive has been appropriately configured. If Autostart is not active, run the <Start.exe> file.
- ▶ Choose the language to be used for the CD navigation.
- ▶ Select <Install software>.
- ▶ Select <Install iTNC programming station> in order to start the installation program.
- ▶ Follow the installation program instructions.





When installing the iTNC530 programming station for the first time, you can decide whether the HEIDENHAIN basic PLC program for the programming station is to be installed as well. Select the setup type **User** and then **Basic PLC Program or Standard (User)** (see "Working with the HEIDENHAIN basic PLC program" on page 45).

You can install this programming station version on your PC even if another version is already installed on your PC, but you cannot start both versions at the same time.



Set the dialog language

The default language for the programming station software is German. Follow the procedure below to switch to another dialog language:

- ▶ Start the programming station software.
 -  Select the Programming and Editing mode of operation.
-  Select the MOD function.
 - ▶ Enter the code number 123 and confirm with the ENT key.
 - ▶ To select machine parameter 7230: Press the GOTO key, enter 7230, and confirm with the ENT key.
 - ▶ Set the desired dialog language (see the table below), and confirm with the END key.

Dialog language	Value from MP7230
English	0
German	1
Czech	2
French	3
Italian	4
Spanish	5
Portuguese	6
Swedish	7
Danish	8
Finnish	9
Dutch	10
Polish	11
Hungarian	12
Russian	14
Chinese (simplified)	15
Chinese (traditional)	16
Slovenian	17
Norwegian	18
Slovak	19



Dialog language	Value from MP7230
Latvian	20
Korean	21
Norwegian	22

Working with the HEIDENHAIN basic PLC program

After you have started the programming station with active PLC, the TNC displays the **98 TNC programming station active** error message.

- Press the HELP key to display a list with the most important programming station functions for machine operation.

The following table lists these functions:

Function	Keys
NC start	CTRL+S
NC stop	CTRL+X
Move the active axis in the positive direction.	CTRL+Arrow up
Move the active axis in the negative direction	CTRL+Arrow down
Emergency stop	ALT+X



Use the mouse in combination with the vertical soft-key row to simulate various machine functions of the basic PLC program:

Soft key	Keys
M	<div>■ NC start</div> <div>■ NC stop</div> <div>■ Move axes in positive and negative direction</div> <div>■ Exit the programming station</div>
S	<div>■ Spindle ON</div> <div>■ Spindle OFF</div> <div>■ Incremental spindle jog M3</div> <div>■ Incremental spindle jog M4</div> <div>■ Oriented spindle stop M19</div>
T	<div>■ Acknowledge tool change</div> <div>■ Clamp/release tool</div>
DIAGNOSIS	<div>■ Show PLC error table</div> <div>■ Diagnostic functions</div> <div>■ Status functions</div> <div>■ Documentation for M functions</div>

If you do not want to use the standard axis configuration, then you can select from various kinematics setups available via the MOD function in the **Programming and Editing** operating mode:



- ▶ Select the Programming and Editing mode of operation.
- ▶ Select the MOD function.
- ▶ Enter the keyword **kinematic** and confirm with the ENT key. The TNC displays a pop-up window with various axis configurations.
- ▶ Move to the desired axis configuration with the arrow keys, and select it with the ENT key or SELECT soft key. The TNC resets, and activates the axis configuration you selected.



A

Axis configuration selection ... 46

C

Compatibility ... 11

D

Data transfer ... 30

Demo version ... 32

Keyboard assignment ... 33

Dialog language, setting ... 44

Downward compatibility ... 12

Differences in machining
cycles ... 13, 15

Differences in miscellaneous
functions M ... 23

Differences in other functions ... 24,
25, 26

Differences in Q parameter
programming ... 24

Differences in touch probe
cycles ... 16, 20

I

Installing the software ... 43

Introduction ... 10

Items supplied ... 40

K

Keyboard assignment ... 32

Keyboard, connecting ... 41, 42

Kinematics selection ... 46

P

PLC program ... 45

Programming station software

 exiting ... 29

 installing ... 43

 starting ... 27

S

Software, installing ... 43

System requirements ... 40

T

TNC keyboard, connecting ... 41

U

USB interface ... 40

