HEIDENHAIN

Operating Instructions for Programming Station

TNC 320
TNC 620
TNC 640

NC Software
TNC 320: 340554-06
TNC 620: 340564-04
TNC 640: 340594-02

English (en)
8/2013

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
TNC 320 programming station	340554-06
TNC 620 programming station	340564-04
TNC 640 programming station	340594-02

TNC 320 programming station

The programming station software 340554-06 is fully compatible with the TNC below.

TNC model	NC software number
TNC 320	340551-06

TNC 620 programming station

The programming station software 340564-04 is fully compatible with the TNC below.

TNC model	NC software number
TNC 620	340560-04
TNC 620 E	340561-04
TNC 620	734980-02
TNC 620 E	734981-02

TNC 640 programming station

The programming station software 340594-02 is fully compatible with the TNC below.

TNC model	NC software number
TNC 640	340590-02
TNC 640 E	340591-02

Training and documentation

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

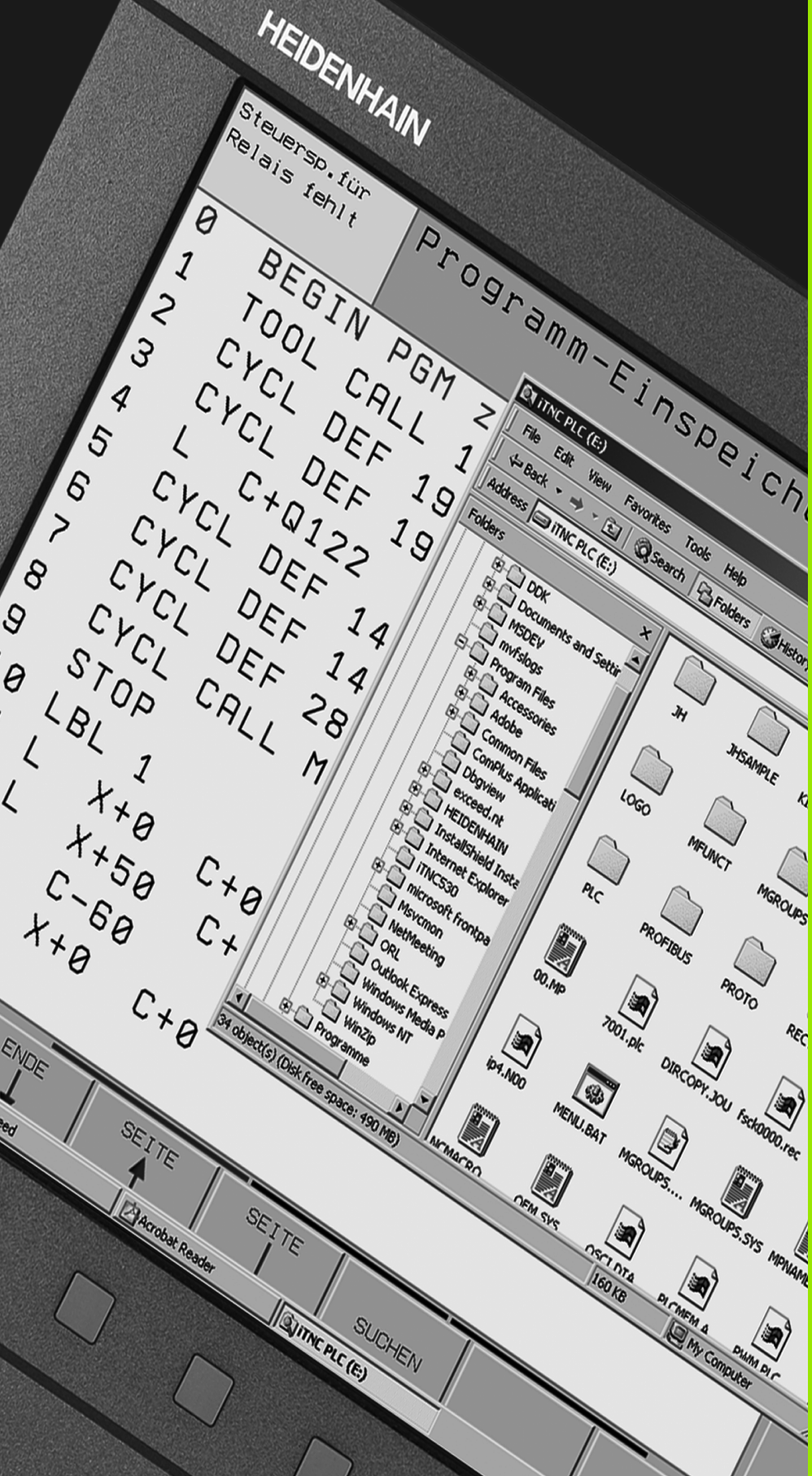
User documentation:

All TNC functions are described in the User's Manuals for the TNC 320, TNC 620 and TNC 640. Please contact HEIDENHAIN if you need a copy of these User's Manuals. Have your NC software number handy. It is shown on the TNC'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 user documentation on the TNCguide DVD (as PDF files) included in delivery.

Contents



1

Important information on the software for the programming station

1.1 Introduction

General information

This manual describes the special features of the TNC 320, TNC 620 and TNC 640 programming stations. All available NC functions are described in the respective User's Manual and the Touch Probe Cycles Manual.

TNC controls from HEIDENHAIN have always been user friendly: Thanks to their simple programming in HEIDENHAIN conversational language, field-proven cycles, unambiguous function keys, and clear and vivid graphic functions, they now count among the most popular shop-floor programmable controls.

Now there is also TNC software for running a programming station using Windows. The HEIDENHAIN keyboard included in delivery is simply connected to the USB port on your PC. For test purposes you can also use the programming station with your standard PC keyboard (see "Special features of the demo version" on page 41).

As an alternative, you can also use the virtual keyboard for the programming station (see "Showing the virtual keyboard" on page 37).

You can also install several programming stations in parallel on your PC. However, it is not possible to use the applications simultaneously. If a programming station is active, close it before starting another programming station.

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 freely available without surcharge on the TNC 320, TNC 620 and TNC 640 programming stations. The free demo version, however, only permits storage of programs up to a certain program length (see "Special features of the demo version" on page 41).

When creating programs with the 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 control, or error messages could appear during testing or program run.

1.2 Compatibility

The programming modes provide the same features as a TNC 320, TNC 620 or TNC 640 connected to a machine tool (NC software: see "TNC model, software and features", page 3). You create programs:

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

You work with the original TNC software—without any compatibility problems. Part programs created with the programming station run on any machine tool equipped with the appropriate control and NC software (see "TNC model, software and features", page 3). 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 must have been enabled on your machine.

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

Downward compatibility

The performance range of the TNC 320, TNC 620 or TNC 640 programming station is exactly the same as the performance range of the respective TNC software (see "TNC model, software and features", page 3). 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:

- Only use functions at the programming station that are available on the 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 have been added (see "Fixed cycles of the iTNC 530, TNC 426/TNC 430, TNC 310 and TNC 410 controls", page 13, or see "Touch-probe cycles of the iTNC 530, TNC 426/TNC 430, TNC 310 and TNC 410 controls", page 19). 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 programming station with the block formatting of your control as it is described in its User's Manual.

Differences in fixed cycles

Fixed cycles of the iTNC 530, TNC 426/TNC 430, TNC 310 and TNC 410 controls

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 340490	TNC 426/ TNC 430 280476	TNC 310 286140	TNC 410 286060
1, Pecking	–	✓	✓	✓	✓
2, Tapping	–	✓	✓	✓	✓
3, Slot milling	–	✓	✓	✓	✓
4, Pocket milling	–	✓	✓	✓	✓
5, Circular pocket	–	✓	✓	✓	✓
6, Rough out (SL I)	–	✓	✓	✓	✓
7, Datum shift	–	✓	✓	✓	✓
8, Mirror image	–	✓	✓	✓	✓
9, Dwell time	–	✓	✓	✓	✓
10, Rotation	–	✓	✓	✓	✓
11, Scaling	–	✓	✓	✓	✓
12, Program call	–	✓	✓	✓	✓
13, Spindle orientation	–	✓	✓	✓	✓
14, Contour definition	–	✓	✓	✓	✓
15, Pilot drilling (SL I)	–	✓	✓	✓	✓
16, Contour milling (SL I)	–	✓	✓	✓	✓
18, Thread cutting	–	✓	✓	x	x
19, Working plane	–	✓	✓	x	x
20, Contour data	–	✓	✓	x	x
21, Pilot drilling	–	✓	✓	x	x
22, Rough-out	Q208 Q401	✓ ✓	– –	x x	x x
23, Floor finishing	–	✓	✓	x	x

Cycle	Additional parameters	iTNC 530 340490	TNC 426/ TNC 430 280476	TNC 310 286140	TNC 410 286060
24, Side finishing	–	✓	✓	x	x
25, Contour train	–	✓	✓	x	x
26, Axis-specific scaling	–	✓	✓	x	✓
27, Cylinder surface	–	✓	✓	x	x
28, Cylinder surface	Q21	✓	–	x	x
29, Cylinder surface ridge	–	✓	x	x	x
30, Run CAM data	–	✓	✓	x	x
32, Tolerance	HSC-MODE TA	✓ ✓	– –	x x	x x
39, Cylinder surface external contour	–	✓	x	x	x
200, Drilling	Q211	✓	✓	–	–
201, Reaming	–	✓	✓	–	–
202, Boring	Q336	✓	✓	–	–
203, Universal drilling	Q256	✓	✓	–	–
204, Back boring	Q336	✓	✓	✓	x
205, Universal pecking	Q379 Q253	✓ ✓	– –	– –	x x
206, Tapping with floating tap holder, new	–	✓	✓	x	x
207, Rigid tapping, new	–	✓	✓	x	x
208, Bore milling	Q342 Q351	✓ ✓	✓ –	x x	x x
209, Tapping with chip breaking	–	✓	✓	x	x
210, Slot with reciprocating plunge	Q338 Q206	✓ ✓	✓ –	– –	– –
211, Circular slot	Q338 Q206	✓ ✓	✓ –	– –	– –
212, Rectangular pocket finishing	–	✓	✓	✓	✓
213, Rectangular stud finishing	–	✓	✓	✓	✓

Cycle	Additional parameters	iTNC 530 340490	TNC 426/ TNC 430 280476	TNC 310 286140	TNC 410 286060
214, Circular pocket finishing	–	✔	✔	✔	✔
215, Circular stud finishing	–	✔	✔	✔	✔
220, Polar pattern	Q301	✔	✔	–	–
	Q365	✔	–	–	–
221, Cartesian pattern	Q301	✔	✔	–	–
225, Engraving	–	✔	–	–	–
230, Multipass milling	–	✔	✔	✔	✔
231, Ruled surface	–	✔	✔	✔	✔
232, Face milling	–	✔	x	x	x
240, Centering	–	✔	x	x	x
241, Single-lip deep-hole drilling	–	✔	x	x	x
247, Datum setting	–	✔	✔	x	x
251, Rectangular pocket (complete)	–	✔	x	x	x
252, Circular pocket (complete)	–	✔	x	x	x
253, Slot milling (complete)	–	✔	x	x	x
254, Circular slot (complete)	–	✔	x	x	x
256, Rectangular stud (complete)	–	✔	x	x	x
257, Circular stud (complete)	–	✔	x	x	x
262, Thread milling	-	✔	✔	x	x
263, Thread milling/counter sinking	–	✔	✔	x	x
264, Thread drilling/milling	–	✔	✔	x	x
265, Helical thread drilling/milling	–	✔	✔	x	x
267, Outside thread milling	–	✔	✔	x	x
270, Contour train data	–	✔	x	x	x
275, Trochoidal slot	–	✔	x	x	x
276, 3-D contour train	–	✔	x	x	x
290, Interpolation turning	–	✔	x	x	x

Fixed cycles of NCK-based TNC 320, TNC 620 and TNC 640 controls

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 320 340551	TNC 620 340560	TNC 640 340590
1, Pecking	–	✓	✓	✓
2, Tapping	–	✓	✓	✓
3, Slot milling	–	✓	✓	✓
4, Pocket milling	–	✓	✓	✓
5, Circular pocket	–	✓	✓	✓
6, Rough out (SL I)	–	x	x	x
7, Datum shift	–	✓	✓	✓
8, Mirror image	–	✓	✓	✓
9, Dwell time	–	✓	✓	✓
10, Rotation	–	✓	✓	✓
11, Scaling	–	✓	✓	✓
12, Program call	–	✓	✓	✓
13, Spindle orientation	–	✓	✓	✓
14, Contour definition	–	✓	✓	✓
15, Pilot drilling (SL I)	–	x	x	x
16, Contour milling (SL I)	–	x	x	x
18, Thread cutting	–	✓	✓	✓
19, Working plane	–	✓	✓	✓
20, Contour data	–	✓	✓	✓
21, Pilot drilling	–	✓	✓	✓
22, Rough-out	Q208 Q401	✓ –	✓ –	✓ –
23, Floor finishing	–	✓	✓	✓
24, Side finishing	–	✓	✓	✓
25, Contour train	–	✓	✓	✓

Cycle	Additional parameters	TNC 320 340551	TNC 620 340560	TNC 640 340590
26, Axis-specific scaling	–	✔	✔	✔
27, Cylinder surface	–	✔	✔	✔
28, Cylinder surface	Q21	✔	✔	✔
29, Cylinder surface ridge	–	✔	✔	✔
30, Run CAM data	–	x	x	x
32, Tolerance	HSC-MODE TA	✔ ✔	✔ ✔	✔ ✔
39, Cylinder surface external contour	–	x	x	x
200, Drilling	Q211	✔	✔	✔
201, Reaming	–	✔	✔	✔
202, Boring	Q336	✔	✔	✔
203, Universal drilling	Q256	✔	✔	✔
204, Back boring	Q336	✔	✔	✔
205, Universal pecking	Q379 Q253	✔ ✔	✔ ✔	✔ ✔
206, Tapping with floating tap holder, new	–	✔	✔	✔
207, Rigid tapping, new	–	✔	✔	✔
208, Bore milling	Q342 Q351	✔ ✔	✔ ✔	✔ ✔
209, Tapping with chip breaking	–	✔	✔	✔
210, Slot with reciprocating plunge	Q338 Q206	✔ ✔	✔ ✔	✔ ✔
211, Circular slot	Q338 Q206	✔ ✔	✔ ✔	✔ ✔
212, Rectangular pocket finishing	–	✔	✔	✔
213, Rectangular stud finishing	–	✔	✔	✔
214, Circular pocket finishing	–	✔	✔	✔
215, Circular stud finishing	–	✔	✔	✔

Cycle	Additional parameters	TNC 320 340551	TNC 620 340560	TNC 640 340590
220, Polar pattern	Q301 Q365	✔ ✔	✔ ✔	✔ ✔
225, Engraving	–	✔	✔	✔
230, Multipass milling	–	✔	✔	✔
231, Ruled surface	–	✔	✔	✔
232, Face milling	–	✔	✔	✔
240, Centering	–	✔	✔	✔
241, Single-lip deep-hole drilling	–	✔	✔	✔
247, Datum setting	–	✔	✔	✔
251, Rectangular pocket (complete)	–	✔	✔	✔
252, Circular pocket (complete)	–	✔	✔	✔
253, Slot milling (complete)	–	✔	✔	✔
254, Circular slot (complete)	–	✔	✔	✔
256, Rectangular stud (complete)	–	✔	✔	✔
257, Circular stud (complete)	–	✔	✔	✔
262, Thread milling	-	✔	✔	✔
263, Thread milling/counter sinking	–	✔	✔	✔
264, Thread drilling/milling	–	✔	✔	✔
265, Helical thread drilling/milling	–	✔	✔	✔
267, Outside thread milling	–	✔	✔	✔
270, Contour train data	–	x	x	x
275, Trochoidal slot	–	x	x	x
276, 3-D contour train	–	x	x	x
290, Interpolation turning	–	x	x	x

Differences in touch probe cycles

Touch-probe cycles of the iTNC 530, TNC 426/TNC 430, TNC 310 and TNC 410 controls

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 340490	TNC 426/ TNC 430 280476	TNC 310 286140	TNC 410 286060
0, Reference plane	–	✓	✓	x	✓
1, Polar datum	–	✓	✓	x	x
2, Calibrate TS	–	✓	✓	x	x
3, Measuring	MB	✓	–	x	x
	REFERENCE SYSTEM	✓	–	–	–
4, Measuring in 3-D	–	✓	x	x	x
9, Calibrate TS length	–	✓	x	x	x
400, Basic rotation	Q305	✓	–	x	x
401, Rotation of 2 holes	Q305	✓	–	x	x
	Q402	✓	–	x	x
	Q337	✓	–	x	x
402, Rotation of 2 studs	Q305	✓	–	x	x
	Q402	✓	–	x	x
	Q337	✓	–	x	x
403, Rotation in rotary axis	Q337	✓	–	x	x
	Q305	✓	–	x	x
	Q303	✓	–	x	x
	Q380	✓	–	x	x
404, Set basic rotation	–	✓	x	x	x
405, Rotation in C axis	–	✓	x	x	x
408, Slot center reference point	–	✓	x	x	x
409, Ridge center reference point	–	✓	x	x	x

Cycle	Additional parameters	iTNC 530 340490	TNC 426/ TNC 430 280476	TNC 310 286140	TNC 410 286060
410, Datum inside rectangle	Q303	✔	–	x	x
	Q381	✔	–	x	x
	Q382	✔	–	x	x
	Q383	✔	–	x	x
	Q384	✔	–	x	x
	Q333	✔	–	x	x
411, Datum outside rectangle	Q303	✔	–	x	x
	Q381	✔	–	x	x
	Q382	✔	–	x	x
	Q383	✔	–	x	x
	Q384	✔	–	x	x
	Q333	✔	–	x	x
412, Datum inside circle	Q303	✔	–	x	x
	Q381	✔	–	x	x
	Q382	✔	–	x	x
	Q383	✔	–	x	x
	Q384	✔	–	x	x
	Q333	✔	–	x	x
	Q423	✔	–	x	x
	Q365	✔	–	x	x
413, Datum outside circle	Q303	✔	–	x	x
	Q381	✔	–	x	x
	Q382	✔	–	x	x
	Q383	✔	–	x	x
	Q384	✔	–	x	x
	Q333	✔	–	x	x
	Q423	✔	–	x	x
	Q365	✔	–	x	x

Cycle	Additional parameters	iTNC 530 340490	TNC 426/ TNC 430 280476	TNC 310 286140	TNC 410 286060
414, Datum outside corner	Q303	✔	–	x	x
	Q381	✔	–	x	x
	Q382	✔	–	x	x
	Q383	✔	–	x	x
	Q384	✔	–	x	x
	Q333	✔	–	x	x
415, Datum inside corner	Q303	✔	–	x	x
	Q381	✔	–	x	x
	Q382	✔	–	x	x
	Q383	✔	–	x	x
	Q384	✔	–	x	x
	Q333	✔	–	x	x
416, Datum circle center	Q303	✔	–	x	x
	Q381	✔	–	x	x
	Q382	✔	–	x	x
	Q383	✔	–	x	x
	Q384	✔	–	x	x
	Q333	✔	–	x	x
	Q320	✔	–	x	x
417, Datum in TS axis	Q303	✔	–	x	x
418, Datum at center of four holes	Q303	✔	–	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
420, Measure angle	Q423	✔	–	x	x
	Q365	✔	–	x	x
421, Measure hole	Q423	✔	–	x	x
	Q365	✔	–	x	x

Cycle	Additional parameters	iTNC 530 340490	TNC 426/ TNC 430 280476	TNC 310 286140	TNC 410 286060
422, Measure circle outside	Q423	✔	–	x	x
	Q365	✔	–	x	x
423, Measure rectangle inside	–	✔	✔	x	x
424, Measure rectangle outside	–	✔	✔	x	x
425, Measure inside width	Q301	✔	–	–	–
	Q320	✔	–	–	–
426, Measure ridge width	–	✔	✔	x	x
427, Measure coordinate	–	✔	✔	x	x
430, Measure bolt hole circle	–	✔	✔	x	x
431, Measure plane	–	✔	✔	x	x
440, Measure axis shift	–	✔	x	x	x
441, Fast probing	–	✔	x	x	x
450, Save kinematics	–	✔	x	x	x
451, Measure kinematics	–	✔	x	x	x
452, Preset compensation	–	✔	x	x	x
460, Calibrate touch probe	–	x	x	x	x
461, Calibrate touch probe length	–	x	x	x	x
462, Calibrate touch probe inside radius	–	x	x	x	x
463, Calibrate touch probe outside radius	–	x	x	x	x
480, Calibrate TT	–	✔	✔	x	x
481, Calibrate tool length	–	✔	✔	x	x
482, Calibrate tool radius	–	✔	✔	x	x
483, Measure tool	–	✔	✔	x	x
484, Calibrate infrared TT	–	✔	x	x	x

Touch probe cycles of NCK-based TNC 320, TNC 620 and TNC 640 controls

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 320 340551	TNC 620 340560	TNC 640 340590
0, Reference plane	–	✓	✓	✓
1, Polar datum	–	✓	✓	✓
2, Calibrate TS	–	x	x	x
3, Measuring	MB	✓	✓	✓
	REFERENCE SYSTEM	✓	✓	✓
4, Measuring in 3-D	–	x	x	x
9, Calibrate TS length	–	x	x	x
400, Basic rotation	Q305	✓	✓	✓
401, Rotation of 2 holes	Q305	✓	✓	✓
	Q402	✓	✓	✓
	Q337	✓	✓	✓
402, Rotation of 2 studs	Q305	✓	✓	✓
	Q402	✓	✓	✓
	Q337	✓	✓	✓
403, Rotation in rotary axis	Q337	✓	✓	✓
	Q305	✓	✓	✓
	Q303	✓	✓	✓
	Q380	✓	✓	✓
404, Set basic rotation	–	✓	✓	✓
405, Rotation in C axis	–	✓	✓	✓
408, Slot center reference point	–	✓	✓	✓
409, Ridge center reference point	–	✓	✓	✓

Cycle	Additional parameters	TNC 320 340551	TNC 620 340560	TNC 640 340590
410, Datum inside rectangle	Q303	✓	✓	✓
	Q381	✓	✓	✓
	Q382	✓	✓	✓
	Q383	✓	✓	✓
	Q384	✓	✓	✓
	Q333	✓	✓	✓
411, Datum outside rectangle	Q303	✓	✓	✓
	Q381	✓	✓	✓
	Q382	✓	✓	✓
	Q383	✓	✓	✓
	Q384	✓	✓	✓
	Q333	✓	✓	✓
412, Datum inside circle	Q303	✓	✓	✓
	Q381	✓	✓	✓
	Q382	✓	✓	✓
	Q383	✓	✓	✓
	Q384	✓	✓	✓
	Q333	✓	✓	✓
	Q423	✓	✓	✓
	Q365	✓	✓	✓
413, Datum outside circle	Q303	✓	✓	✓
	Q381	✓	✓	✓
	Q382	✓	✓	✓
	Q383	✓	✓	✓
	Q384	✓	✓	✓
	Q333	✓	✓	✓
	Q423	✓	✓	✓
	Q365	✓	✓	✓

Cycle	Additional parameters	TNC 320 340551	TNC 620 340560	TNC 640 340590
414, Datum outside corner	Q303	✓	✓	✓
	Q381	✓	✓	✓
	Q382	✓	✓	✓
	Q383	✓	✓	✓
	Q384	✓	✓	✓
	Q333	✓	✓	✓
415, Datum inside corner	Q303	✓	✓	✓
	Q381	✓	✓	✓
	Q382	✓	✓	✓
	Q383	✓	✓	✓
	Q384	✓	✓	✓
	Q333	✓	✓	✓
416, Datum circle center	Q303	✓	✓	✓
	Q381	✓	✓	✓
	Q382	✓	✓	✓
	Q383	✓	✓	✓
	Q384	✓	✓	✓
	Q333	✓	✓	✓
	Q320	✓	✓	✓
417, Datum in TS axis	Q303	✓	✓	✓
418, Datum at center of four holes	Q303	✓	✓	✓
	Q381	✓	✓	✓
	Q382	✓	✓	✓
	Q383	✓	✓	✓
	Q384	✓	✓	✓
	Q333	✓	✓	✓
419, Datum in one axis	–	✓	✓	✓
420, Measure angle	–	✓	✓	✓
421, Measure hole	Q423	✓	✓	✓
	Q365	✓	✓	✓
422, Measure circle outside	Q423	✓	✓	✓
	Q365	✓	✓	✓

Cycle	Additional parameters	TNC 320 340551	TNC 620 340560	TNC 640 340590
423, Measure rectangle inside	–	✔	✔	✔
424, Measure rectangle outside	–	✔	✔	✔
425, Measure inside width	Q301 Q320	✔ ✔	✔ ✔	✔ ✔
426, Measure ridge width	–	✔	✔	✔
427, Measure coordinate	–	✔	✔	✔
430, Measure bolt hole circle	–	✔	✔	✔
431, Measure plane	–	✔	✔	✔
440, Measure axis shift	–	x	x	x
441, Fast probing	–	x	x	x
450, Save kinematics	–	x	✔	✔
452, Preset compensation	–	x	✔	✔
460, Calibrate touch probe	–	✔	✔	✔
461, Calibrate touch probe length	–	✔	✔	✔
462, Calibrate touch probe inside radius	–	✔	✔	✔
463, Calibrate touch probe outside radius	–	✔	✔	✔
480, Calibrate TT	–	✔	✔	✔
481, Calibrate tool length	–	✔	✔	✔
482, Calibrate tool radius	–	✔	✔	✔
483, Measure tool	–	✔	✔	✔
484, Calibrate infrared TT	–	x	x	x

Differences in miscellaneous functions M

Miscellaneous functions M of the iTNC 530, TNC 426/TNC 430, TNC 310 and TNC 410 controls

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 340490	TNC 426/ TNC 430 280476	TNC 310 286140	TNC 410 286060
Two miscellaneous functions can be programmed in one NC block	✓	✓	x	✓
M00	✓	✓	✓	✓
M01	✓	✓	✓	✓
M02	✓	✓	✓	✓
M03, M04 / M05	✓	✓	✓	✓
M06	✓	✓	✓	✓
M08/M09	✓	✓	✓	✓
M13, M14	✓	✓	✓	✓
M30	✓	✓	✓	✓
M89	✓	✓	✓	✓
M90	✓	✓	✓	✓
M91	✓	✓	✓	✓
M92	✓	✓	✓	✓
M94	✓	✓	✓	✓
M97	✓	✓	✓	✓
M98	✓	✓	✓	✓
M99	✓	✓	✓	✓
M101 / M102	✓	✓	x	✓
M103	✓	✓	x	✓
M104	✓	✓	x	x
M105	✓	✓	x	x
M107 / M108	✓	✓	x	x
M109, M110 / M111	✓	✓	x	✓

M function	iTNC 530 340490	TNC 426/ TNC 430 280476	TNC 310 286140	TNC 410 286060
M112 / M113	✓	✓	x	✓
M114 / M115	✓	✓	x	x
M116 / M117	✓	✓	x	x
M118	✓	✓	x	x
M120	✓	✓	x	✓
M124	✓	✓	x	✓
M126 / M127	✓	✓	x	✓
M128 / M129	✓	✓	x	x
M130	✓	✓	x	x
M134 / M135	✓	✓	x	x
M136 / M137	✓	✓	x	x
M138	✓	✓	x	x
M140	✓	✓	x	x
M141	✓	✓	x	x
M142	✓	✓	x	x
M143	✓	✓	x	x
M144/M145	✓	✓	x	x
M148 / M149	✓	✓	x	x
M150	✓	✓	x	x
M197	x	x	x	x
M200 - M204	✓	✓	x	x

Miscellaneous functions M of NCK-based TNC 320, TNC 620 and TNC 640 controls

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 320 340551	TNC 620 340560	TNC 640 340590
Two miscellaneous functions can be programmed in one NC block	✓	✓	✓
M00	✓	✓	✓
M01	✓	✓	✓
M02	✓	✓	✓
M03, M04 / M05	✓	✓	✓
M06	✓	✓	✓
M08/M09	✓	✓	✓
M13, M14	✓	✓	✓
M30	✓	✓	✓
M89	✓	✓	✓
M90	x	x	x
M91	✓	✓	✓
M92	✓	✓	✓
M94	✓	✓	✓
M97	✓	✓	✓
M98	✓	✓	✓
M99	✓	✓	✓
M101 / M102	✓	✓	✓
M103	✓	✓	✓
M104	x	x	x
M105	x	x	x
M107 / M108	✓	✓	✓
M109, M110 / M111	✓	✓	✓
M112 / M113	x	x	x
M114 / M115	x	x	x

M function	TNC 320 340551	TNC 620 340560	TNC 640 340590
M116 / M117	✔	✔	✔
M118	✔	✔	✔
M120	✔	✔	✔
M124	x	x	x
M126 / M127	✔	✔	✔
M128 / M129	x	✔	✔
M130	✔	✔	✔
M134 / M135	x	x	x
M136 / M137	✔	✔	✔
M138	x	✔	✔
M140	✔	✔	✔
M141	✔	✔	✔
M142	x	x	x
M143	✔	✔	✔
M144/M145	✔	✔	✔
M148 / M149	✔	✔	✔
M197	✔	✔	✔
M143	✔	✔	✔
M200 - M204	x	x	x

Differences in Q-parameter programming

Q-parameter functions of the iTNC 530, TNC 426/TNC 430, TNC 310 and TNC 410 controls

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 340490	TNC 426/ TNC 430 280476	TNC 310 286140	TNC 410 286060
Formula entry: **SGN**	✓	x	✓	✓
Formula entry: **%**	✓	x	x	x
FN14	✓	✓	✓	✓
FN15	✓	✓	✓	✓
FN16	✓	✓	x	x
FN19	✓	✓	✓	✓
FN20	✓	✓	x	x
FN23	✓	✓	x	x
FN24	✓	✓	x	x
FN25	✓	✓	x	x
FN26	✓	✓	x	x
FN27	✓	✓	x	x
FN28	✓	✓	x	x
FN29	x	x	x	x
Contour formula **QC...**	✓	x	x	x
String formula **QS...**	✓	x	x	x
SQL commands	x	x	x	x
Local parameters **QL...**	✓	x	x	x
Nonvolatile parameters **QR...**	✓	x	x	x

Q-parameter functions of NCK-based TNC 320, TNC 620 and TNC 640 controls

Meaning of the symbols used in the table:

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

Function	TNC 320 340551	TNC 620 340560	TNC 640 340590
Formula entry: **SGN**	✓	✓	✓
Formula entry: **%**	✓	✓	✓
FN14	✓	✓	✓
FN15	x	x	x
FN16	✓	✓	✓
FN19	✓	✓	✓
FN20	✓	✓	✓
FN23	✓	✓	✓
FN24	✓	✓	✓
FN25	x	x	x
FN26	x	x	x
FN27	x	x	x
FN28	x	x	x
FN29	✓	✓	✓
Contour formula **QC...**	x	x	x
String formula **QS...**	✓	✓	✓
SQL commands	✓	✓	✓
Local parameters **QL...**	✓	✓	✓
Nonvolatile parameters **QR...**	✓	✓	✓

Differences in other functions

Other functions of the iTNC 530, TNC 426/TNC 430, TNC 310 and TNC 410 controls

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 340490	TNC 426/ TNC 430 280476	TNC 310 286140	TNC 410 286060
CYCL CALL PAT	✓	✓	x	x
CYCL CALL POS	✓	x	x	x
Cutting data tables (**WMAT** block)	✓	✓	x	x
Feed rate **F** in the **CHF** block	✓	✓	✓	✓
Contour definition **DECLARE CONTOUR**	✓	x	x	x
Tilting the working plane: **PLANE**	✓	x	x	x
Positioning behavior of rotary axes **TCPM FUNCTION**	✓	x	x	x
FK programming: **FL P2X/FLT P2X**	✓	✓	x	x
FK programming: **FL P2Y/FLT P2Y**	✓	✓	x	x
FK programming: **FL RX/FLT RX**	✓	✓	x	x
FK programming: **FL RY/FLT RY**	✓	✓	x	x
FK programming: **FL RPR/FLT RPR**	✓	✓	x	x
FK programming: **FL RPA/FLT RPA**	✓	✓	x	x
FK programming: **FL RAN**	✓	✓	x	x
FK programming: **FC LEN/FCT LEN**	✓	✓	x	x
FK programming: **FC AN/FCT AN**	✓	✓	x	x
FK programming: **FC P2X/FCT P2X**	✓	✓	x	x
FK programming: **FC P2Y/FCT P2Y**	✓	✓	x	x
FK programming: **FC P3X/FCT P3X**	✓	✓	x	x
FK programming: **FC P3X/FCT P3Y**	✓	✓	x	x
FK programming: **FC RX/FCT RX**	✓	✓	x	x
FK programming: **FC RY/FCT RY**	✓	✓	x	x
FK programming: **FC RPR/FCT RPR**	✓	✓	x	x

Function	iTNC 530 340490	TNC 426/ TNC 430 280476	TNC 310 286140	TNC 410 286060
FK programming: **FC RPA/FCT RPA**	✓	✓	x	x
FK programming: **FC RAN/FCT RAN**	✓	✓	x	x
FK programming: **FC RCCX/FCT RCCX**	✓	✓	x	x
FK programming: **FC RCCY/FCT RCCY**	✓	✓	x	x
FK programming: **FC RCCPR/FCT RCCPR**	✓	✓	x	x
FK programming: **FC RCCPA/FCT RCCPA**	✓	✓	x	x

Other functions of NCK-based TNC 320, TNC 620 and TNC 640 controls

Meaning of the symbols used in the table:

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

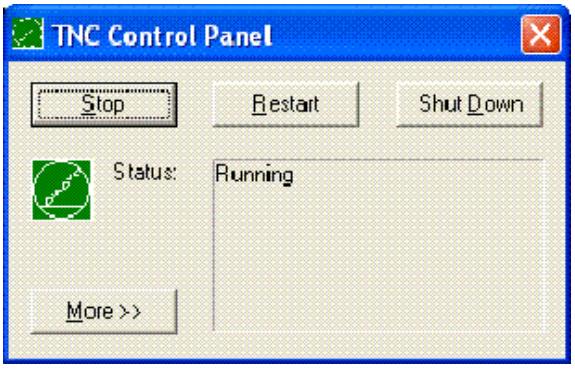
Function	TNC 320 340551	TNC 620 340560	TNC 640 340590
CYCL CALL PAT	✓	✓	✓
CYCL CALL POS	✓	✓	✓
Cutting data tables (**WMAT** block)	x	x	x
Feed rate **F** in the **CHF** block	✓	✓	✓
Contour definition **DECLARE CONTOUR**	✓	✓	✓
Tilting the working plane: **PLANE**	✓	✓	✓
Positioning behavior of rotary axes **TCPM FUNCTION**	x	✓	✓
FK programming: **FL P2X/FLT P2X**	✓	✓	✓
FK programming: **FL P2Y/FLT P2Y**	✓	✓	✓
FK programming: **FL RX/FLT RX**	✓	✓	✓
FK programming: **FL RY/FLT RY**	✓	✓	✓
FK programming: **FL RPR/FLT RPR**	✓	✓	✓
FK programming: **FL RPA/FLT RPA**	✓	✓	✓
FK programming: **FL RAN**	✓	✓	✓
FK programming: **FC LEN/FCT LEN**	✓	✓	✓
FK programming: **FC AN/FCT AN**	✓	✓	✓
FK programming: **FC P2X/FCT P2X**	✓	✓	✓

Function	TNC 320 340551	TNC 620 340560	TNC 640 340590
FK programming: **FC P2Y/FCT P2Y**	✔	✔	✔
FK programming: **FC P3X/FCT P3X**	✔	✔	✔
FK programming: **FC P3X/FCT P3Y**	✔	✔	✔
FK programming: **FC RX/FCT RX**	✔	✔	✔
FK programming: **FC RY/FCT RY**	✔	✔	✔
FK programming: **FC RPR/FCT RPR**	✔	✔	✔
FK programming: **FC RPA/FCT RPA**	✔	✔	✔
FK programming: **FC RAN/FCT RAN**	✔	✔	✔
FK programming: **FC RCCX/FCT RCCX**	✔	✔	✔
FK programming: **FC RCCY/FCT RCCY**	✔	✔	✔
FK programming: **FC RCCPR/FCT RCCPR**	✔	✔	✔
FK programming: **FC RCCPA/FCT RCCPA**	✔	✔	✔

1.3 Working with the programming station

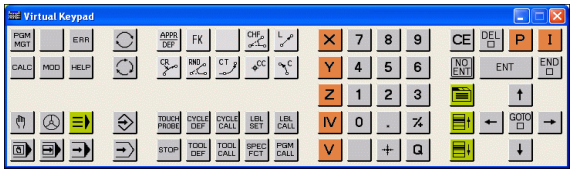
Starting the programming station

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



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

Showing the virtual keyboard



Your active screen resolution must be higher than 1024x768 in order to work effectively with the virtual keyboard.

You use the mouse in combination with the virtual keyboard to simulate keys available on the TNC keyboard. You can activate the soft keys directly with the mouse in the TNC window, or with the F1 to F8 function keys on the PC keyboard. The keys normally available on the TNC screen housing are also operated via the function keys on the PC keyboard:

- 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

Proceed as follows to show the virtual keyboard:

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

If you want to automatically show the keyboard when starting the programming station software, proceed as follows:

- Start the programming station software
- Double-click the green HEIDENHAIN symbol to the lower right in the taskbar 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 **OK**

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

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

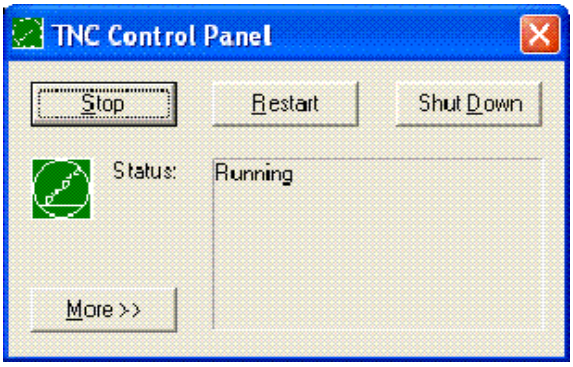
- Select the Manual Operation mode
- Shift the soft-key row until the soft key for shutting down the system appears

- Select the function for shutting down and confirm the following dialog prompt 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 taskbar
- Double-click the green HEIDENHAIN symbol to the lower right in the taskbar for the TNC Control Panel to appear (see figure at right)

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



1.4 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 TNC 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 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
- 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
- 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 or CycleDesign) and the programming station, the programming station has a special IP address, the so-called **loop-back IP**. Use the **local (TCP/IP localhost)** type of connection or enter the loop-back IP **127.0.0.1** in the corresponding 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.

1.5 Special features of the demo version

General information

After you have installed the programming station on your PC, you can call the TNC features with your standard PC keyboard for test purposes. All features of the respective TNC are available. As an alternative, you can also use the virtual keyboard for the demo version (see "Showing the virtual keyboard" on page 37).

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

Starting the demo version

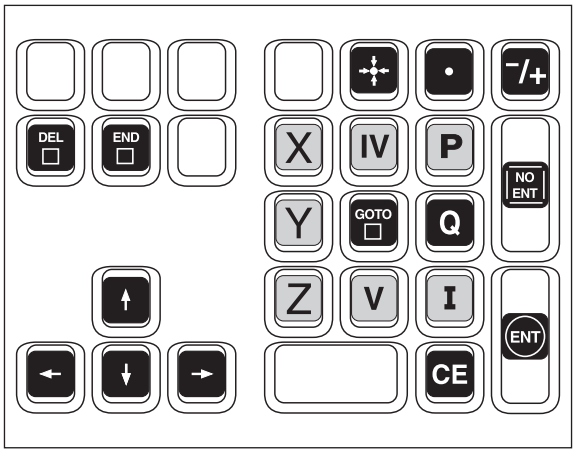
After you have started the demo version of the programming station using the Windows Start menu, a pop-up window appears and informs you about the special features of the demo version. In this case, proceed as follows:

- To confirm the information about the demo version, press the ENT key on the TNC's virtual keyboard. If you are using the standard PC keyboard, press the ENTER key of your keyboard. Then the TNC software closes the pop-up window and displays the **Power interrupted** message.
- To confirm the power interruption, press the CE key on the TNC keyboard again. If you are using the standard PC keyboard, press the comma (,) key of your keyboard. If you are using a laptop, activate the NUM LOCK function and then press the period (.) key in the numeric keypad of your keyboard. The programming station software is now ready for use in the Manual Operation mode.

There is more information on the keyboard assignment (see "Keyboard assignment" on page 42).

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.

Function of the screen keys	TNC key	PC key
Shift soft-key row left	◁	F9
Shift soft-key row right	▷	F10
Select split screen layout		F11
Toggle the display between machining and programming modes		F12
Horizontal soft key 1		F1
Horizontal soft key 2		F2
Horizontal soft key 3		F3
Horizontal soft key 4		F4
Horizontal soft key 5		F5
Horizontal soft key 6		F6
Horizontal soft key 7		F7
Horizontal soft key 8		F8

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 of the PC keyboard.

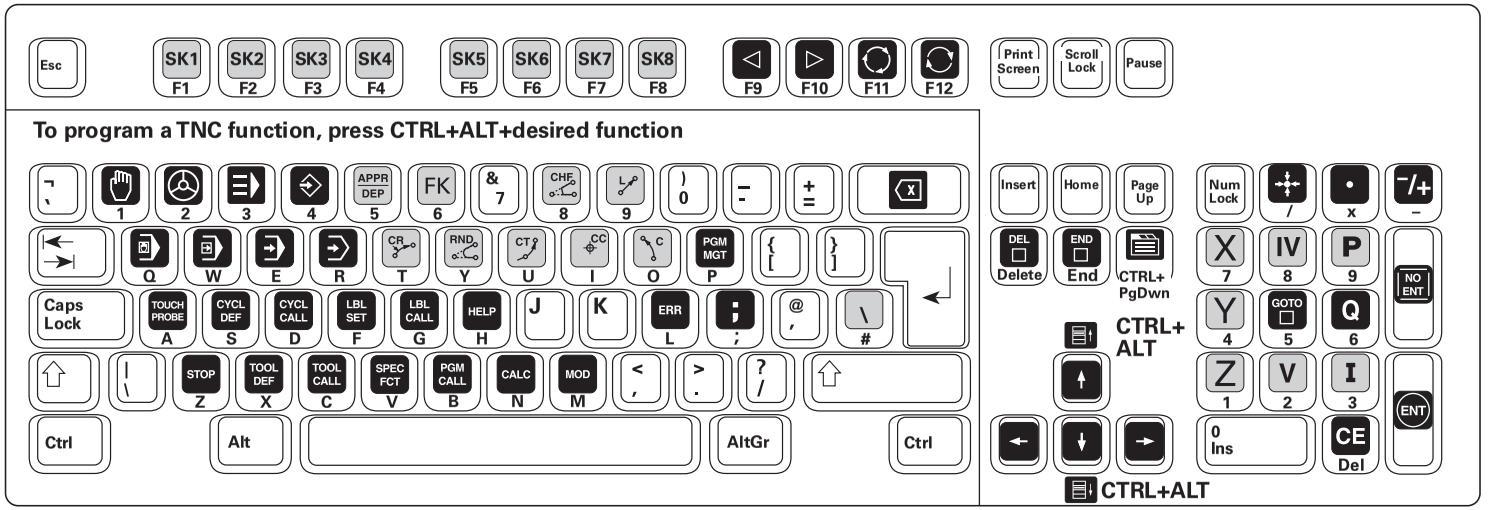
TNC function	TNC key	PC key
Select or delete programs and files, external data transfer	PGM MGT	P
Show calculator	CALC	N
Select MOD functions	MOD	M
Display help text for NC error messages	HELP	H
Display all current error messages	ERR	L
Approach/depart contour	APPR DEP	% 5
FK free contour programming	FK	& 6
Program a chamfer	CHF	(8 [
Program a straight line segment	L	) 9]
Program a circular arc with radius	CR	T
Program a corner rounding	RND	Z
Program a circular arc with tangential connection	CT	U
Program the circle center/pole for polar coordinates	CC	I
Program a circular arc with center	C	O
Select the Manual Operation mode		! 1

TNC function	TNC key	PC key
Select the smarT.NC mode (not for TNC 320, TNC 620 and TNC 640)		§ 3
Select the Handwheel operating mode		" 2
Select the Positioning with MDI operating mode		Q
Select the Program Run, Single Block operating mode		W
Select the Program Run, Full Sequence operating mode		E
Select the Programming and Editing operating mode.		$ 4
Select the Test Run operating mode		R
Define touch probe cycles	TOUCH PROBE	A
Define fixed cycles	CYCL DEF	S
Call fixed cycles	CYCL CALL	D
Define subprograms and program section repeats	LBL SET	F
Call subprograms and program section repeats	LBL CALL	G
Enter a program stop	STOP	Y
Define tool in the program	TOOL DEF	X
Call the tool	TOOL CALL	C
Call the soft-key menu for special functions	SPEC FCT	V
Enter program call	PGM CALL	B

Keyboard assignment to American PC keys arrangement

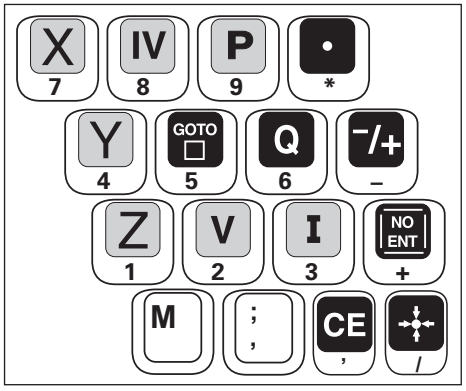
The following keys are arranged differently if you use a PC keyboard with an American key layout.

TNC function	TNC key	PC key
FK free contour programming	FK	^ 6
Program a chamfer	CHF	* 8
Program a straight line segment	L	(9
Select the smarT.NC mode (not for TNC 320, TNC 620 and TNC 640)		# 3
Select the Handwheel operating mode		@ 2
Program a corner rounding	RND	Y
Enter a program stop	STOP	Z

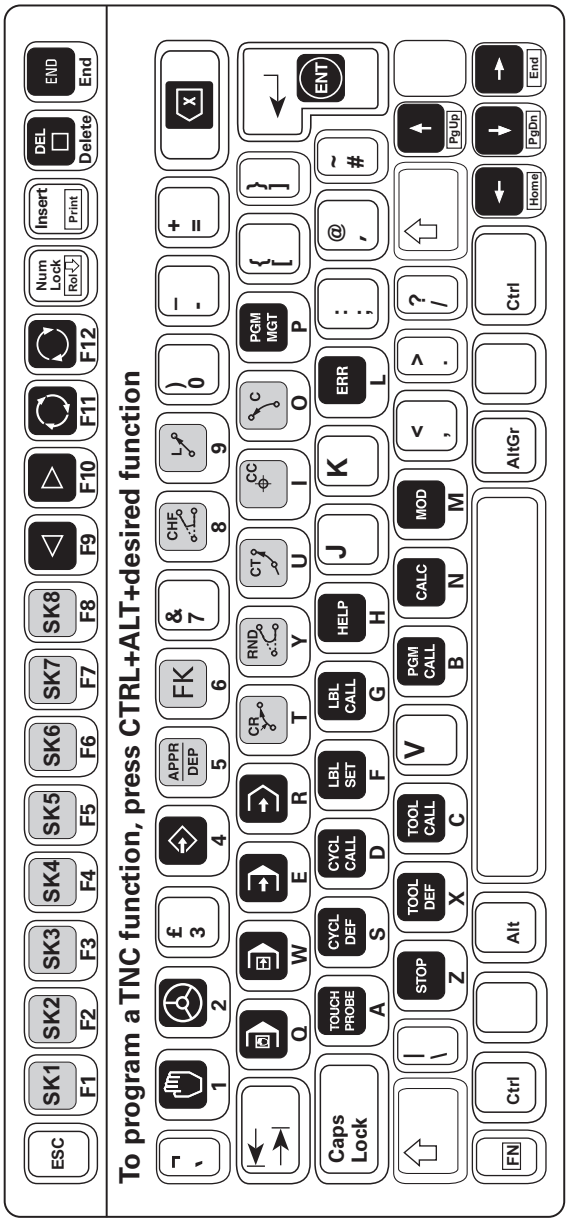

Esc
SK1 F1
SK2 F2
SK3 F3
SK4 F4
SK5 F5
SK6 F6
SK7 F7
SK8 F8
F9
F10
F11
F12
Print Screen
Scroll Lock
Pause
To program a TNC function, press CTRL+ALT+desired function
Insert
Home
Page Up
Num Lock
DEL Delete
END End
CTRL+ PgDwn
NO ENT
CTRL+ ALT
Caps Lock
TOUCH PROBE
CYCL DEF
CYCL CALL
LBL SET
LBL CALL
HELP
ERR
PGM MGT
STOP
TOOL DEF
TOOL CALL
SPEC FCT
PGM CALL
CALC
MOD
GOTO
ENT
0 Ins
CE Del
Ctrl
Alt
AltGr
Ctrl
CTRL+ALT

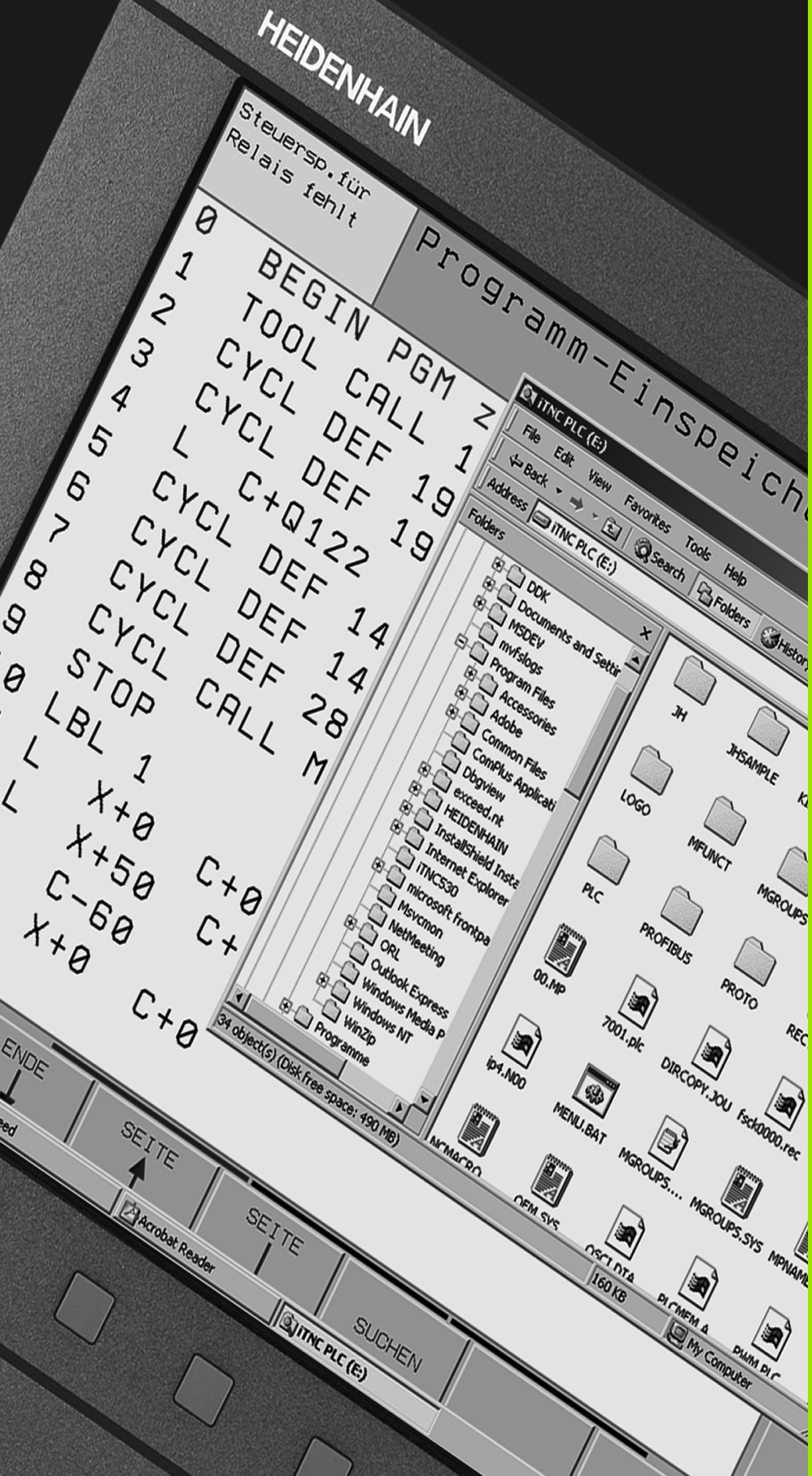
Key assignment on portable computers

On portable computers (laptops, notebooks), the number of available key 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 the 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 the ENT key is not available. 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

Three versions of the TNC programming station are available.

Programming station with TNC keyboard

Included in delivery are:

- The programming station software on CD
- Programming station keyboard for connection to the PC via the USB interface
- 1 USB cable (2 meters)
- 4 cable clamps 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 avoiding the memory restrictions
- User documentation on DVD

Programming station with network license

Included in delivery are:

- The programming station software on CD
- USB dongle with 14 or 20 network licenses for avoiding 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, XP, Vista, 7 or 8 (32-bit and 64-bit) with the recommended system prerequisites
- Processor: at least Pentium III with 800 MHz
- USB interface
- LAN interface for multiple-station version with network dongle
- 512 MB RAM minimum
- Graphics card: At least 16-bit color depth
- Minimum 1 GB of free hard-disk space
- At least 15-inch screen with a resolution of 1024 x 768 pixels; the virtual keyboard requires a minimum resolution of 1280 x 800 (WXGA) pixels, or for the TNC 640 a minimum resolution of 1680 x 1050 (WSXGA+) pixels
- LAN connection when the TNC programming station with network license is used

2.2 Connecting the programming station

Connecting the keyboard for the programming station

Connect the programming station keyboard to an available USB port on your computer.

Then install the driver for the USB dongle from the programming station CD.

The programming station software then 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 must not draw more than 200 mA of current.

Mounting the cable clamps for the 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 clamps. The cable clamps serve to relieve the strain on the USB connector.

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

To fasten the cable clamps, 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 clamp
- Place the cable clamp 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 or 4 if the cable is to exit straight out
- Connect the USB cable and insert it in the cable clamp
- Place the fastener from above on the clamp 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 PC's monitor.

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

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

Connect the USB dongle to an available USB port on your computer.

Then install the driver for the USB dongle from the programming station CD.

The programming station software then detects the connected dongle automatically.

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

Connect the USB network dongle to an available USB port on the computer you want to use as a server.

Then install the driver for the USB network dongle from the programming station CD.

The programming station software on the computer automatically recognizes the connected dongle after the network driver has been installed successfully. You must appropriately configure the computers to which you want to give access to the network dongle after installing the programming station software (see "Configuring the programming station software for use with a USB network dongle" on page 55).

Installing the programming station software

Note that all files are overwritten during installation of an update.

If you have created files or made any changes to files, make backup copies of these before installation.

- 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 TNC programming station> in order to start the installation program
- Follow the installation program instructions

You can install this programming station version on your PC together with another version (e.g. iTNC 530) already installed on your PC. However, it is not possible to use the applications simultaneously. If a programming station is active, close it before starting another programming station.

Configuring the programming station software for use with a USB network dongle

All computers to which you want to give access to a network dongle must be configured in the following way:

- Start the programming station software
- Double-click the green HEIDENHAIN symbol to the lower right in the taskbar for the TNC Control Panel to appear (see figure)
- Click the **More >>** button: Additional settings are displayed
- Click the **Settings** button: Programming station settings are displayed
- Select the **Use Hardlock Server** option
- In the **Server** input field, enter the IP address or the network name of the computer which you are using as a server, i.e. the computer to which you have connected the USB network dongle
- Press **OK** to confirm your settings

Changes do not take effect until you restart the programming station software: Press the **Restart TNC** button.

You do not need to install **any additional drivers** on the client PC to be able to access the USB network dongle.

Network license for multiple workstations

Before you can use a network license for multiple workstations, you must start the server program "SmarxOS" on one of the computers, and connect the USB dongle. This computer is then designated the "dongle server," and monitors the connections to the client PCs on which the programming stations are installed.

Proceed as follows in order to start the "SmarxOS" server program on the dongle server PC:

- Copy the entire "CBServer" directory from the CD (\JH\CBServer) to a local drive on your PC
- Start the server by running "CBIOSSrv.exe"

After the server program has started, the  icon is shown in the taskbar.

The programming station PCs log on to the dongle server when they start. During operation the server checks whether a logged-on programming station is active. If the programming station PC has not contacted the dongle server for some time, the dongle server logs that PC off (default setting: 15 minutes). When you exit the programming station, the respective PC logs off from the dongle server.

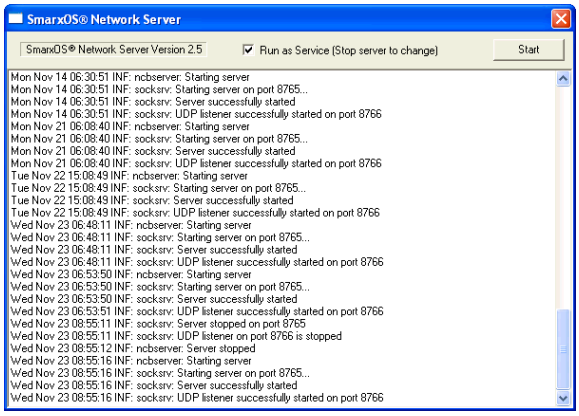
You can also install the "SmarxOS" server program as a service on the dongle server PC. Services are automatically run when Windows is started, and run in the background. That way you don't need to start the server program manually each time you boot the PC. To install the server program as a service, proceed as follows:

- Copy the entire "CBServer" directory from the CD (\JH\CBServer) to a local drive on your PC
- Start the server by running "CBIOSSrv.exe"
- Stop the server by pressing the "Stop" button in the server window
- Select the option "Run as Service"
- Start the server by pressing the "Start" button in the server window

The service is now run automatically when Windows is started. The  icon will not be shown in the taskbar anymore after the next time the system is booted.

You may have to enter the CBIOSSRV.SRV program as an exception in the Windows firewall software.

You can also check and change the settings for the "CBIOS Server" service in the Control Panel under Administrative Tools\Services.



You can make other changes to the server settings with the "Server Administration" program. Start the AdminApp.exe program (directory: CBServer) for this:

- Press the "Search" button to show all PCs to which a dongle is connected
- Click an IP address to add it to the address block for connection settings
- Use the IP address 127.0.0.1 (default setting) to load the data from the local dongle server
- Press the "Connect" button to open a window with information about the dongle and other possible settings (e.g. "Connection timeout")

You will find more information on the Server Administration program in the readme.txt file (directory: CBServer).

Setting the conversational language

When installing the programming station software, you can select either English or German as the conversational language. Follow the procedure below to switch to another conversational language:

- Start the programming station software

- Select the Programming and Editing mode of operation

- Select MOD functions
- Enter the code number 123 and confirm with the ENT key
- To select machine parameter CfgDisplayLanguage: Press the GOTO key, enter CfgDisplayLanguage, and confirm with the ENT key
- Set the desired conversational language (see the table below), and confirm with the END key

Conversational language	Value from
English	ENGLISH
German	GERMAN
Czech	CZECH
French	FRENCH
Italian	ITALIAN
Spanish	SPANISH
Portuguese	PORTUGUESE
Swedish	SWEDISH
Danish	DANISH
Finnish	FINNISH
Dutch	DUTCH
Polish	POLISH
Hungarian	HUNGARIAN
Russian	RUSSIAN
Chinese (simplified)	CHINESE
Chinese (traditional)	CHINESE_TRAD
Slovenian	SLOVENIAN
Norwegian	NORWEGIAN

Conversational language	Value from
Slovak	SLOVAK
Latvian	LATVIAN
Korean	KOREAN
Estonian	ESTONIAN
Turkish	TURKISH
Romanian	ROMANIAN
Lithuanian	LITHUANIAN

Working with the HEIDENHAIN basic PLC program

After you have started the programming station, the most important programming station functions for machine tool operation become available.

The vertical soft-key row includes important soft keys for machine tool operation such as NC start, NC stop, positive or negative traverse direction of axis.

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

Working with a machine-specific PLC program

Some machine tool builders provide more than one PLC program for the programming station, each specific to a different type of machine. Please contact your machine tool builder to request a description of the adapted PLC program's functions.

In order to increase protection of files and other data, the machine tool builder can encrypt the PLC, making it more difficult for third parties to access this information.

HEIDENHAIN points out that no data encryption method offers 100 % protection of the data, especially against access, damage or destruction by unauthorized persons, etc. Therefore, HEIDENHAIN cannot guarantee that data stored there will not be tampered with, and as a consequence cannot assume liability for any resulting damage.

Displaying additional drives

When you call the TNC file manager, the left window shows all drives available in the TNC. You can also have the TNC file manager display further network drives that you connected in the Windows Explorer:

- Start the programming station software

- Select the Programming and Editing mode of operation

- Select MOD functions
- Enter the code number 123 and confirm with the ENT key
- Select **Paths for the end user** (CfgUserPath)
- Select **List of drives and/or directories** (ncDir)
- Press the **MORE FUNCTIONS** soft key
- Press the **INSERT** soft key
- Enter the drive name, e.g. **C:**