



HEIDENHAIN

Pilot smarT.NC

iTNC 530

NC Software 340 490-03 340 491-03 340 492-03 340 493-03 340 494-03

English (en) 8/2006

The smarT.NC Pilot

... is your concise programming guide for the new **smarT.NC** operating mode of the iTNC 530. For more comprehensive information on programming and operating the iTNC 530, refer to the User's Manual.

Symbols in the Pilot

Certain symbols are used in the Pilot to denote specific types of information:



Important note

Control	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
iTNC 530 programming station	340 494-03



The TNC and the machine tool must be prepared by the machine tool builder to perform this function.



Warning: danger for the user or machine!

Contents

The smarT.NC Pilot	3
Fundamentals	5
Defining Machining Operations	32
Defining Machining Positions	121
Defining Contours	137
DXF Data Processing (Software Option)	147
Graphically Testing and Running a Unit Program	162

Fundamentals

Introduction to smarT.NC

With smarT.NC you can easily write structured conversational programs in separate working steps (units) and, if you want, edit them with the conversational editor. Since the **only data basis** for smarT.NC is the conversational program, you can modify data with the conversational editor, for example, and show them in the form view.

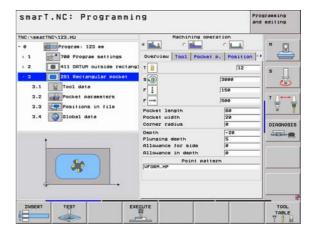
Immediately understandable input forms in the right half of the screen make it easier to define the required machining parameters, which are also displayed in a help graphic (lower left quarter of the screen). The structured program layout in a tree diagram (top left quarter of the screen) provides you with a quick overview of the working steps in a part program.

The separate and universal operating mode smarT.NC provides an alternative to the well known conversational programming. As soon as you have defined a machining step you can graphically test it and/or run it in the new operating mode.

Parallel programming

You can also create and edit smarT.NC programs at the same time that the TNC is running another program. Simply switch to the Programming and Editing operating mode and open the desired smarT.NC program there.

If you want to edit the smarT.NC program with the conversational editor, select the OPEN WITH function in the file manager, and then press CONVERSTL.



Fundamentals

5

Programs/Files

The TNC keeps its programs, tables and texts in files. A file designation consists of two components:

PROG20	.HU	
DDOCOO		

File name

File type

smarT.NC uses mainly three file types:

- Unit programs (file type .HU)
- Unit programs are conversational programs that contain two additional structuring elements: The beginning (UNIT XXX) and the end (END OF UNIT XXX) of a machining step.
- Contour descriptions (file type .HC)
- Contour descriptions are conversational programs. They must only contain path functions that can be used to describe a contour in the machining plane. The following elements are permitted: L, C with CC, CT, CR, RND, CHF as well as the FPOL, FL, FLT, FC and FCT elements for FK free contour programming
- Point tables (file type .HP)

smarT.NC saves in point tables the machining positions that you defined using the powerful pattern generator.



As a default, smarT.NC automatically saves all files in the **TNC:\smarTNC** directory. However, you can also select any other directory.

Files in the TNC	Туре
Programs In HEIDENHAIN format In ISO format	.H .I
smarT.NC files Structured unit program Contour descriptions Point tables for machining positions	.HU .HC .HP
Tables forToolsTool changersPalletsDatumsPresets (reference points)Cutting dataCutting materials, workpiece materials	.T .TCH .P .D .PR .CDT .TAB
Texts as ASCII files Help files	.A .CHM
Drawing data as DXF files	.DXF

Selecting the new operating mode the first time

- E)
- Select the smarT.NC operating mode: The file manager of the TNC appears.
- Select one of the available example programs with the arrow keys and press ENTER, or
- ▶ In order to write a new machining program, press the NEW FILE soft key. smarT.NC opens a pop-up window.
- ▶ Enter a file name with the extension .HU, and confirm with ENT.
- Confirm with the MM (or INCH) soft key or screen button. smarT.NC creates an .HU program with the selected units of measurement and automatically inserts the program header form.
- The data for the program header form are mandatory, since they are globally valid for the entire machining program. The default values are specified internally. Change the data if necessary, and save them with the END key.
- In order to define machining steps, press the EDIT soft key to select the desired machining step.

File management with smarT.NC

As mentioned previously, smarT.NC differentiates between three file types: unit programs (.HU), contour descriptions (.HC) and point tables (.HP). These three file types can be selected and edited in the file manager in the smarT.NC operating mode. Contour descriptions and point tables can also be edited if you are currently defining a working unit.

You can also open DXF files from within smarT.NC in order to extract contour descriptions (.HC files) and machining positions (.HP files) from them (software option).

The file manager in smarT.NC can also be fully operated with the mouse. You can even use the mouse to change the sizes of the windows within the file manager. Click the horizontal or vertical separating line, and drag it to the desired position with the mouse.

smarT.N	C:	File ma	inageme	nt					cameing editing
TNC:\searTNC	_	FR1.HP		_					1
01228	2	TNC:\searTNC	Ne. e						8 🖂
320 300R0F		File name		•	Size	Changed	5	tat 📩	E
		Scap_eno_2001	5		1707k	26.09.05 0	6:33 -		Concession of Co
BHB		CONT1			812	25.10.05 1	0:16 -		-
DEMO		COORD			939	82.12.85 6	7:46 -		s 🗌
DUMPPEN		ECOORD1			1390	02.12.05 0	9:47 -		4
didxf		ECPOC1			1130	06.12.05 0	10:22 -		C
> CFK		CPOCENE				26.07.05 1			
CHI	1000 5	CPOCKLINKS				29.04.05 0			T
CHGB		CSTUDLINKS				29.04.05 4			8
MHL		DREIECKRECH	rs			29.04.05 0			- Maria
NEWDEHO		flansch				17.11.05 0			
. DPENDELN	1	€FR1				11.04.05 1			DIAGNOS
SCHULE		GEARUHEEL		1		05.04.04 1			
- DsearTNC		GEARUHEEL				22.09.05 0			THE ROAD
++++++		GEARWHEEL				12.10.05 1			-
A A	4	HAKEN Ithaus				09.07.05 0			
4 4 The A	č –	HEBEL				21.03.05 1			
4	4	HEBEL				18.05.05 0			
4 4	÷ .	THEBEL PLONE				18.05.05 0			-
+ ;	4	HEBELPLANE				28.07.05			
· ++++++	P	HEBELSTUD				29.04.05 0			
\$ **++**		*				2010-1100 0			
+++++	P-0-3	79 Objects / E	570.0KBytes	/ 21214	. SHBy t	es free		_	
PAGE	PAGE	SELECT	COPY	BELEC	т	NEU	LR	ST	
4			D.D	1990	2	FILE	FIL		END
	•		ABC ** XYZ	TYPE	1	B	16,8	2.0	- 14 -

Calling the file manager

To select the file manager, press the PGM MGT key: The TNC displays the file manager window (the figure at right shows the default setting). If the TNC displays a different screen layout, press the WINDOW soft key on the second soft-key row.

The top window on the left shows the available drives and directories. Drives designate devices with which data are stored or transferred. A drive can be the hard disk of the TNC, directories connected via the network, or USB devices. A directory is always identified by a folder symbol to the left and the directory name to the right. The control displays a subdirectory to the right of and below its parent directory. A triangle pointing to the right in front of the folder symbol indicates that there are further subdirectories, which can be shown with the right arrow key.

The lower window on the left shows a preview of the file contents if an .HP or .HC file is highlighted.

TNC:\searTNC	FR1.HP			
320	= TNC:\searTNC\+.+	I av Ia	120010	H D
C) 3DGRAF	File name	Size Changed	Stat	-101-
CIANT	Bcap_eso_2005	1707k 26.09.05 06:3		-
ВНВ	CONT1	812 25.10.05 10:1		
DEMO	ECOORD	930 02.12.05 07:4		S
DUMPPGH	IECOORD1	1390 02.12.05 09:4		1
🗋 dx f	IECPOCI IECPOCENE	1130 06.12.05 00:2		
▷ □FK		1020 20.07.05 13:5		1000
CH1	CSTUDI INKS	124 29.04.05 06:2		1 1
C)HG8	OPFIECKRECHTS	150 29.04.05 05:2		
OHHL	T 1aosch	4788 17, 11, 05 09:5		1.5.5
NEWDEMO	EFP1	2888 11.84.85 15:1		
PENDELN	INGEORUHEEL	16573 05.04.04 14:2		DIAGNOSI
SCHULE	GEARNHEEL	3580 22,09,05 08:3		THE RO-C
- issarTNC	C BEARWHEEL	815 12,10,05 14:3	7	100000000000000000000000000000000000000
++++++	HAKEN	875 89.87.85 87:3	2	
A + + +	Phaus	2522 21.03.05 14:0	5	
+ +	HEBEL	512 09.07.05 15:4	6	
· · · · · · · · · · · · · · · · · · ·	IEHEBEL	772 18.05.05 07:0	7	
2 4 2	THEBELPLANE	914 27.84.85 e7:5	6	-
- + · · · ·	IEHEBELPOC	870 20.07.05 09:0	4	
A \$4.44\$	HEBELSTUD	150 29.04.05 05:2	9 +	
+++++++++	79 Objects / 6570,0K8ytes / 21			<u> </u>
			-	
PAGE PS		LECT NEW	LAST	
		PP FILE	FILES	END

The wide window on the right shows you all files that are stored in the selected directory. Each file is shown with additional information, illustrated in the table below.

Display	Meaning		
File name	Name with max. 16 characters		
Туре	File type		
Size	File size in bytes		
Changed	Date and time that the file was last changed		
Status	 File properties: E: Program is selected in the Programming and Editing mode of operation. S: Program is selected in the Test Run mode of operation. M: Program is selected in a Program Run mode of operation. P: File is protected against deletion and editing. +: Dependent files exist (structure file, toolusage file) 		

i

PGM MGT	Call the file manager.
	keys or the soft keys, you can move the highlight to the n on the screen:
9 8	Moves the highlight from the left to the right window, and vice versa.
	Moves the highlight up and down within a window.
PAGE PAGE	Moves the highlight one page up or down within a window.

Step 1: Select drive

Move the highlight to the desired drive in the left window:

SELECT	To select a drive, press the SELECT soft key, or
ENT	Press the ENT key.
Step 2: Sele	ct a directory

Move the highlight to the desired directory in the left-hand window—the right-hand window automatically shows all files stored in the highlighted directory.

i

Step 3: Select a file

SELECT TYPE	Press the SELECT TYPE soft key.
SELECT	Press the soft key for the desired file type, or
SHOU ALL	Press the SHOW ALL soft key to display all files, or
Move the	e highlight to the desired file in the right window
SELECT	Press the SELECT soft key, or
ENT	Press the ENT key: The TNC opens the selected file.
G.	If you type in a name from the keyboard, the TNC synchronizes the highlight with the entered characters so that you can find the file easier.

Fundamentals

Creating a new directory

- ▶ Press the PGM MGT key to select the file manager.
- Select the directory tree with the left arrow key.
- Select the TNC:\ drive if you want to create a new main directory, or select an existing directory to create a new subdirectory in it.
- ▶ Enter the name of the new directory, and confirm it with the ENT key. smarT.NC then displays a pop-up window with the new path name.
- Press ENT or the OK button to confirm. In order to cancel the procedure, press the ESC key or the Cancel button.

~	1
Lè	∃

You can also create a new program with the NEW DIRECTORY soft key. Then enter the directory name in the pop-up window and confirm with the ENT key.

Creating a new file

- Press the PGM MGT key to select the file manager.
- Select the file type of the new file, as described earlier
- ▶ Enter a file name without an extension, and confirm with ENT.
- Confirm with the MM (or INCH) soft key or screen button. smarT.NC creates a file with the selected units of measurement. In order to cancel the procedure, press the ESC key or the Cancel button.

48	

You can also create a new file with the NEW FILE soft key. Then enter the file name in the pop-up window and confirm with the ENT key.

Copying files into the same directory

- ▶ Press the PGM MGT key to select the file manager.
- ▶ Use the arrow keys to place the highlight on the file you want to copy.
- ▶ Press the COPY soft key. smarT.NC opens a pop-up window.
- Enter the file name of the target file without the file type, and confirm with the ENT key or the OK button. smarT.NC copies the contents of the selected file into a new file of the same file type. In order to cancel the procedure, press the ESC key or the Cancel button.
- If you want to copy the file to another directory, press the soft key for path selection, select the desired directory from the pop-up window, and confirm with ENT or the OK button.

Copying files into another directory

- Press the PGM MGT key to select the file manager.
- ▶ Use the arrow keys to place the highlight on the file you want to copy.
- Select the second soft-key row and press the WINDOW soft key to split the TNC screen.
- ▶ Shift the highlight to the left window with the left arrow key.
- ▶ Press the PATH soft key. smarT.NC opens a pop-up window.
- In the pop-up window, select the directory to which you want to copy the file, and confirm with ENT or the **0K** button.
- ▶ Shift the highlight to the right window with the right arrow key.
- ▶ Press the COPY soft key. smarT.NC opens a pop-up window.
- If necessary, enter the new file name of the target file without the file type, and confirm with the ENT key or the **0K** button. smarT.NC copies the contents of the selected file into a new file of the same file type. In order to cancel the procedure, press the ESC key or the **Cancel** button.

If you want to copy multiple files, select these with the mouse button. Press the CTRL key and select the desired files.

			and	editing
		FR1.HP		[
TNC: N		=TNC:\searTNC\+.+		H 🖂
File name	· Size -	File name	+ Size	- M
32e	<dir:< td=""><td>Cap_eso_2005</td><td>1707</td><td>Constant of Constant of Consta</td></dir:<>	Cap_eso_2005	1707	Constant of Consta
C) 3DGRAF	<dir:< td=""><td>CONT1</td><td>812</td><td></td></dir:<>	CONT1	812	
CAUT	<dir:< td=""><td>ECOORD</td><td>938</td><td>s T</td></dir:<>	ECOORD	938	s T
<u>0</u> 8H8	<dir:< td=""><td>IECOORD1</td><td>1390</td><td>4</td></dir:<>	IECOORD1	1390	4
DEMO	<dir:< td=""><td>IECPOC1</td><td>1130</td><td>0</td></dir:<>	IECPOC1	1130	0
DUMPPGM	<dir:< td=""><td>IE CPOCENE</td><td>1020</td><td></td></dir:<>	IE CPOCENE	1020	
🗋 dx f	<dir:< td=""><td>CPOCKLINKS</td><td>138</td><td>T</td></dir:<>	CPOCKLINKS	138	T
C)FK	<dir:< td=""><td>CSTUDLINKS</td><td>124</td><td></td></dir:<>	CSTUDLINKS	124	
CH1	<dir:-< td=""><td>DREIECKRECHTS</td><td>150</td><td>- Million</td></dir:-<>	DREIECKRECHTS	150	- Million
CHCB	<dir:< td=""><td>1 flansch</td><td>478 -</td><td></td></dir:<>	1 flansch	478 -	
C)MHL	<dir:< td=""><td>¥FR1</td><td>2688</td><td>DIAGNOST</td></dir:<>	¥FR1	2688	DIAGNOST
DNEUDEHO	<dir:< td=""><td>IS GEARWHEEL</td><td>16573</td><td></td></dir:<>	IS GEARWHEEL	16573	
PENDELN	<dir:< td=""><td>GEARWHEEL</td><td>3588</td><td>120.00</td></dir:<>	GEARWHEEL	3588	120.00
CISCHULE	<dir:< td=""><td>C GEARWHEEL</td><td>815</td><td></td></dir:<>	C GEARWHEEL	815	
isearTNC	<dir:< td=""><td>-HAKEN</td><td>876</td><td></td></dir:<>	-HAKEN	876	
🗅 tncguide	<dir:< td=""><td>Thaus</td><td>2622</td><td></td></dir:<>	Thaus	2622	
zyklen	<dir:< td=""><td>MEBEL</td><td>512</td><td></td></dir:<>	MEBEL	512	
DSHDI	2248	HE HEBEL	772	
I SHDI	350	THEBELPLANE	914	
I STCHPRMT	0	IE HEBELPOC	878	
STCHPRNT	132 .	HEBELSTUD	150 .	
B Objects / 3063,5KBytes	/ 21212.7HByte	1 79 Objects / 6570.0KBytes / 3	1214.9HByte	
PAGE PAGE	DELETE	RENAME WINDOW		

Deleting a file

- ▶ Press the PGM MGT key to select the file manager.
- ▶ Use the arrow keys to place the highlight on the file you want to delete.
- Select the second soft-key row.
- ▶ Press the DELETE soft key. smarT.NC opens a pop-up window.
- ▶ In order to delete the selected file, press either the ENT key or the Yes button. In order to cancel the delete procedure, press the ESC key or the No button.

Renaming a file

- Press the PGM MGT key to select the file manager.
- Use the arrow keys to place the highlight on the file you want to rename.
- Select the second soft-key row.
- ▶ Press the RENAME soft key. smarT.NC opens a pop-up window.
- Enter the new file name and confirm your entry with the ENT key or OK button. In order to cancel the procedure, press the ESC key or the Cancel button.

Selecting one of the last 15 files selected

- ▶ Press the PGM MGT key to select the file manager.
- Press the LAST FILES soft key. smarT.NC displays the last 15 files that you selected in the smarT.NC operating mode.
- ▶ Use the arrow keys to place the highlight on the file you want to select.
- ▶ Press the ENT key to select the file.

Updating directories

If you are navigating on an external data carrier, it might become necessary to update the directory tree:

- Press the PGM MGT key to select the file manager.
- Select the directory tree with the left arrow key.
- ▶ Press the UPDATE TREE soft key: The TNC updates the directory tree.

File sorting

Use the mouse to perform the file-sorting functions. You can sort the files by name, type, size, change date and file status, in ascending or descending order:

- Press the PGM MGT key to select the file manager.
- Click with the mouse the column header you want to sort by. A triangle in the column header indicates the sorting sequence. Click the header again to reverse the sequence.



Adapting the file manager

You can adapt the file manager as follows:

Bookmarks

You can use bookmarks to manage your favorite directories. You can add or delete the current directory to or from the list, or delete all bookmarks. All directories that you have added appear in the bookmark list, making them available for rapid selection.

🔳 View

In the View menu you specify the type of information the TNC is to display in the file window.

Date format

In the Date format menu you specify the format in which the TNC displays the date in the **Changed** column.

You open the menu for adapting the file manager either by clicking the path name 1 or with soft keys:

- ▶ Press the PGM MGT key to select the file manager.
- Select the third soft-key row.
- Press the MORE FUNCTIONS soft key.
- Press the OPTIONS soft key: The TNC displays the menu for adapting the file manager.
- ▶ Use the arrow keys to move the highlight to the desired setting.
- Activate or deactivate the desired setting with the space bar.

smarT.1	NC:	File managemen	t		editing
TNC:\S88FTNC 920 920GRAF 6AUT 80H8 00EM0 00UMPPCH 04Xf 9 0FK 4H1 10B8 MHL	2	File name View Add	AssarTNC active directory ste active directory te all 1930 02.12.05 1130 02.12.05 1020 25.07.05 1130 25.07.05 1130 25.04.05 1130 29.04.05 1150 29.04.05	10:16 07:46 09:47 00:22 13:53 06:29 06:29 06:29	* _
SAEUDEHO DEPROELN SECHULE SEARTINC	-	I Jarosh I Jarosh I Jarosh Jarosh Jarosh Hesel Hesel Hesel Hesel Hesel Hesel Hesel Jarosh Hesel Hesel Hesel Jarosh Hesel Jarosh Hesel Jarosh Hesel Jarosh Jaros	4764 17.11.45 2000 11.04.40 15573 05.04.04 3500 22.05.05 015 12.10.05 2022 21.02.06 512 09.07.05 772 10.05.06 914 27.04.05 150 29.07.05 150 29.04.05 150 29.04.05	15:15	DIAGNOSI
PAGE	PAGE	79 Objects / 6570,0KBytes /	21210.4MBries free	HORE	END

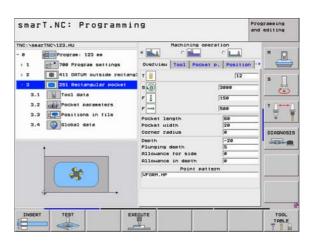
When developing smarT.NC, care was taken to ensure that the operating keys familiar from conversational programming (ENT, DEL, END, ...) are usable in a nearly identical manner in the new operating mode. The keys have the following functions:

Function when tree view is active (left side of screen)	Key
Activate form in order to enter or change data	
Conclude editing: smarT.NC automatically calls the file manager	END
Delete selected machining step (entire unit)	DEL
Position highlight to next/previous machining step	+ +
Show symbols for detail forms in tree view, if an arrow	

Show symbols for detail forms in tree view, if an **arrow pointing to the right** is displayed in front of the tree view symbol, or switch to the form if the detail view is already open

Hide symbols for detail forms in tree view, if an **arrow pointing down** is displayed in front of the tree view symbol

-	



Fundamentals

Function when tree view is active (left side of screen)	Кеу
Go to previous page	PAGE
Go to next page	PAGE
Go to beginning of file	BEGIN
Go to end of file	

Function when the form is active (right side of screen)	Кеу
Select next input field	ENT
Conclude editing of the form: smarT.NC saves all changed data	
Cancel editing of the form: smarT.NC does not save the changed data	
Position highlight to next/previous input field or element	+ +
Position the cursor within the active input field in order to change individual parts of the values, or if an option box is active: Select next/previous option	+ +

			d editing
INC:\searTNC\123.HU • 0 Program: 123 mm • 1 2 700 Program settings	Machining o * Overview Tool Pock	۰ 💶	. "
2 411 DATUM outside rectangl 2 251 Rectangular pocket 3.1 Tool data 3.2 proc Pocket parameters	5 (1) F 1	12 3000 150	s
2.2 Positions in file 3.4 Slobal data	F Pocket length Pocket width Corner radius	599 69 29 9	DIAGNOSI
i	Depth Plunging depth Allowance for side Allowance in depth	-20 5 0 0	12000
	Point pa VFORM.HP	ttern	
INSERT TEST EXE	сите		TOOL

i

Function when the form is active (right side of screen) Key

Reset an already entered numerical value to 0

Delete completely the contents of the active input field

NO ENT

CE

In addition, the TE 530 B keyboard unit has three new keys to allow you to navigate even faster within the forms:

Function when the form is active (right side of screen)	Кеу
Select next subform	
Select first input parameter in next frame	
Select first input parameter in previous frame	[]

i

When you edit contours you can also position the cursor with the orange axis keys so that the coordinate entry is identical to the conversational entry. You can also switch between absolute and incremental or Cartesian and polar coordinate programming with the relevant plainlanguage keys.

Function when the form is active (right side of screen)	Кеу
Select input field for X axis	X
Select input field for Y axis	Y
Select input field for Z axis	Ζ
Switch between incremental and absolute input	Ι
Switch between Cartesian and polar coordinate input	Ρ



Screen layout during editing

The screen layout while editing in the smarT.NC mode depends on the file type currently selected for editing.

Editing unit programs

- 1 Header: Operating mode text, error messages
- 2 Active background mode of operation
- 3 Tree view in which the defined working units are shown in a structured format
- 4 Form window with the various input parameters. Depending on the machining step, there can be up to five forms.

4.1: Overview form

Entry of the parameters in the overview form suffices to perform the basic functions of the current machining step. The data in the overview form are an excerpt of the most important data, which can also be entered in the detail forms.

4.2: Tool detail form

Input of additional tool-specific data

4.3: Optional parameters detail form

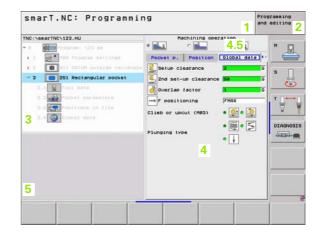
Input of additional, optional machining parameters

4.4: Positions detail form

Input of additional machining positions

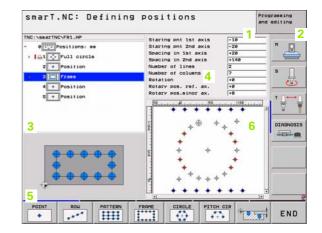
- 4.5: Global data detail form
 - List of globally effective data
- 5 Support graphics window, in which the input parameter currently active in the form is displayed

smarT.NC: Programmin	9	1	Programming and editing
TNC \SearTNG \22.HU C C C C C C C C C C C C C C C C C C C	* 4.1 4.2 Overview Tool T 11 S 10 F 11 F	4.3 4.4 Pocket p. Positio 12 9000 150 500	
3 3.4 💽 Global data	Pocket length Pocket width Corner radius Depth Plunging depth	68 20 9 -20 5	DIAGNOS
	Allowance for s Allowance in de	eth e	_
5	PO VFORM.HP	int pattern	



Editing machining positions

- 1 Header: Operating mode text, error messages
- 2 Active background mode of operation
- 3 Tree view in which the defined working patterns are shown in a structured format
- 4 Form window with the appropriate input parameters
- 5 Support graphics window, in which the input parameter currently active is displayed
- 6 Graphics window, in which the programmed machining positions are shown immediately after being saved in the form



Ť

Editing contours

- 1 Header: Operating mode text, error messages
- 2 Active background mode of operation
- 3 Tree view in which the contour elements are shown in a structured format
- 4 Form window with the various input parameters. In FK programming there can be up to four forms.
 - 4.1: Overview form
 - Contains the input possibilities used most often
 - 4.2: Detail form 1

Contains input possibilities for auxiliary points (FL/FLT) and for circle data (FC/FCT)

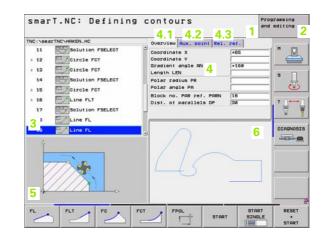
■ 4.3: Detail form 2

Contains input possibilities for relative references (FL/FLT) and for auxiliary points (FC/FCT)

■ 4.4: Detail form 3

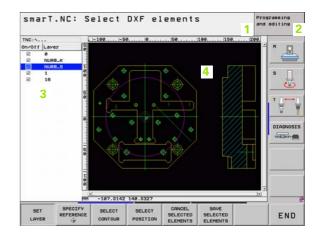
Only available for FC/FCT, contains input possibilities for relative references

- 5 Support graphics window, in which the input parameter currently active is displayed
- 6 Graphics window, in which the programmed contours are shown immediately after being saved in the form



Displaying DXF files

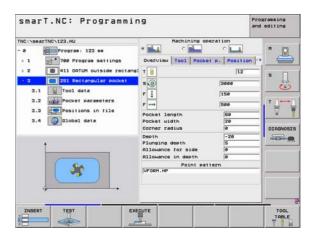
- 1 Header: Operating mode text, error messages
- 2 Active background mode of operation
- 3 Layers or already selected contour elements or positions in the DXF file
- 4 Drawing window in which smarT.NC shows the DXF file contents



Mouse operation

Using the mouse is also very easy. Please note the following specifics:

- In addition to the mouse functions familiar from Windows, you can also click the smarT.NC soft keys with the mouse.
- If multiple soft-key rows are present (represented by lines directly above the soft keys), you can activate a row by pressing the corresponding line.
- In the tree view, click the arrows pointing to the right to show detail forms, and click the arrows pointing down to hide them again.
- In order to change values in a form, click any input field or option box, and smarT.NC automatically switches to editing mode.
- To exit the form (to end the editing mode): Click anywhere in the tree view. Then smarT.NC asks whether to save changes in the form
- If you move the mouse over any window element, smarT.NC displays a tooltip. The tooltip contains brief information on the respective function of the element



Copying units

You can copy individual machining units very easily with the familiar Windows shortcut keys:

- Ctrl+C to copy the unit
- Ctrl+X to cut the unit out
- Ctrl+V to insert the unit subsequently to the currently active one

If you want to copy two or more units at the same time, proceed as follows:



- Switch to the top level of the soft-key row
- Use the cursor keys or the mouse to select the first unit to be copied
- BLOCK

COPY

BLOCK

INSERT

- Activate the marking function
- Use the cursor keys or the MARK NEXT BLOCK soft key to select all units to be copied
- Copy the marked block to the clipboard (also works with Ctrl+C)
- Use the cursor keys or soft key to select the unit just before the desired insertion point for the copied block

▶ Insert the block from the clipboard (also works with Ctrl+V)

smarT.NC: Programmi	iig		rogramming and editing
TNC:\searTNC\123.HU - 0 Prograe: 123 mm > 1 ***********************************	* 1	Pocket p. Position	•
 2 + 11 DATUH outside rectangle 3 251 Rectangular pocket 	T 10 S (0) F 11 F	12 3000 150	
	Pocket length Pocket width Corner radius	80 20 0	DIAGNOSI
	Depth Plunging depth Allowance for sid Allowance in dept		-
*	Poin VFORM.HP	t pattern	_
SELECT INSERT INSERT			

Fundamentals

Editing the tool table

You can edit the tool table TOOL.T immediately after you have selected the smarT.NC operating mode. The TNC displays the tool data structured in forms. The tool table is navigated in a manner identical to the rest of smarT.NC (see "Navigating in smarT.NC" on page 20).

The tool data are structured in the following groups:

Overview tab:

Summary of the tool data most often used, such as tool name, length and radius

Add. Data tab:

Additional tool data necessary for special applications

Add. Data tab:

Management of replacement tools and further additional tool data

Tch. Probe tab:

Data for 3-D touch probes and table touch probes

PLC tab:

Data that is necessary for interfacing your machine with the TNC, and that has been specified by your machine manufacturer

TNC:NTO		Overview Add.	data Add. dat	a 🕩
> 1 > 2	TZEROTOOL -	Tool name Comment Tool type	ORILLOG Comment Tool BOR	
TRdd. data T⊪Rdd. data *gTch. probe ≞PLC	Tool length Tool radius Oversize for tool	+0 +3	S	
S-COT 4 (NTLLOB 5 (TAPH10) 6 (TAPH10) 7 (12,42) 8 (LEP018)		Oversize for tool Maximum plunge ang Cutting length Number of teeth	radius +0	

CDT tab:

Data for automatic calculation of cutting data

~	1
ΓÀ	╤
~	3

Also refer to the detailed description of the tool data in the Conversational User's Manual.

The tool type is used by the TNC to determine the symbol shown in the treeview. Additionally, the TNC also shows the entered tool name in the treeview.

On the corresponding tabs, smarT.NC does not show tool data that have been deactivated via machine parameter. This may mean that one or more tabs are not even visible.

Defining Machining Operations

Fundamentals

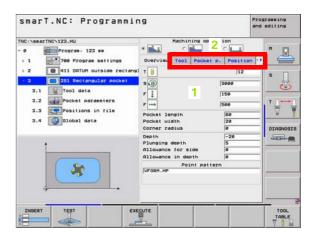
Machining operations are basically defined in smarT.NC as working steps (units), which as a rule consist of several conversational-language programming blocks. smarT.NC automatically creates the conversational blocks in the background in an .HU file (HU: **H**EIDENHAIN **U**nit program), which looks just like a **normal** conversational language program.

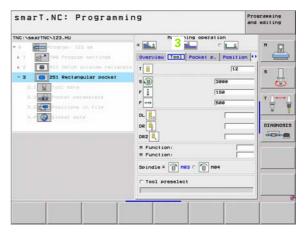
The actual machining operation is performed by a cycle available on the TNC. You specify the parameters via the input fields on the forms.

You can define a machining step with just a few entries in the overview form (1, see figure at top right). smarT.NC then performs the basic machining operation. Detail forms (2) are available for entering additional machining data. Values entered in the detail forms are automatically synchronized with the data entered in the overview form, so you don't have to enter them twice. The following detail forms are available:

Tool detail form (3)

On the tool detail form you can enter additional tool-specific data, such as delta values for the length and radius, or M functions.



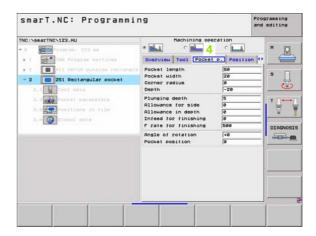


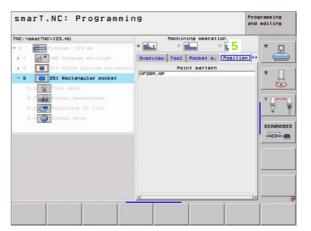
Optional parameters detail form (4)

On the optional parameters detail form you can define additional machining parameters which are not listed on the overview form, such as decrements for drilling or pocket lengths for milling.

Positions detail form (5)

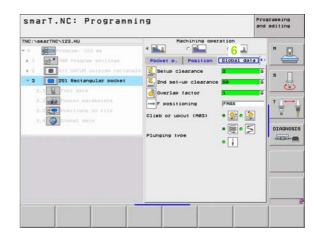
On the positions detail form you can define additional machining positions if the three machining locations on the overview form do not suffice. If you define machining positions in point tables, then both the overview form and the positions detail form only show the names of the point table files (see "Defining Machining Positions" on page 121.)





Global data detail form (6)

The globally effective machining parameters defined in the program header are listed on the global data detail form. If necessary, you can change these parameters for each unit locally.



⊥ 34

Program settings

After you have created a new unit program, smarT.NC automatically inserts the **Unit 700 Program Settings.**



The **Unit 700 Program Settings** must exist in a program, otherwise that program cannot be executed by smarT.NC.

The following data must be defined in the program settings:

- Workpiece blank definition for determining the machining plane and for the graphic simulation
- Options for selection of the workpiece preset and the datum table to be used.
- Global data, valid for the entire program. The global data are automatically assigned default values by smarT.NC. These can be changed at any time.

빤

Please note that later changes to the program settings affect the entire machining program, and can therefore change the machining procedure significantly.

NC:\searTNC\123.HU	Tool axis	z
e Progras: 122 es -1 22 Progras settings 1.1 22 Dottors ce blank 1.2 0 pottors -1.2 0 pottors 1.2.1 0 pottors 1.2.2 0 Postitioning 1.2.3 0 prilling 1.3.4 0 procket silling	Control sees blank Collions Bloba Beneral Beneral Setur Clearance 2 Prositioning PHe Positioning Behavior 4 Positioning Behavior 4 Positioning Behavior 4 Positioning Behavior 4 Positioning Behavior 4 Positioning Behavior 4 Dual time at top 0 Dual time at top 0 Class or useut (H93) 4 Setur Clearance height 1 Duaran factor 1 Cliss or useut (H93) 4 Setur Clearance height 1 Dual time at top 1 Dual time a	
INSERT TEST	EXECUTE	TOOL

Defining Machining Operations

Global data

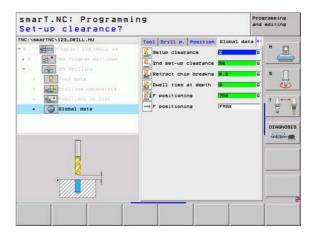
The global data are divided into six groups:

- Global data valid everywhere
- Global data valid only for boring and drilling operations
- Global data that determine the positioning behavior
- Global data valid only for milling operations with pocket cycles
- Global data valid only for milling operations with contour cycles
- Global data valid only for probing functions

As already mentioned, global data are valid for the entire machining program. Of course you can change the global data for any machining step, if necessary:

- Switch to the global data detail form for the machining step: In the form, smarT.NC shows the parameters valid for this machining step along with the currently active value. On the right side of the green input field is a G as an identifier that this value is valid globally.
- Select the global parameter that you want to change.
- Enter the new value and confirm with the ENTER key. smarT.NC changes the color of the input field to red.
- On the right side of the red input field there is now an L as an identifier that the value is valid locally.

smarT.NC: Programm	ing	Programming and editing
TKC \SearTKC\123.HJ - 0 Program: 122 mm - 1 Program: 122 mm - 1 Program: 122 mm - 2 Program: 122 mm - 1 Progr	Tool axis Upper action bank Collons Clob Belup Clearance 2 2 positioning 794 Positioning 794 Positionin	
INSERT	Cliab or upout (M03) * Stor	TOOL



Changing a global parameter on the **global data** detail form only effects a local change of the parameter, valid for that one machining step. smarT.NC displays the input fields of locally changed parameters with a red background. On the right side of the input field is an **L** which identifies the value as valid **locally**.

Press the SET STANDARD VALUES soft key to load and therefore activate the value of the global parameter from the program header. The input field of a global parameter whose value from the program header is in effect is displayed with a green background by smarT.NC. On the right side of the input field is a **G** which identifies the value as valid **globally**.



Global data valid everywhere

- Set-up clearance: Distance between tool tip and workpiece surface for automated approach of the cycle start position in the tool axis.
- 2nd setup clearance: Position to which smarT.NC positions the tool at the end of a machining step. The next machining position is approached at this height in the machining plane.
- **F positioning:** Feed rate at which smarT.NC traverses the tool within a cycle.
- **F** retraction: Feed rate with at smarT.NC retracts the tool.

Global data for positioning behavior

Positioning behavior: Retraction in the tool axis at the end of the machining step: Return to the 2nd set-up clearance or to the position at the beginning of the working unit.

INC:\searTNC\123.HU	Tool axis	z
 Program: 122 mm 10 mm 20 Program: settings 1.1 mm 20 orkpiece blank 1.2 mm 1.2.1 mm 1.2.2 mm 2.2 mm 2.3.3 mm 2.4 mm 2.4 mm 2.4 mm 2.4 mm 2.5 mm <l< td=""><td>2 Optimises blank Optimis Glob Beneral Setup Clearance 2 Ederation Setup Clearance Setup Clearance 2nd Setup Clearance Setup Clearance Setup Clearance 2nd Setup Clearance Setup Clearance Setup Clearance Positioning Positioning Positioning Positioning<</td><td></td></l<>	2 Optimises blank Optimis Glob Beneral Setup Clearance 2 Ederation Setup Clearance Setup Clearance 2nd Setup Clearance Setup Clearance Setup Clearance 2nd Setup Clearance Setup Clearance Setup Clearance Positioning Positioning Positioning Positioning<	

Global data for drilling operations

- Retraction rate for chip breaking: Value by which smarT.NC retracts the tool during chip breaking.
- Dwell time at depth: Time in seconds that the tool remains at the hole bottom.
- Dwell time at top: Time in seconds that the tool remains at the setup clearance.

Global data for milling operations with pocket cycles

- Overlap factor: The tool radius multiplied by the overlap factor equals the lateral stepover.
- **Climb or up-cut:** Select the type of milling.
- Plunging type: Plunge into the material helically, in a reciprocating motion, or vertically.

1.3.4 W Pocket Billing Docket Billing I	TNC:\searTNC\123.HU	Tool axis	z
	- 1 Per Program settings 1.1 Uorkpiece blank 1.2 Ortopic blank 1.2 Ortopic blank 1.2 Ortopic blank 1.2.1 Ortopic blank 1.2.2 Ortopic blank 1.2.2 Ortopic blank 1.2.2 Ortopic blank 1.2.2 Ortopic blank 1.2.2 Ortopic blank	General 2 Beture Clearance 2 Znd set-up Clearance 96 P restriction 9589 Positioning 9696 Positioning 9696 Positioning 9697 Positioning 9697	

Global data for milling operations with contour cycles

- Set-up clearance: Distance between tool tip and workpiece surface for automated approach of the cycle start position in the tool axis.
- Clearance height: Absolute height at which the tool cannot collide with the workpiece (for intermediate positioning and retraction at the end of the cycle).
- Overlap factor: The tool radius multiplied by the overlap factor equals the lateral stepover.
- **Climb or up-cut:** Select the type of milling.

Global data for probing functions

- Set-up clearance: Distance between stylus and workpiece surface for automated approach of the probing position.
- Clearance height: The coordinate in the touch probe axis to which smarT.NC traverses the touch probe between measuring points, if the Move to clearance height option is activated.
- Move to clearance height: Select whether smarT.NC moves the touch probe to the set-up clearance or clearance height between the measuring points.

Move to clearanc	mming e height (0/1)?	Programming and editing
NC:VSasTNC-122.HU 1.1 Startuce glay 2.2 Startuce glay 1.3 Startuce	Positioning behavior * Positioning behavior * Drilling Reiract chip breaking 8.2 Duell lise at dooth 8 Pocket silling Ouerlas factor Ouerlas factor Clastance r2 Clastance factor Clastance factor	

Tool selection

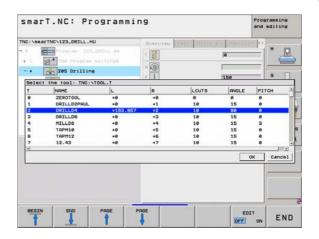
As soon as an input field in the tool selection is active, you can choose via the TOOL NAME soft key if you want to enter the tool number or tool name.

There is also a SELECT soft key for calling a window from which you can select a tool defined in the tool table TOOL.T. smarT.NC then automatically writes the tool number or name of the selected tool into the corresponding input field.

You can also edit the displayed tool data:

- Use the arrow keys to select the line and then the column of the value to be edited: The light-blue background marks the editable field
- Set the EDIT soft key to ON, enter the desired value and confirm with the ENT key.
- ▶ If needed, select further columns and repeat the described procedure

NC:\searTNC\123_DRILL.HU	Overview Tool Drill p. Position	1
 Toyras 172.001L es Toyras saturas 	T S T S S S S S S S S S S S S S S S S S	*
Number		
	SELECT	



RPM/cutting-speed switchover

As soon as an input field for defining the spindle speed is active, you can choose whether the speed will be displayed in rpm or as cutting speed (m/min or ipm).

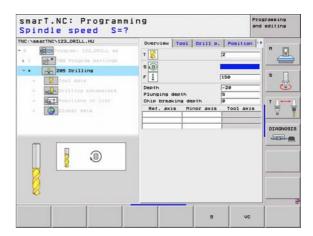
- To enter a cutting speed
- Press the VC soft key: the TNC switches the input field
- To switch from cutting speed to input in rpm
- ▶ Press the NO ENT key: the TNC deletes the cutting-speed input
- ▶ To enter rpm: use the arrow key to move back to the input field

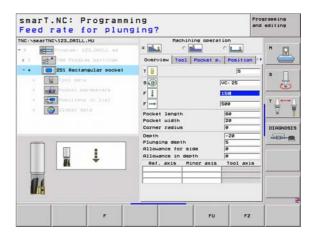
F/FZ/FU/FMAX switchover

As soon as an input field for defining the feed rate is active, you can choose whether the feed rate will be displayed in mm/min (F), in rpm (FU) or in mm/tooth (FZ). The permissible feed rate types depend on the respective machining operation. For some input fields, an FMAX entry (rapid) is also allowed.

To enter a type of feed rate

Press soft key F, FZ, FU or FMAX

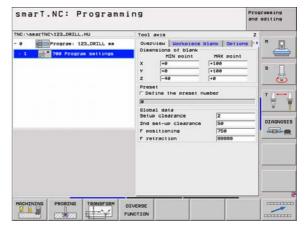




Available working steps (units)

After choosing the smarT.NC operating mode, you select the available working steps with the EDIT soft key. The working steps are divided into the following main groups:

Main group	Soft key	Page
MACHINING Boring, drilling, thread milling, milling	MACHINING	Page 44
PROBING Probe functions for 3-D touch probes	PROBING	Page 103
CONVERT Functions for coordinate transformation		Page 111
DIVERSE FUNCTIONS Program call, conversational unit	DIVERSE	Page 117



ᇝ

The CONTR. PGM and POSITIONS soft keys on the third soft-key row start the contour programming and pattern generator, respectively.

Machining main group

In the Machining main group you select the following machining groups:

Machining group	Soft key	Page
DRILLING Centering, drilling, reaming, boring, back boring	DRILLING	Page 45
THREAD Tapping with floating tap holder, rigid tapping, thread milling	THREAD	Page 56
POCKETS/STUDS Bore milling, rectangular pockets, circular pockets, slots, circular slots	POCKETS/ STUDS	Page 70
CONTR. PGM Run contour programs: contour train, contour pocket roughing, fine roughing and finishing	CONTR. PEN	Page 82
SURFACES Face milling	SURFACES	Page 99

NC:\searTNC\123_DRILL.HU	Tool axis	7	
0 Program: 122_ORILL am	Overview Verview Vervi	HAX point +100 +100 +0	
DRILLING THREAD POCKETS/ CO	ONTR. POH SURFACES		



Drilling machining group

The following working units are available for drilling operations in the Drilling machining group:

Unit	Soft key	Page
Unit 240 Centering	UNIT 248	Page 46
Unit 205 Drilling	UNIT 285	Page 48
Unit 201 Reaming	UNIT 281	Page 50
Unit 202 Boring	UNIT 282	Page 52
Unit 204 Back Boring	UNIT 284	Page 54

• 1 gg 200 700 Program settings	NIN Point HAX point Y re Y re Y re Preset re Postine the preset number Global data Setue clearance Setue clearance F Persetioning F retraction		
---------------------------------	--	--	--

Unit 240 Centering

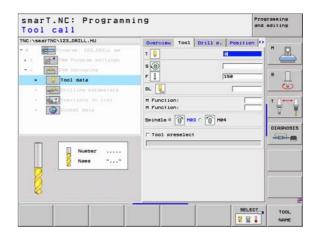
Parameters on the **overview** form:

- ► T: Tool number or name (switchable via soft key)
- S: Spindle speed [rpm] or cutting speed [m/min or ipm]
- ▶ F: Centering feed rate [mm/min] or FU [mm/rev]
- Select Depth/Diameter: Select whether centering is based on the depth or diameter.
- Diameter: Centering diameter. Input of T-ANGLE from TOOL.T required.
- **Depth:** Centering depth.
- Machining positions (see "Defining Machining Positions" on page 121.)
- Additional parameters on the **tool** detail form:
- **DL:** Delta length for tool T.
- **M** function: Any miscellaneous function M.
- **Spindle:** Direction of spindle rotation. As a default, smarT.NC sets M3.
- Tool preselect: If needed, this is the number of the next tool for faster tool change (machine-dependent).

Additional parameters on the $\ensuremath{\textbf{drilling}}\xspace$ parameters detail form:

▶ None

IC:\searTNC\123_DRILL.HU	Overview Tool Drill p. Position ()	
 Toyrae 172.001LL es Toyrae 172.001LL es Toyrae sollings Toyrae contacting Toyrae contactin	T S S Select depth/dia. 4 Diaseter -10 Depth Ref. exis Tool axis	* _
Nuster Nase ""		DIAGNO



Globally effective parameters on the **global data** detail form:

▶ Set-up clearance

<mark>8</mark>3

- ▶ 2nd set-up clearance
- Dwell time at depth
- ▶ Feed rate for traversing between machining positions

TRO: SAMATTRO: 122_0FILL HU • Martine Transmission (122_0FILL HU • 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1	Tool Drill P. Position Blobal da Setue Clearance P Setue Clearance B Debil the at depth B Debil the at depth B Positioning FMRX	
--	--	--

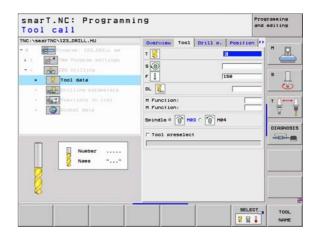


Unit 205 Drilling

Parameters on the overview form:

- ▶ T: Tool number or name (switchable via soft key)
- S: Spindle speed [rpm] or cutting speed [m/min or ipm]
- **F:** Drilling feed rate [mm/min] or FU [mm/rev]
- **Depth:** Drilling depth.
- ▶ Plunging depth: Dimension by which the tool plunges in each infeed before retraction from the hole.
- Chip breaking depth: Depth at which smarT.NC carries out chip breaking.
- Machining positions (see "Defining Machining Positions" on page 121.)
- Additional parameters on the **tool** detail form:
- **DL:** Delta length for tool T.
- **M** function: Any miscellaneous function M.
- **Spindle:** Direction of spindle rotation. As a default, smarT.NC sets M3.
- Tool preselect: If needed, this is the number of the next tool for faster tool change (machine-dependent).

Tool call Tho:\secThC.12.DRL.HU	T	and editing
		SELECT TOOL

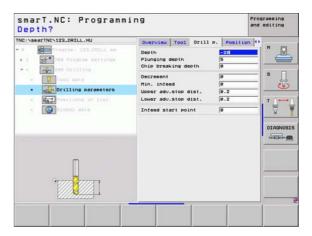


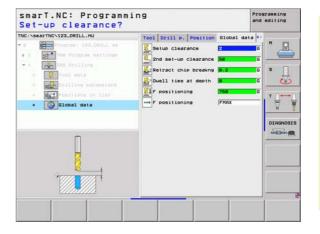
Additional parameters on the **drilling parameters** detail form:

- Chip breaking depth: Depth at which smarT.NC carries out chip breaking.
- **Decrement:** Value by which smarT.NC decreases the plunging depth.
- Min. infeed: If a decrement has been entered: Limit for minimum infeed.
- Upper adv.stop dist.: Upper set-up clearance for repositioning after chip breaking.
- Lower adv.stop dist.: Lower set-up clearance for repositioning after chip breaking.
- Infeed start point: Lower starting point relative to the surface coordinates for pre-machined holes

Globally effective parameters on the **global data** detail form:

- Set-up clearance
- ▶ 2nd set-up clearance
- Retraction value for chip breaking
- Dwell time at depth
- ▶ Feed rate for traversing between machining positions



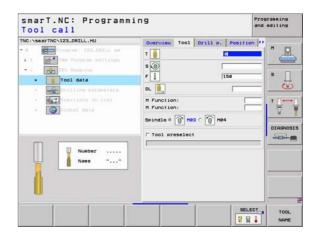


Unit 201 Reaming

Parameters on the **overview** form:

- **T:** Tool number or name (switchable via soft key)
- S: Spindle speed [rpm] or cutting speed [m/min or ipm]
- ▶ F: Reaming feed rate [mm/min] or FU [mm/rev]
- Depth: Reaming depth
- Machining positions (see "Defining Machining Positions" on page 121.)
- Additional parameters on the **tool** detail form:
- **DL:** Delta length for tool T.
- **M** function: Any miscellaneous function M.
- **Spindle:** Direction of spindle rotation. As a default, smarT.NC sets M3.
- Tool preselect: If needed, this is the number of the next tool for faster tool change (machine-dependent).

TNC:\searTNC\123_DRILL.HU	Overview Tool Orill p. Pos	ition ()
 Improvement 172_001L es Improvement 172_001L es Improvement 172_001L es Improvement 172_001 ests Improvement 172_001 ests Improvement 172_001 ests 	r i i i i i i i i i i i i i i i i i i i	



Additional parameters on the **drilling parameters** detail form:

▶ None

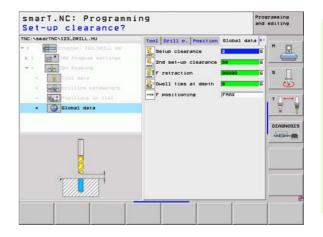
<mark>8</mark>3

....)

Globally effective parameters on the global data detail form:

- ▶ Set-up clearance
- 2nd set-up clearance
- Retraction feed rate
- Dwell time at depth
- ▶ Feed rate for traversing between machining positions

Depth? TNC:\searTNC\123_DRILL.HU				1.4
<pre>Mol CaseThe Vizzontal 192.COTLL as 0 1 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0</pre>	Depth	Tcol	Position	



Unit 202 Boring

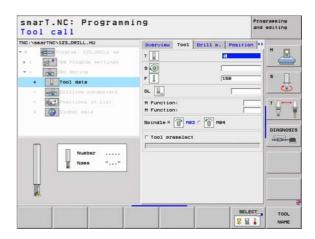
Parameters on the **overview** form:

- ▶ T: Tool number or name (switchable via soft key)
- S: Spindle speed [rpm] or cutting speed [m/min or ipm]
- ▶ F: Drilling feed rate [mm/min] or FU [mm/rev]
- Depth: Boring depth
- Disengaging direction: Direction in which smarT.NC moves the tool away from the counterbore floor.
- Machining positions (see "Defining Machining Positions" on page 121.)

Additional parameters on the **tool** detail form:

- **DL:** Delta length for tool T.
- **M** function: Any miscellaneous function M.
- **Spindle:** Direction of spindle rotation. As a default, smarT.NC sets M3.
- Tool preselect: If needed, this is the number of the next tool for faster tool change (machine-dependent).

TNC:\SearTNC\122_DRILL.HU - 0 - 0 - 0 - 0 - 0 - 0 - 0 - 0 - 0 -	Ouerview Tool Orill p. Positi T I I I I S I I I I F I I I I	
 mer prisitive waraneters mer tane (n 16) pristant data 	Desth -28 C C C C Disensaging directin C C C C Ref. axis Minor axis Tool d	



Additional parameters on the **drilling parameters** detail form:

Angle of spindle: Angle to which smarT.NC positions the tool before retracting it.

Globally effective parameters on the **global data** detail form:

▶ Set-up clearance

<mark>8</mark>3

- ▶ 2nd set-up clearance
- Retraction feed rate
- Dwell time at depth
- ▶ Feed rate for traversing between machining positions

NC:\searTNC\123_DRILL.HU	Overview Tool Drill P. Position ()
0 Program 122_DRILL am	Depth 201 1
Tool date Orilling parameters	Disensasing directn C C C S
 a mattines in 31a) a mattines in 31a) a mattines in 31a) 	Angle of spindle

<pre>************************************</pre>	Tool Drill P. Position 01 Setup Clearance 22 Setup Clearance 23 Setup Clearance 24 Setup Clearance 25 Duell time at depth 25 Duell time at depth 25 Positioning 77	
---	---	--



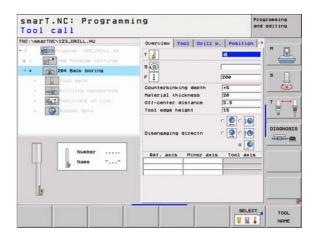
Unit 204 Back Boring

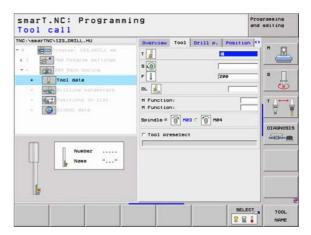
Parameters on the **overview** form:

- **T:** Tool number or name (switchable via soft key)
- S: Spindle speed [rpm] or cutting speed [m/min or ipm]
- F: Drilling feed rate [mm/min] or FU [mm/rev]
- **Countersinking depth:** Depth of cavity.
- Material thickness: Thickness of the workpiece.
- **Off-center distance:** Off-center distance of the boring bar.
- ▶ **Tool edge height:** Distance between the underside of the boring bar and the main cutting tooth; value from tool data sheet.
- Disengaging direction: Direction in which smarT.NC moves the tool by the off-center distance.
- Machining positions (see "Defining Machining Positions" on page 121.)

Additional parameters on the **tool** detail form:

- **DL:** Delta length for tool T.
- **M** function: Any miscellaneous function M.
- **Spindle:** Direction of spindle rotation. As a default, smarT.NC sets M3.
- Tool preselect: If needed, this is the number of the next tool for faster tool change (machine-dependent).





Additional parameters on the **drilling parameters** detail form:

Angle of spindle: Angle at which smarT.NC positions the tool before it is plunged into or retracted from the bore hole.

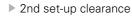


Dwell time at counterbore floor

Globally effective parameters on the **global data** detail form:

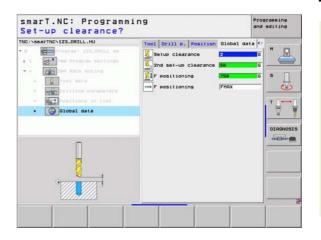


▶ Set-up clearance



- ► Feed rate for positioning
- ▶ Feed rate for traversing between machining positions

NC:\searTNC\123_DRILL.HU	Overview	T001	Drill p.	Positio	n O	
9 Program 123_ORILL mm 1 W* TWP Program mettings • • Con 204 Seck boring • • Con data	Countersin Material Off-center Tool edge	hickne dista	ss [nce [20 3.5 15		" s
Drilling parameters Drilling parameters Drilling parameters Drilling parameters	Disensagin	ng dire	etn (- Q
	Angle of t Dwell time			+0	=	DIAGNOS



Ť

Thread machining group

The following working units are available for thread operations in the Thread machining group:

Unit	Soft key	Page
Unit 206 Tapping with a Floating Tap Holder	UNIT_205	Page 57
Unit 209 Rigid Tapping (also with Chip Breaking)	UNIT 209	Page 58
Unit 262 Thread Milling	UNIT 262	Page 60
Unit 263 Thread Milling / Countersinking	UNIT 263	Page 62
Unit 264 Thread Drilling / Milling	UNIT 254	Page 64
Unit 265 Helical Thread Drilling / Milling	UNIT 265	Page 66
Unit 267 Outside Thread Milling	UNIT 257	Page 68

TNC:VS##ITNCY122_DRILL.HU ■ ■ Program: 122_DRILL em • 1 Bet Program: 122_DRILL em		
UNIT 200 UNIT 200 UNIT 202 U	UNIT 283 UNIT 284 UNIT 285 UNIT 287	

56

Unit 206 Tapping with a Floating Tap Holder

Parameters on the **overview** form:

- ▶ T: Tool number or name (switchable via soft key)
- S: Spindle speed [rpm] or cutting speed [m/min or ipm]
- F: Drilling feed rate: Calculate from S multiplied by thread pitch p
- **Depth of thread:** Depth of the thread.
- Machining positions (see "Defining Machining Positions" on page 121.)

Additional parameters on the **tool** detail form:

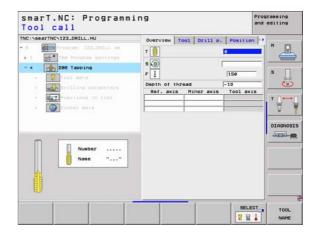
- **DL:** Delta length for tool T.
- **M** function: Any miscellaneous function M.
- **Spindle:** Direction of spindle rotation. As a default, smarT.NC sets M3.
- Tool preselect: If needed, this is the number of the next tool for faster tool change (machine-dependent).

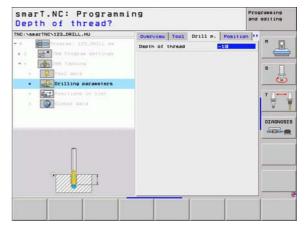
Additional parameters on the drilling parameters detail form:

▶ None

Globally effective parameters on the global data detail form:

- Set-up clearance
- 2nd set-up clearance
- Dwell time at depth
- ▶ Feed rate for traversing between machining positions



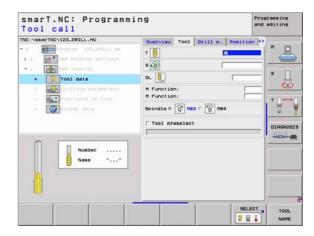


Unit 209 Rigid Tapping

Parameters on the **overview** form:

- ▶ T: Tool number or name (switchable via soft key)
- S: Spindle speed [rpm] or cutting speed [m/min or ipm]
- Depth of thread: Depth of the thread.
- **Thread pitch:** Pitch of the thread.
- Machining positions (see "Defining Machining Positions" on page 121.)
- Additional parameters on the **tool** detail form:
- **DL:** Delta length for tool T.
- **M** function: Any miscellaneous function M.
- **Spindle:** Direction of spindle rotation. As a default, smarT.NC sets M3.
- Tool preselect: If needed, this is the number of the next tool for faster tool change (machine-dependent).

INC:\searTNC\123_DRILL.HU	Overview Tool Drill p. Position (*	-
<pre> Trustras: 122.021Lk as 1</pre>	Vertile Total Utility P. Position	
88		



Additional parameters on the **drilling parameters** detail form:

- **Chip breaking depth:** Depth at which chip breaking is to occur.
- Angle of spindle: Angle to which smarT.NC positions the tool before thread cutting: This permits regrooving of the thread, if needed.
- S factor for retraction Q403: Factor by which the TNC increases the spindle speed—and therefore also the retraction feed rate—when retracting from the drill hole.

Globally effective parameters on the **global data** detail form:

Set-up clearance

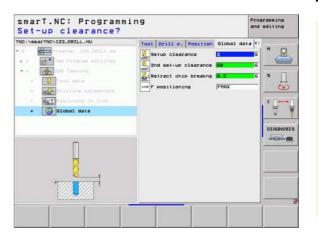
8

87

.....

- ▶ 2nd set-up clearance
- Retraction value for chip breaking
- ▶ Feed rate for traversing between machining positions

INC:\searTNC\123_DRILL.HU	Overview Tool Drill P.	Position 0	-
• 0 Program: 122_ORILL mm • 1 Program settings	Depth of thread Thread pitch	+1.5	" 🖉
• • Taboing	Chip breaking depth Angle of spindle S factor for retraction	0 +0 1	s []
· Drilling parameters			
* Presitions in list			• ∯•••
Π			DIAGNOSI



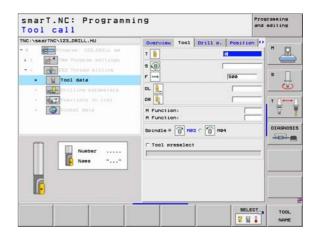
Unit 262 Thread Milling

Defining Machining Operations

Parameters on the overview form:

- **T:** Tool number or name (switchable via soft key)
- S: Spindle speed [rpm] or cutting speed [m/min or ipm]
- ▶ F: Feed rate for milling
- **Diameter:** Nominal diameter of the thread.
- **Thread pitch:** Pitch of the thread.
- **Depth:** Depth of thread.
- Machining positions (see "Defining Machining Positions" on page 121.)
- Additional parameters on the **tool** detail form:
- **DL:** Delta length for tool T.
- **DR:** Delta radius for tool T.
- **M** function: Any miscellaneous function M.
- **Spindle:** Direction of spindle rotation. As a default, smarT.NC sets M3.
- Tool preselect: If needed, this is the number of the next tool for faster tool change (machine-dependent).

 Solutions Tool acts 	INC:\searTNC\123_DRILL.HU		ition 🕑 H
Desth -18 Tool axis Desth -18 Tool axis Discol ax	 1 262 Thread silling 1 262 Thread silling 1 100 cots 	5 (8) F ==== 500 Diameter [10	
Nunber	Lateral		DIAGNOS



Additional parameters on the drilling parameters detail form:

Threads per step: Number of thread revolutions by which the tool is offset.

Globally effective parameters on the **global data** detail form:

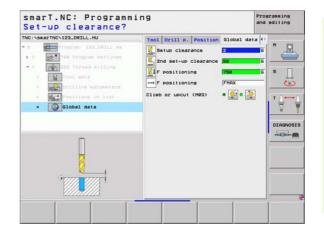
▶ Set-up clearance

<u>8</u>

.....

- ▶ 2nd set-up clearance
- Positioning feed rate
- ▶ Feed rate for traversing between machining positions
- Climb milling, or
- ▶ Up-cut milling

NC:\searTNC\123_DRILL.HU	Overview	Tool	Drill p.	Position	
Program 122_DRILL em 1 The Program Settings	Diameter Thread pit Depth	ch		10 +1.5 -18	
* * Thread ailling	Threads pe	r step		0	s 🗍
· Orilling parameters					
 Impositions in list Impositional data 					
· · · · · · · ·					
Manilli.					



Unit 263 Thread Milling / Countersinking

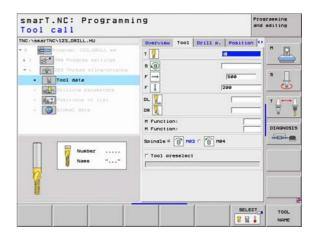
Parameters on the **overview** form:

- ▶ T: Tool number or name (switchable via soft key)
- S: Spindle speed [rpm] or cutting speed [m/min or ipm]
- ▶ F: Feed rate for milling
- ▶ F: Countersinking feed rate [mm/min] or FU [mm/rev]
- **Diameter:** Nominal diameter of the thread.
- **Thread pitch:** Pitch of the thread.
- **Depth:** Depth of thread.
- Countersinking depth: Distance between the top surface of the workpiece and the tool tip during countersinking.
- **Clearance to side:** Distance between tool tooth and the wall.
- Machining positions (see "Defining Machining Positions" on page 121.)

Additional parameters on the **tool** detail form:

- **DL:** Delta length for tool T.
- **DR:** Delta radius for tool T.
- M function: Any miscellaneous function M.
- **Spindle:** Direction of spindle rotation. As a default, smarT.NC sets M3.
- Tool preselect: If needed, this is the number of the next tool for faster tool change (machine-dependent).

Image: second	smart.NC: Programmi Tool call Tocismarthe.122.081L.HU - 0 Frence 122.081L e - 1 Program 122.081L e - 1 Program 122.081L e - 1 Programmi - 1 Pr	Overview Tool Drill T	Position ()	editing
Nusber	· Positions in 1181	Diameter Thread pitch Depth Countersinking depth Clearance to side	10 +1.5 -18 -20 0.2	



Additional parameters on the **drilling parameters** detail form:

- **Depth at front:** Depth for sinking at front.
- Offset at front: Distance by which the TNC moves the tool center out of the hole during countersinking at front.

Globally effective parameters on the global data detail form:

▶ Set-up clearance

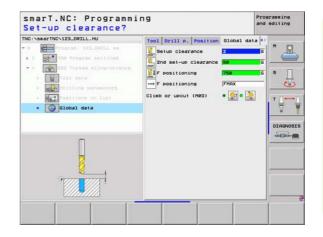
8

<u>8</u>1

....)

- 2nd set-up clearance
- Positioning feed rate
- ▶ Feed rate for traversing between machining positions
- Climb milling, or
- ▶ Up-cut milling

Image: The Program satisfies	NC:\searTNC\123_DRILL.HU	OUSTUISH	Tool	Drill p.	Position	0
Countersinking depth -28 Countersink	• 1 2 764 Program sallings	Thread pit	ch		+1.5	
Griget at front Go atobal date Griget at front Go atobal date Griget at front Go atobal date	Linkol					= °
· () Dichai data	· Drilling parameters	Depth at f	ront		+0	
						DIAGNOS



Ť

Unit 264 Thread Drilling / Milling

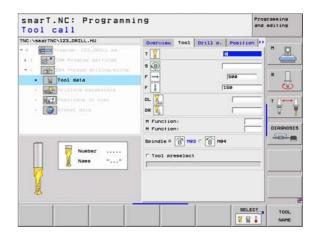
Parameters on the **overview** form:

- **T:** Tool number or name (switchable via soft key)
- S: Spindle speed [rpm] or cutting speed [m/min or ipm]
- ▶ F: Feed rate for milling
- F: Drilling feed rate [mm/min] or FU [mm/rev]
- **Diameter:** Nominal diameter of the thread.
- **Thread pitch:** Pitch of the thread.
- **Depth:** Depth of thread.
- **Total hole depth:** Total hole depth.
- Plunging depth for drilling
- Machining positions (see "Defining Machining Positions" on page 121.)

Additional parameters on the **tool** detail form:

- **DL:** Delta length for tool T.
- **DR:** Delta radius for tool T.
- **M** function: Any miscellaneous function M.
- **Spindle:** Direction of spindle rotation. As a default, smarT.NC sets M3.
- Tool preselect: If needed, this is the number of the next tool for faster tool change (machine-dependent).

Tool call The:\searthCvi2.cRit.,HU	Overview Tool Orill P T S Overview Tool Orill P T S P Diameter Thread plich Depin Total hole depth Ping. dpth. drilling Ref. axis Minor axis	Position 500 150	
Nase ""		SELECT	TOOL



Additional parameters on the **drilling parameters** detail form:

- Chip breaking depth: Depth at which the TNC is to carry out chip breaking during drilling.
- Upper adv. stop dist.: Set-up clearance for when the TNC returns the tool to the current plunging depth after chip breaking.
- **Depth at front:** Depth for sinking at front.
- Offset at front: Distance by which the TNC moves the tool center from the hole center

Globally effective parameters on the **global data** detail form:

Set-up clearance

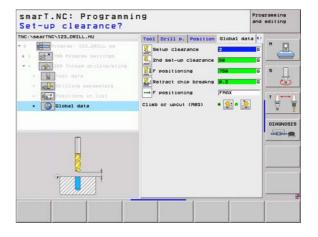
8

8

.....

- 2nd set-up clearance
- Positioning feed rate
- Retraction value for chip breaking
- ▶ Feed rate for traversing between machining positions
- Climb milling, or
- ▶ Up-cut milling

INC:\searTNC\123_DRILL.HU	Overview	T001	Drill p.	Position	
Program: 123_D91LL mm Togram: 123_D91LL mm Togram: sellings ** Poil Program selling-miling	Diameter Thread pit Depth	ch		10 +1.5 -10	 .
• Story data • Control data • Control data	Total hole Plng. dpth Chip break Upper adv.	. dril ing de	ling Pth	-20 5 0 0.2	
 Interference in list Interference in list 	Depth at f Offset at			9 +0	
			0		



Unit 265 Helical Thread Drilling / Milling

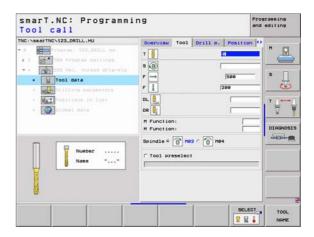
Parameters on the **overview** form:

- **T:** Tool number or name (switchable via soft key)
- S: Spindle speed [rpm] or cutting speed [m/min or ipm]
- ▶ F: Feed rate for milling
- ▶ F: Countersinking feed rate [mm/min] or FU [mm/rev]
- **Diameter:** Nominal diameter of the thread.
- **Thread pitch:** Pitch of the thread.
- **Depth:** Depth of thread.
- Countersink: Select whether countersinking occurs before or after thread milling.
- **Depth at front:** Depth for sinking at front.
- Offset at front: Distance by which the TNC moves the tool center from the hole center
- Machining positions (see "Defining Machining Positions" on page 121.)

Additional parameters on the **tool** detail form:

- **DL:** Delta length for tool T.
- **DR:** Delta radius for tool T.
- **M** function: Any miscellaneous function M.
- **Spindle:** Direction of spindle rotation. As a default, smarT.NC sets M3.
- Tool preselect: If needed, this is the number of the next tool for faster tool change (machine-dependent).

INC: SearTNC 123_DRILL.HU	Internet to Description of the second second second	in the second se
Orogram 122.0011L me Orogram Sellings Orogram Sellings Orogram Sellings Orogram Sellings Orogram Sellings Orogram Sellings Orogram Sellings	Overview Tool Drill p. Pos T	
 A Descriptions in list A Description deta 	Diameter 10 Thread pitch +1.5 Depth -18	=
	Countersink Depth at front Offset at front	OIRGNOS
Nusber	Ref. axis Minor axis Too	1 axis
U		



Additional parameters on the **drilling parameters** detail form:

▶ None

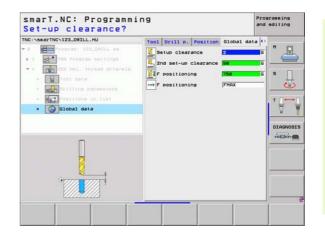
8

81

Globally effective parameters on the global data detail form:

- ▶ Set-up clearance
- ▶ 2nd set-up clearance
- Positioning feed rate
- ▶ Feed rate for traversing between machining positions

Contersink C	C:\searTNC\123_DRILL.HU	Overview	T001	Drill p.	Position	0	
Countersink Countersi	1 200 Program sattings	Thread pit	ch		+1.5		"_
(WE) UTILING parameters (If set at front (If set at front	• Tod3 data				Commits Card		s []
	* Positions in 1181						• ₽ ⊷
							DIAGNOS

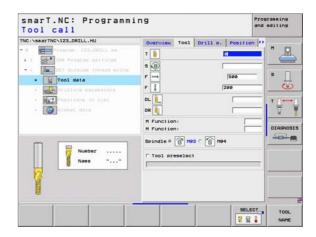


Unit 267 Thread Milling

Parameters on the **overview** form:

- ▶ T: Tool number or name (switchable via soft key)
- S: Spindle speed [rpm] or cutting speed [m/min or ipm]
- ▶ F: Feed rate for milling
- ▶ F: Countersinking feed rate [mm/min] or FU [mm/rev]
- **Diameter:** Nominal diameter of the thread.
- **Thread pitch:** Pitch of the thread.
- **Depth:** Depth of thread.
- Machining positions (see "Defining Machining Positions" on page 121.)
- Additional parameters on the **tool** detail form:
- **DL:** Delta length for tool T.
- **DR:** Delta radius for tool T.
- **M** function: Any miscellaneous function M.
- **Spindle:** Direction of spindle rotation. As a default, smarT.NC sets M3.
- ► **Tool preselect:** If needed, this is the number of the next tool for faster tool change (machine-dependent).

S D F ma Jackter Diaseter Thread pitch Depth Ref. axis Minor axis Too	3 axis OTAGNOT
	F 100 F 100 Diameter 100 Thread pitch +1.5 Depth -13



Additional parameters on the **drilling parameters** detail form:

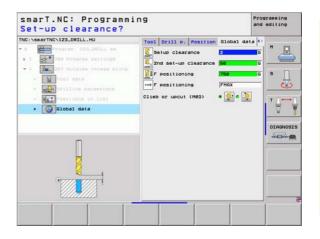
- Threads per step: Number of thread revolutions by which the tool is offset.
- **Depth at front:** Depth for sinking at front.
- Offset at front: Distance by which the TNC moves the tool center from the stud center

Globally effective parameters on the **global data** detail form:



- Set-up clearance
- ▶ 2nd set-up clearance
- Positioning feed rate
- ▶ Feed rate for traversing between machining positions
- Climb milling, or
- ▶ Up-cut milling

NC:\searTNC\123_DRILL.HU	Overview	Tcol	Drill p.	Position	
0 Program 122_DATL em	Diameter Thread pit Depth	ch		101 +1.5 -10	
 CET Dutside thread aling Tool data Drilling parameters 	Threads pe Depth at f Offset at	ront		0 +0 0	- s _
 Partions in list Closel data 					
,,,,,,,,,,,,,,,,,,,,,,,,,,,,,,,,,,,,,,,					



Ť

Pockets/Studs machining group

The following working units are available for the milling of simple pockets and slots in the Pockets/Studs machining group:

Unit	Soft key	Page
Unit 251 Rectangular Pocket	UNIT 251	Page 71
Unit 252 Circular Pocket	UNIT 252	Page 73
Unit 253 Slot	UNIT 253	Page 75
Unit 254 Circular Slot	UNIT 254	Page 77
Unit 208 Bore Milling	UNIT 288	Page 80

	Tool axis	2
• 0	x ref rst y ref rst z -ref rst r ref rst Preset r rst Coloring the preset number rst rst B rst rst Stelue clearence rst Znd set-up clearence rst r pesticioning	IAX point ee

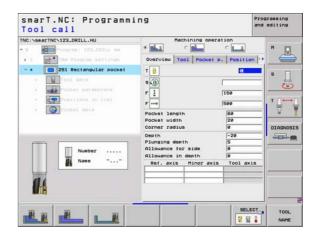
Unit 251 Rectangular Pocket

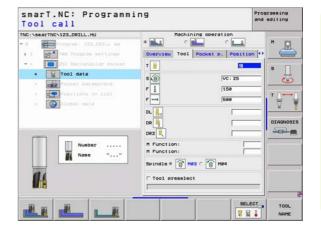
Parameters on the **overview** form:

- Machining operation: Select via soft key roughing and finishing, or only roughing, or only finishing.
- ▶ T: Tool number or name (switchable via soft key)
- S: Spindle speed [rpm] or cutting speed [m/min or ipm]
- ▶ F: Feed rate for plunging [mm/min], FU [mm/rev] or FZ [mm/tooth]
- ▶ F: Feed rate for plunging [mm/min] or FU [mm/rev] or FZ [mm/tooth]
- Pocket length: Pocket length in the reference axis.
- Pocket width: Pocket width in the auxiliary axis.
- Corner radius: If not entered, smarT.NC sets the corner radius equal to the tool radius.
- **Depth:** Final depth of the pocket.
- Plunging depth: Infeed per cut.
- ▶ Allowance for side: Finishing allowance for the side.
- ▶ Allowance in depth: Finishing allowance for the depth.
- Machining positions (see "Defining Machining Positions" on page 121.)

Additional parameters on the **tool** detail form:

- **DL:** Delta length for tool T.
- **DR:** Delta radius for tool T.
- **DR2:** Delta radius 2 (corner radius) for tool T.
- **M** function: Any miscellaneous function M.
- **Spindle:** Direction of spindle rotation. As a default, smarT.NC sets M3.
- Tool preselect: If needed, this is the number of the next tool for faster tool change (machine-dependent).





8

8

.....

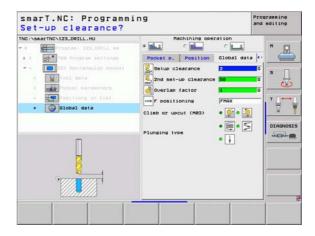
Additional parameters on the **pocket parameters** detail form:

- Infeed for finishing: Infeed for finishing on the side. If nothing is entered, finishing is performed in one infeed.
- ► F rate for finishing: Feed rate for finishing [mm/min], FU [mm/rev] or FZ [mm/tooth]
- > Angle of rotation: Angle by which the entire pocket is rotated.
- Pocket position: Position of the pocket referenced to the programmed position.

Globally effective parameters on the global data detail form:

- Set-up clearance
- 2nd set-up clearance
- Overlap factor
- ▶ Feed rate for traversing between machining positions
- Climb milling, or
- Up-cut milling
- Plunge in a helical motion, or
- Plunge in a reciprocating motion, or
- Plunge vertically

C:\searTNC\123_DRILL.HU	Machining opera	it ion	-
9 Program: 122_DRILL mm		° 🛄	H P
1 700 Program mettings	Overview Tool Pocket s	. Position **	
• • ESI Rectangular pocket	Pocket length	64	8 0
· Tool data	Pocket width Corner radius	20	
· Pocket parameters	Depth	-20	
* Positions in list	Plunging depth	15	T
· O global data	Allowance for side Allowance in depth	0	8
	Infeed for finishing	8	
	F rate for finishing	500	DIAGNO
	Angle of rotation	+0	TELECO
1	Pocket position	0	
			5



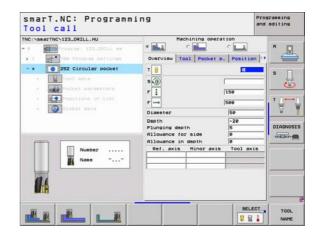
Unit 252 Circular Pocket

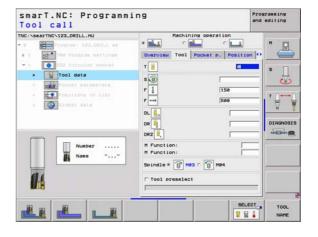
Parameters on the overview form:

- Machining operation: Select via soft key roughing and finishing, or only roughing, or only finishing.
- ▶ T: Tool number or name (switchable via soft key)
- S: Spindle speed [rpm] or cutting speed [m/min or ipm]
- ▶ F: Feed rate for plunging [mm/min], FU [mm/rev] or FZ [mm/tooth]
- ▶ F: Feed rate for plunging [mm/min] or FU [mm/rev] or FZ [mm/tooth]
- **Diameter:** Finished diameter of the circular pocket
- **Depth:** Final depth of the pocket.
- Plunging depth: Infeed per cut.
- ▶ Allowance for side: Finishing allowance for the side.
- ▶ Allowance in depth: Finishing allowance for the depth.
- Machining positions (see "Defining Machining Positions" on page 121.)

Additional parameters on the $\ensuremath{\textbf{tool}}$ detail form:

- **DL:** Delta length for tool T.
- **DR:** Delta radius for tool T.
- **DR2:** Delta radius 2 (corner radius) for tool T.
- **M** function: Any miscellaneous function M.
- **Spindle:** Direction of spindle rotation. As a default, smarT.NC sets M3.
- Tool preselect: If needed, this is the number of the next tool for faster tool change (machine-dependent).





8

8

....

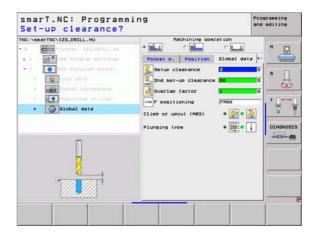
Additional parameters on the **pocket parameters** detail form:

- ▶ **Infeed for finishing:** Infeed for finishing on the side. If nothing is entered, finishing is performed in one infeed.
- ► F rate for finishing: Feed rate for finishing [mm/min], FU [mm/rev] or FZ [mm/tooth]

Globally effective parameters on the **global data** detail form:

- Set-up clearance
- ▶ 2nd set-up clearance
- Overlap factor
- ▶ Feed rate for traversing between machining positions
- Climb milling, or
- Up-cut milling
- Plunge in a helical motion, or
- Plunge vertically

Circle diameter?			
NC:\searTNC\123_DRILL.HU	Machining oper	ration	
0 Program: 123_DRILL mm		- Long	
• 1 700 Program settings	Overview Tool Pocket		-
• • DIST Gircular pockat	Diameter	-20	s 🗍
• Tool data	Plunging depth	15	1
· Pocket parameters	Allowance for side	0	
· Positions in list	Allowance in depth Infeed for finishing	0	T
* 🚱 Global data	F rate for finishing	500	2
			DIAGNOS
· · · · · · · · · · · · · · · · · · ·			THE ROOM
			-
1 1 1		1	1



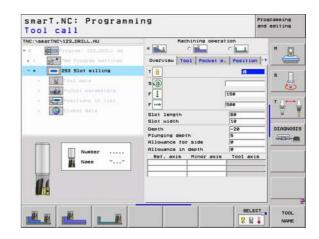
Unit 253 Slot

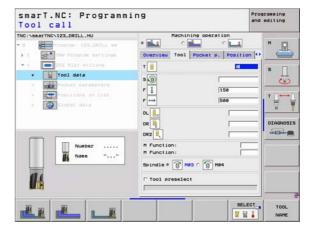
Parameters on the overview form:

- Machining operation: Select via soft key roughing and finishing, or only roughing, or only finishing.
- ▶ T: Tool number or name (switchable via soft key)
- S: Spindle speed [rpm] or cutting speed [m/min or ipm]
- F: Feed rate for plunging [mm/min], FU [mm/rev] or FZ [mm/tooth]
- ▶ F: Feed rate for plunging [mm/min] or FU [mm/rev] or FZ [mm/tooth]
- Slot length: Slot length in the reference axis.
- **Slot width:** Slot width in the auxiliary axis.
- **Depth:** Final depth of the slot.
- Plunging depth: Infeed per cut.
- ▶ Allowance for side: Finishing allowance for the side.
- ▶ Allowance in depth: Finishing allowance for the depth.
- Machining positions (see "Defining Machining Positions" on page 121.)

Additional parameters on the **tool** detail form:

- **DL:** Delta length for tool T.
- **DR:** Delta radius for tool T.
- **DR2:** Delta radius 2 (corner radius) for tool T.
- **M** function: Any miscellaneous function M.
- **Spindle:** Direction of spindle rotation. As a default, smarT.NC sets M3.
- Tool preselect: If needed, this is the number of the next tool for faster tool change (machine-dependent).





8

8

....

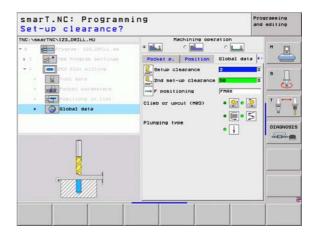
Additional parameters on the **pocket parameters** detail form:

- Infeed for finishing: Infeed for finishing on the side. If nothing is entered, finishing is performed in one infeed.
- ► F rate for finishing: Feed rate for finishing [mm/min], FU [mm/rev] or FZ [mm/tooth]
- > Angle of rotation: Angle by which the entire pocket is rotated.
- Slot position: Position of the slot referenced to the programmed position.

Globally effective parameters on the global data detail form:

- Set-up clearance
- 2nd set-up clearance
- ▶ Feed rate for traversing between machining positions
- Climb milling, or
- Up-cut milling
- Plunge in a helical motion, or
- Plunge in a reciprocating motion, or
- Plunge vertically

NC:\searTNC\123_DRILL.HU	Hachining ope	c L	H
• 1 700 Program settings	Overview Tool Pocket	P. Position **	
* * 253 Miot milling * Tool data	Slot length Slot width Depth	10 -20	s []
Pocket parameters Postitions in list Opsilional data	Plunging depth Allowance for side Allowance in depth Infeed for finishing F rate for finishing	5 0 0 500	· ₽
	Angle of rotation Slot position	+0 0	DIAGNOST



Unit 254 Circular Slot

Parameters on the **overview** form:

- Machining operation: Select via soft key roughing and finishing, or only roughing, or only finishing.
- ▶ T: Tool number or name (switchable via soft key)
- S: Spindle speed [rpm] or cutting speed [m/min or ipm]
- F: Feed rate for plunging [mm/min], FU [mm/rev] or FZ [mm/tooth]
- ▶ F: Feed rate for plunging [mm/min] or FU [mm/rev] or FZ [mm/tooth]
- **Center in 1st axis:** Center of pitch circle in reference axis.
- **Center in 2nd axis:** Center of pitch circle in auxiliary axis.
- Pitch circle diameter
- **Starting angle:** Polar angle of the starting point.
- ▶ Angular length
- ▶ Slot width
- **Depth:** Final depth of the slot.
- Plunging depth: Infeed per cut.
- ▶ Allowance for side: Finishing allowance for the side.
- > Allowance in depth: Finishing allowance for the depth.
- Machining positions (see "Defining Machining Positions" on page 121.)

fool call	Hack in inc.		-
NC:\searTNC\123_DRILL.HU 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0	Hachining open * • • • • • • • • • • • • • • • • • • •	c 🛄	. " 💻
- 254 Circular slot - 1001 data - 1001 dat	T	0 150 500	s T
· O Trocat sala	Center in 1st axis Center in 2nd axis Pitch circle diametr Starting angle Angular length Slot width	+50 +50 80 +0 0 10	DIRGNOSI
Number	Depth Plunging depth Allowance for side Allowance in depth Ref. axis Minor axis	-20 5 0 0 Tool axis	-



Additional parameters on the **tool** detail form:

- **DL:** Delta length for tool T.
- **DR:** Delta radius for tool T.
- **DR2:** Delta radius 2 (corner radius) for tool T.
- **M** function: Any miscellaneous function M.
- **Spindle:** Direction of spindle rotation. As a default, smarT.NC sets M3.
- Tool preselect: If needed, this is the number of the next tool for faster tool change (machine-dependent).

smarT.NC: Programm Tool call	ing	Programming and editing
TNC:\searTNC\123_DRILL.HU	Hechining operation	Lon +>
Construction and electronical and e	Y Image: Constraint of the second secon	
Nusber Nase ""	DR	
	SE 2	LECT TOOL

Additional parameters on the **pocket parameters** detail form:

- Infeed for finishing: Infeed for finishing on the side. If nothing is entered, finishing is performed in one infeed.
- F rate for finishing: Feed rate for finishing [mm/min], FU [mm/rev] or FZ [mm/tooth]
- **Stepping angle:** Angle by which the entire slot is rotated further.
- **Number of repetitions:** Number of machining operations on a pitch circle.
- Slot position: Position of the slot referenced to the programmed position.

Globally effective parameters on the global data detail form:

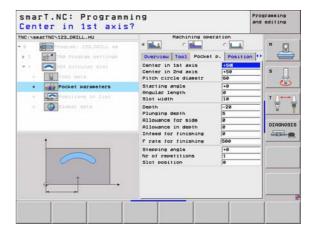
Set-up clearance

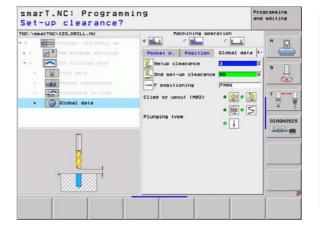
8

.....

- 11/ 100 🐼 🕸

- 2nd set-up clearance
- Feed rate for traversing between machining positions
- Climb milling, or
- ▶ Up-cut milling
- Plunge in a helical motion, or
- Plunge in a reciprocating motion, or
- Plunge vertically





Unit 208 Bore Milling

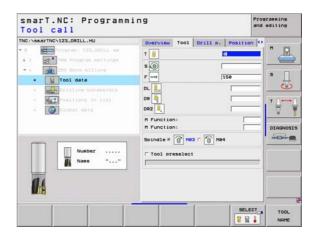
Parameters on the **overview** form:

- ▶ T: Tool number or name (switchable via soft key)
- S: Spindle speed [rpm] or cutting speed [m/min or ipm]
- ▶ F: Feed rate for plunging [mm/min] or FU [mm/rev] or FZ [mm/tooth]
- **Diameter:** Nominal diameter of the hole.
- **Depth:** Milling depth.
- ▶ Plunging depth: Depth of the tool plunge with each helix (360°).
- Machining positions (see "Defining Machining Positions" on page 121.)

Additional parameters on the **tool** detail form:

- **DL:** Delta length for tool T.
- **DR:** Delta radius for tool T.
- **DR2:** Delta radius 2 (corner radius) for tool T.
- **M** function: Any miscellaneous function M.
- **Spindle:** Direction of spindle rotation. As a default, smarT.NC sets M3.
- Tool preselect: If needed, this is the number of the next tool for faster tool change (machine-dependent).

Tool call The Searthe Size Parameter Size Settings	T B S B P mm Diaseter Depth	and editing
		SELECT TOOL



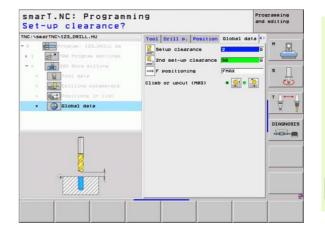
Additional parameters on the **drilling parameters** detail form:

Predrilled diameter: Enter if pre-drilled holes are to be machined again. This allows you to rough-mill holes more than twice as large as the tool diameter.

Globally effective parameters on the global data detail form:

- ▶ Set-up clearance
- ▶ 2nd set-up clearance
- ▶ Feed rate for traversing between machining positions
- ▶ Climb milling, or
- ▶ Up-cut milling

NC:\searTNC\123_DRILL.HU	Overview Tool Drill	p. Position	
O Program 122_DRILL am 1 The Program mettings	Diameter Depth Plunging depth	-28 0.25	-
* * Tool data	Predrilled diameter	(e	s 🗍
· Orilling parameters			$-\omega$
 A Prositional in 1141 A Prositional data 			• ⊡ ⊷
m			OIAGNOS
a.			
1 1 1		1	1



Contour Program machining group

The following working units for milling pockets and contour trains of any shape are available in the Contour Program machining group:

Unit	Soft key	Page
Unit 122 Contour Pocket Rough-Out	UNIT 122	Page 83
Unit 22 Contour Pocket Fine Roughing	UNIT 22	Page 87
Unit 123 Contour Pocket Floor Finishing	UNIT 123	Page 89
Unit 124 Contour Pocket Side Finishing	UNIT 124	Page 90
Unit 125 Contour Train	UNIT 125	Page 92
Unit 130 Contour Pocket on Point Pattern	UNIT 138	Page 95

TNC:\SmarTNC\123_DRILL.HU - 0	Tool axis Overview Deriview District D Hernstons of Diank Hit Point X = 0 - 2 - 0 - 0 - 0 - 0 - 0 - 0 - 0 - 0	Z lank Options (*) HAX point (*100 (*0	N S S S S S S S S S S S S S S S S S S S
UNIT 122 UNIT 22 UNIT 123 U	NIT 124	UNIT 125	

i 82

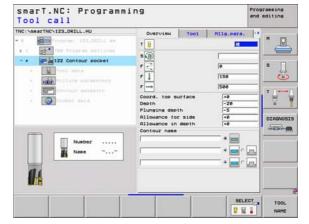
Unit 122 Contour Pocket

With the Contour Pocket working unit you can rough out pockets, which may contain islands, of any shape.

If necessary, you can assign every subcontour its own depth (FCL 2 function) in the **contour** detail form. In this case you must always begin with the deepest pocket.

Parameters on the overview form:

- ▶ T: Tool number or name (switchable via soft key)
- S: Spindle speed [rpm] or cutting speed [m/min or ipm]
- F: Feed rate for reciprocating plunge [mm/min], FU [mm/rev] or FZ [mm/tooth] Enter 0 for perpendicular infeed
- F: Feed rate for plunging [mm/min], FU [mm/rev] or FZ [mm/tooth]
- ▶ F: Feed rate for plunging [mm/min] or FU [mm/rev] or FZ [mm/tooth]
- ► Top surface coordinate: Workpiece top-surface coordinate given with respect to the entered depths.
- **Depth:** Milling depth.
- Plunging depth: Infeed per cut.
- ▶ Allowance for side: Finishing allowance for the side.
- > Allowance in depth: Finishing allowance for the depth.
- Contour name: List of the subcontours (.HC files) to be linked. If the DXF converter option is available, you can use it to make a contour directly from the form.



1 84

- 빤
- Specify via soft key for each subcontour whether it is a pocket or an island.
- As a rule, always start the list of subcontours with a pocket (if required, the deepest pocket)!
- On the contour detail form you can define up to a maximum of nine subcontours (see figure at bottom right).

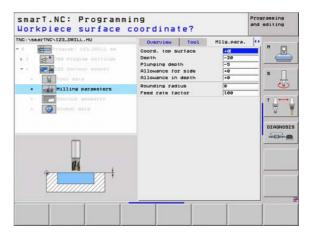
Additional parameters on the tool detail form:

- **DL:** Delta length for tool T.
- DR: Delta radius for tool T.
- **DR2:** Delta radius 2 (corner radius) for tool T.
- **M** function: Any miscellaneous function M.
- **Spindle:** Direction of spindle rotation. As a default, smarT.NC sets M3.
- Tool preselect: If needed, this is the number of the next tool for faster tool change (machine-dependent).

Additional parameters on the milling parameters detail form:

- Rounding radius: Rounding radius of the tool midpoint path at inside corners
- ▶ Feed rate factor in %: Percentage factor by which the TNC reduces the machining feed rate as soon as the tool moves within the material over its entire circumference during roughing. If you use the feed rate reduction, then you can define the feed rate for roughing so large that there are optimum cutting conditions with the specified path overlap (global data). The TNC then reduces the feed rate as per your definition at transitions and narrow places, so the machine time should be reduced in total.

NC:\searTNC\123_DRILL.HU	OURTUINN TOO	1 Hilg.para.	0
Constant 170.051Lb as Constant 170.051Lb as Constant 200 State Constate Constate Constate Constate Constate Constate Co	1 0 5 0 F = F = F = F = F = F = DR = DR = DR = DR = DR = DR = H Function: H Fu	 0 150 500	
		SEU	ECTTOOL



Additional parameters on the **contour** detail form:

- Depth: Separately definable depths for each subcontour (FCL 2 function)

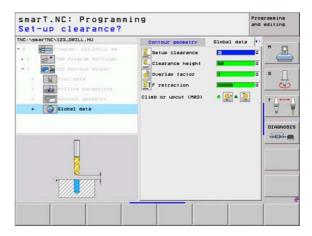
ᇞ

- As a rule, always start the list of subcontours with the deepest pocket!
- If the contour is defined as an island, the TNC interprets the entered depth as the island height. The entered value (without an algebraic sign) then refers to the workpiece top surface!
- If the depth is entered as 0, then for pockets the depth defined in the overview form is effective. Islands then rise up to the workpiece top surface!

Globally effective parameters on the global data detail form:

- ▶ Set-up clearance
- 2nd set-up clearance
- Overlap factor
- Retraction feed rate
- Climb milling, or
- ▶ Up-cut milling

TNC:\smarTNC\123_ORILL.HU	Tool Hilg.pa Contour name	ra. Contour secsets	* <mark>* </mark> * _
 1 Program settings 4 Program settings 5 Tool data 	Depth		
+ ROA Milling waranatars	Depth	• • •	
Contour geometry	Depth	*	
	Depth		IL DIAGNOS
	Depth		
	Depth	· · · · ·	
	Depth	• •	
	Depth	· · · · · · · · · · · · · · · · · · ·	



Unit 22 Fine Roughing

With the fine roughing unit you can use a smaller tool to rework a contour pocket that has been roughed-out with unit 122. In this step, smarT.NC machines only the places the previous tool did not reach.

Parameters on the **overview** form:

- ▶ T: Tool number or name (switchable via soft key)
- **S:** Spindle speed [rpm] or cutting speed [m/min or ipm]
- F: Feed rate for plunging [mm/min], FU [mm/rev] or FZ [mm/tooth]
- **F:** Feed rate for plunging [mm/min] or FU [mm/rev] or FZ [mm/tooth]
- ► **Coarse roughing tool:** Number or name of the tool (switchable via soft key) with which you have coarse-roughed the contour.
- Plunging depth: Infeed per cut.

Tool call The Case The Call of	Overview Tool Mlis.sa T	ra. Global dar C a 150 500	
Nusber Nase ""	Plunging depth	-5	DIAGNOSIS



Additional parameters on the **tool** detail form:

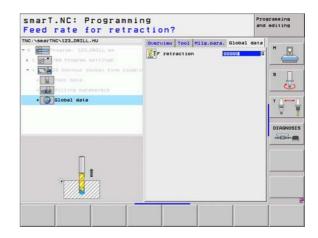
- **DL:** Delta length for tool T.
- **DR:** Delta radius for tool T.
- **DR2:** Delta radius 2 (corner radius) for tool T.
- **M** function: Any miscellaneous function M.
- **Spindle:** Direction of spindle rotation. As a default, smarT.NC sets M3.
- Tool preselect: If needed, this is the number of the next tool for faster tool change (machine-dependent).
- Additional parameters on the milling parameters detail form:
- ▶ None

Globally effective parameters on the global data detail form:



Retraction feed rate

NC:\searTNC\123_DRILL.HU	Increased and Increased Increased	Concession in the second
Torras IZLOPILLAN Torras IZLOPILAN Torras alling Torras alling	Dueruleu Tool Hils.para. Global T	
Nusber	DE2 L. R Function: R Function: Spindle * 1 Page () He4 (Tool preselect	



Unit 123 Contour Pocket Floor Finishing

With the floor finishing unit you can finish the floor of a contour pocket that has been roughed-out with unit 122.

Always run the floor finishing operation before side finishing!

Parameters on the overview form:

- ▶ T: Tool number or name (switchable via soft key)
- S: Spindle speed [rpm] or cutting speed [m/min or ipm]
- F: Feed rate for plunging [mm/min], FU [mm/rev] or FZ [mm/tooth]
- ▶ F: Feed rate for plunging [mm/min] or FU [mm/rev] or FZ [mm/tooth]

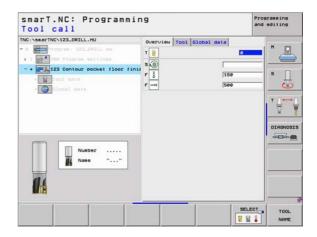
Additional parameters on the **tool** detail form:

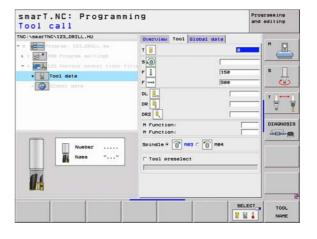
- **DL:** Delta length for tool T.
- DR: Delta radius for tool T.
- **DR2:** Delta radius 2 (corner radius) for tool T.
- **M** function: Any miscellaneous function M.
- **Spindle:** Direction of spindle rotation. As a default, smarT.NC sets M3.
- Tool preselect: If needed, this is the number of the next tool for faster tool change (machine-dependent).

Globally effective parameters on the **global data** detail form:



Retraction feed rate





Unit 124 Contour Pocket Side Finishing

With the side finishing unit you can finish the side of a contour pocket that has been roughed-out with unit 122.

빤

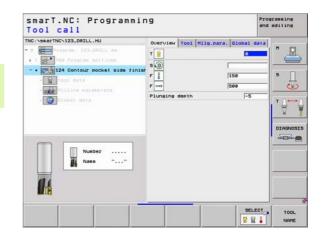
Always run the side finishing operation after floor finishing!

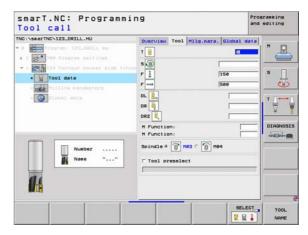
Parameters on the **overview** form:

- ▶ T: Tool number or name (switchable via soft key)
- **S:** Spindle speed [rpm] or cutting speed [m/min or ipm]
- F: Feed rate for plunging [mm/min], FU [mm/rev] or FZ [mm/tooth]
- F: Feed rate for plunging [mm/min] or FU [mm/rev] or FZ [mm/tooth]
- Plunging depth: Infeed per cut.

Additional parameters on the tool detail form:

- **DL:** Delta length for tool T.
- **DR:** Delta radius for tool T.
- DR2: Delta radius 2 (corner radius) for tool T.
- **M** function: Any miscellaneous function M.
- **Spindle:** Direction of spindle rotation. As a default, smarT.NC sets M3.
- Tool preselect: If needed, this is the number of the next tool for faster tool change (machine-dependent).





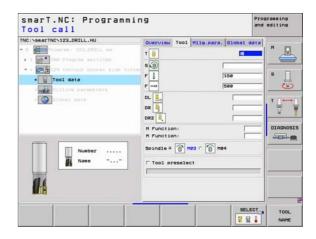
Additional parameters on the **milling parameters** detail form:

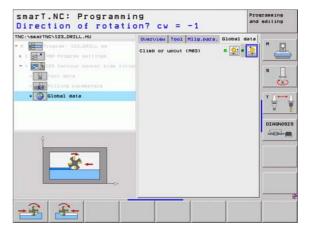
Finishing allowance for side: Finishing allowance for finishing in two or more steps

Globally effective parameters on the **global data** detail form:



- Climb milling, or
- ▶ Up-cut milling





i

Unit 125 Contour Train

With Contour Train you can machine open and closed contours that you defined in an .HC program or generated with the DXF converter.



Defining Machining Operations

Choose start and end points of the contour that leave enough space for tool approach and departure!

Parameters on the **overview** form:

- ▶ T: Tool number or name (switchable via soft key)
- S: Spindle speed [rpm] or cutting speed [m/min or ipm]
- F: Feed rate for plunging [mm/min], FU [mm/rev] or FZ [mm/tooth]
- ▶ F: Feed rate for plunging [mm/min] or FU [mm/rev] or FZ [mm/tooth]
- ▶ Top surface coordinate: Workpiece top-surface coordinate given with respect to the entered depths.
- **Depth:** Milling depth.
- Plunging depth: Infeed per cut.
- ▶ Allowance for side: Finishing allowance.
- Type of milling: Climb milling, up-cut milling or reciprocating machining.
- **Radius compensation:** Machine the contour with compensation to the left, to the right, or without compensation.
- **Type of approach:** Approach the contour tangentially on a circular path, or tangentially on a straight line, or vertically.
- Approach radius (Only in effect if tangential approach on a circular path was selected): Radius of the circular arc.

smarT.NC: Programm: Tool call	ing	Programming and editing
TRC:SeeTRC:123_DRILL.HU	Ouerview Tool Mile.mars. Blobal T B B C Coord. top surface Coord. top surface D D D D D D D D D D D D D	
Nase ""	Radius comensation Type of approach Represent radius Center angle Center angle Contour name Contour name Contou	
	SEL 2	ECT TOOL NAME

- Center angle (Only in effect if tangential approach on a circular path was selected): Angle of the circular arc.
- Distance to aux. point (Only in effect if tangential approach on a straight path or vertical approach was selected): Distance to the auxiliary point from which the contour is approached.
- Contour name: Name of the contour file (.HC) to be machined. If the DXF converter option is available, you can use it to make a contour directly from the form.

Additional parameters on the **tool** detail form:

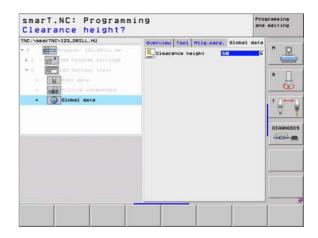
- **DL:** Delta length for tool T.
- **DR:** Delta radius for tool T.
- **DR2:** Delta radius 2 (corner radius) for tool T.
- **M** function: Any miscellaneous function M.
- **Spindle:** Direction of spindle rotation. As a default, smarT.NC sets M3.
- Tool preselect: If needed, this is the number of the next tool for faster tool change (machine-dependent).
- Additional parameters on the milling parameters detail form:
- ▶ None

Globally effective parameters on the global data detail form:



2nd set-up clearance

TNC:\searTNC\123_DRILL.HU	Overview Tool Hilg.	para. Global data
O O O O O	T B S (0) F I F m OL I DR DR DR DR T Punction: H Function: H Function: Spindle # (1) Hes c (1) T Tool preselect	



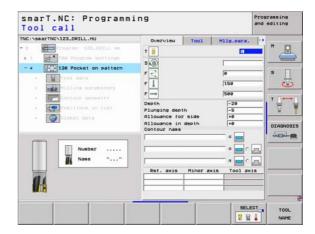
Unit 130 Contour Pocket on Point Pattern (FCL 3 Function)

With this working unit you can align on any pattern and rough out pockets, which may contain islands, of any shape.

If necessary, you can assign every subcontour its own depth (FCL 2 function) in the **contour** detail form. In this case you must always begin with the deepest pocket.

Parameters on the overview form:

- ▶ T: Tool number or name (switchable via soft key)
- S: Spindle speed [rpm] or cutting speed [m/min or ipm]
- F: Feed rate for reciprocating plunge [mm/min], FU [mm/rev] or FZ [mm/tooth] Enter 0 for perpendicular infeed
- ▶ F: Feed rate for plunging [mm/min], FU [mm/rev] or FZ [mm/tooth]
- ▶ F: Feed rate for plunging [mm/min] or FU [mm/rev] or FZ [mm/tooth]
- **Depth:** Milling depth.
- Plunging depth: Infeed per cut.
- ▶ Allowance for side: Finishing allowance for the side.
- > Allowance in depth: Finishing allowance for the depth.
- Contour name: List of the subcontours (.HC files) to be linked. If the DXF converter option is available, you can use it to make a contour directly from the form.



Positions or point pattern: Define the positions at which the TNC is to machine the contour pocket (see "Defining Machining Positions" on page 121.)

ᇞ

Specify via soft key for each subcontour whether it is a pocket or an island.

- As a rule, always start the list of subcontours with a pocket (if required, the deepest pocket)!
- On the contour detail form you can define up to a maximum of nine subcontours.

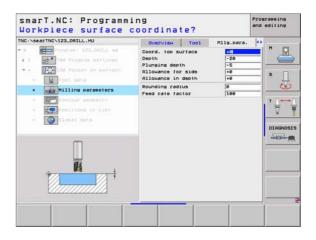
Additional parameters on the tool detail form:

- **DL:** Delta length for tool T.
- DR: Delta radius for tool T.
- **DR2:** Delta radius 2 (corner radius) for tool T.
- **M** function: Any miscellaneous function M.
- **Spindle:** Direction of spindle rotation. As a default, smarT.NC sets M3.
- Tool preselect: If needed, this is the number of the next tool for faster tool change (machine-dependent).

Additional parameters on the milling parameters detail form:

- Rounding radius: Rounding radius of the tool midpoint path at inside corners
- ▶ Feed rate factor in %: Percentage factor by which the TNC reduces the machining feed rate as soon as the tool moves within the material over its entire circumference during roughing. If you use the feed rate reduction, then you can define the feed rate for roughing so large that there are optimum cutting conditions with the specified path overlap (global data). The TNC then reduces the feed rate as per your definition at transitions and narrow places, so the machine time should be reduced in total.

The 'samerine'izz_delit, HU Overview Tool Pile.pare. *	s
Number Number Name "" Spindle © No3 C () No4 Tool preselect	



Additional parameters on the **contour** detail form:

- Depth: Separately definable depths for each subcontour (FCL 2 function)

ᇞ

8

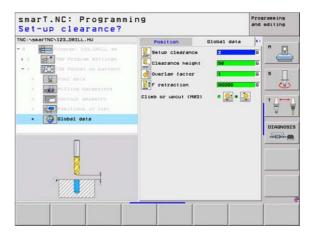
.....

- As a rule, always start the list of subcontours with the deepest pocket!
- If the contour is defined as an island, the TNC interprets the entered depth as the island height. The entered value (without an algebraic sign) then refers to the workpiece top surface!
- If the depth is entered as 0, then for pockets the depth defined in the overview form is effective. Islands then rise up to the workpiece top surface!

Globally effective parameters on the global data detail form:

- Set-up clearance
- 2nd set-up clearance
- Overlap factor
- Retraction feed rate
- Climb milling, or
- ▶ Up-cut milling

INC: NSBBTTNCN123_DRILL.HU	Tool Milg.para. Contour Contour name	geosetry O H
• 1 700 Program settings		• 🖻 🧲
* + Ist Pocket on pattern	Depth	• 🔤 ° 🗖 S 🔽
• Tool data	Depth	• 🖃 • 📃 🤳
Contour geometry Continue in 1111	Depth	• 🖃 • 🖾 🔰 🖓
- COICOSI data	Depth	
	Depth	• 🔤 • 🗖 🔤
	Depth	• • •
	Depth	• 🖬 • 🗖 —
	Depth	• <u>-</u> - <u>-</u>



Surfaces machining group

The following working unit is available for surfacing operations in the Surfaces machining group:

Unit	Soft key	Page
Unit 232 Face Milling	UNIT 232	Page 100

TRC:vsearTNCv122.DELL.HU • ●	X +0 +1	MAX point 100	
---------------------------------	---------	------------------	--



Unit 232 Face Milling

Parameters on the **overview** form:

- ▶ T: Tool number or name (switchable via soft key)
- S: Spindle speed [rpm] or cutting speed [m/min or ipm]
- ▶ F: Feed rate for plunging [mm/min] or FU [mm/rev] or FZ [mm/tooth]
- **Milling plan:** Selection of the milling plan.
- **Starting point in 1st axis:** Starting point in the reference axis.
- **Starting point in 2nd axis:** Starting point in the auxiliary axis.
- **Starting point in 3rd axis:** Starting point in the tool axis.
- **End point in 3rd axis:** End point in the tool axis.
- ▶ Allowance in depth: Finishing allowance for the depth.
- Ist side length: Length of the surface to be milled in the reference axis, referenced to the starting point.
- 2nd side length: Length of the surface to be milled in the auxiliary axis, referenced to the starting point.
- Maximum infeed: Maximum infeed per cut.
- Clearance to side: Lateral distance by which the tool moves beyond the surface.

Tocivestive: The search transformer is a second to the second terms of terms	Overview Tool Nile.per T B S C Hilling plan Hilling plan Startne pnt 1st axis Startne pnt 2rd axis Fred coint 2rd axis	e, Global da	s _
Nuxber	hilowno in denth hilowno in denth Ist side length And side length Haximum infeed Clearance to side	0 +60 +20 5 2	

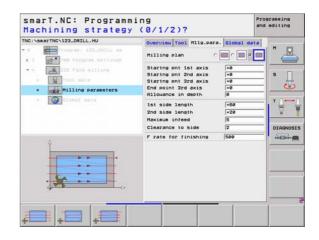
Additional parameters on the **tool** detail form:

- **DL:** Delta length for tool T.
- **DR:** Delta radius for tool T.
- **DR2:** Delta radius 2 (corner radius) for tool T.
- **M** function: Any miscellaneous function M.
- **Spindle:** Direction of spindle rotation. As a default, smarT.NC sets M3.
- Tool preselect: If needed, this is the number of the next tool for faster tool change (machine-dependent).

Additional parameters on the milling parameters detail form:

F rate for finishing: Feed rate for the last finishing cut.

INC:\searTNC\123_DRILL.HU	Overview Tool Milg.para. Global de	ta l
Portan 122.00114 es Se Program meltine Tool data Tool data	T III IIIIIIIIIIIIIIIIIIIIIIIIIIIIIIII	
		ECT TOOL NAME



i



Globally effective parameters on the ${\tt global}$ data detail form:

- Set-up clearance
- ▶ 2nd set-up clearance
- Positioning feed rate
- Overlap factor

smarT.NC: Programm. Set-up clearance?	ing	Program and edi	
THE 'SBEFTHC'123.DELL.HU	Overview Tool M130.pare, Blobal	6	

i

Probing main group

In the Probing main group you select the following function groups:

Function group

ROTATION

Touch probe functions for automatic determination of a basic rotation

PRESET

Touch probe functions for automatic determination of a reference point

MEASURING

Touch probe functions for automatic workpiece measurement

DIVERSE FUNCTION:

Special function for setting touch-probe data

DIVERSE FUNCTION

Soft key

ROTATION

PRESET

MEASURING

TOOL Touch probe functions for automatic tool measurement



For a detailed description of the probing cycles, see the Touch Probe Cycles Manual, Chapter 4.

TNC:VSB#2TNCV122.0RILL.HU • 0 Porres: 122.0RILL es • 1 Porres: 122.0RILL es • 1 Porres: 122.0RILL es	Y +8	MAX point 1000	
---	------	-------------------	--

Defining Machining Operations

Ť

Unit

Rotation function group

The following working units for automatic determination of a basic rotation are available in the Rotation function group:

	smarT.NC: Programm	ing	Programming and editing
rmination of a basic roup: Soft key	TNC:\SearTNC\123_DRILL.HU • 0 Program: 122_DRILL as • 1 Program: 122_DRILL as	Tool axis Overview Vorbsies blank Colio Disensions of blank NIN point NIX point X Te Preset Preset Coloring to the preset number Coloring to the preset numb	
UNIT 493		AIT 492 URIT 485	

- ----

Unit 402 Rotation around two Studs

Unit 401 Rotation around two Holes

Unit 400 Rotation around a Straight Line

Unit 403 Rotation around a Rotary Axis

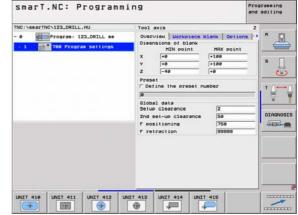
Unit 405 Rotation around the C Axis



Preset (Datum) function group

The following working units for automatic datum setting are available in the Preset function group:

Unit	Soft key
Unit 408 Slot Center Reference Point (FCL 3 Function)	
Unit 409 Ridge Center Reference Point (FCL 3 Function)	UNIT 489
Unit 410 Datum Inside Rectangle	UNIT 418
Unit 411 Datum Outside Rectangle	UNIT 411
Unit 412 Datum Inside Circle	UNIT 412
Unit 413 Datum Outside Circle	UNIT 413
Unit 414 Datum Outside Corner	UNIT 414
Unit 415 Datum Inside Corner	UNIT 415
Unit 416 Datum in Circle Center	UNIT 416





Unit	Soft key
Unit 417 Datum in Touch Probe Axis	UNIT 417
Unit 418 Datum from 4 Holes	
Unit 419 Datum in One Axis	UNIT 419

Measuring function group

The following working units for automatic workpiece measurement are available in the Measuring function group:

Unit	Soft key
Unit 420 Measure Angle	UNIT 428
Unit 421 Measure Hole	UNIT 421
Unit 422 Measure Cylindrical Stud	UNIT 422
Unit 423 Measure Rectangle Inside	UNIT 423
Unit 424 Measure Rectangle Outside	UNIT 424
Unit 425 Measure Width Inside	UNIT 425
Unit 426 Measure Width Outside	UNIT 426
Unit 427 Measure Coordinate	UNIT 427

TRC:VsearTRC:V22_DRIL,HU • 0	V +0	HAX point +100 +100	
---------------------------------	------	---------------------------	--



Unit	Soft key
Unit 430 Measure Bolt Hole Circle	UNIT 438
Unit 431 Measure Plane	UNIT 431

i

Diverse functions function group

The following working unit is available in the special functions function group:

Soft key

UNIT 441

Unit

Unit 441 Touch Probe Parameters

TNC:\searTNC\123_DRILL.HU	Tool axis	2	
- 0 Program: 123_DRILL mm - 1 24 700 Program settings	Overview Workpiece bi Disensions of blank MIN point X +8	HAX point	
Workpiece bl of the X dir	ank definition: Seallest co ection. Note the algebraic 2 -40 Preset F Define the preset num	1+8	S
	e Global data Setup clearance 2nd set-up clearance	[2 [50	DIRGNOS
	F positioning F retraction	99999 99999	- meret



Tool function group

The following working units for automatic tool measurement are available in the Tool function group:

Unit	Soft key				
Unit 480 TT: Calibrate TT	UNIT 498	smarT.NC: Programm	ing		rogramming and editing
Unit 481 TT: Measure Tool Length	UNIT 481	TNC:\smarTNC\123_DRILL.HU - 0 program: 123_DRILL ms - 1 program: settings		HAX point +100 +100	
Unit 482 TT: Measure Tool Radius	UNIT 482		Preset C Define the preset nu (e Global data		
Unit 483 TT: Measure Entire Tool	UNIT 483		Setup clearance 2nd set-up clearance F positioning F retraction	2 50 750 89989	OIAGNOSI
	U.				-
		UNIT 400 UNIT 401 UNIT 402 UN COL. A A A A	NIT 403	1	

110

i

Conversion main group

The following functions for coordinate transformation are available in the Conversion main group:

Function	Soft key	Page
UNIT 7 (FCL 2 function): Datum shift using the datum table	UNIT 7	Page 112
UNIT 8 (FCL 2 function): Mirroring		Page 112
UNIT 10 (FCL 2 function): Rotation	UNIT 18	Page 113
UNIT 11 (FCL 2 function): Scaling	UNIT 11	Page 113
UNIT 140 (FCL 2 function): Tilting the working plane with the PLANE function	UNIT 148	Page 114
UNIT 247: Preset number	UNIT 247	Page 116
UNIT 404 (2nd soft-key row): Set basic rotation	UNIT 484	Page 116

TRC:VS##TTRC/122_DRTLL,HU • 0	Tool axis Overview Workpiece blank Ortion Disensions of blank NUX point X re r	
----------------------------------	--	--

Defining Machining Operations



Unit 7 Datum shift (FCL 2 function)



Before you use unit 7 you must select in the program head the datum table in which smarT.NC will apply the datum numbers (see "Program settings" on page 35.).

Reset datum shift: Define unit 7 with number 0. Ensure that in line 0 all coordinates are defined as 0.

If you want to define a datum shift by entering coordinates, use the conversational unit (see "Unit 40 Conversational Unit" on page 120.).

With unit 7, datum shift, you define a datum number from the datum table that you have specified in the program header. Select the datum number via soft key.

Unit 8 Mirroring (FCL 2 function)

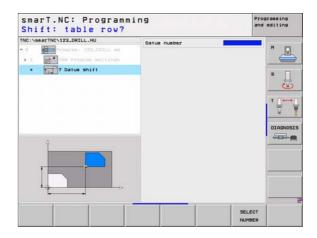
With unit 8 you use check boxes to define the desired mirrored axes.



If you define only one mirrored axis, the TNC changes the machining direction.

Reset mirroring: Define unit 8 without mirrored axes.

INC:\Sea	TINC 123_DRIL	L.HU	Dation	nusber		_	
• •		122.09311 es		100000			" _
•	7 Datus :						s 🗌
FORM_	40_DATASELECTI	V	Z	0	c	lu.	6
0	+0	+0	+0	+0	+0	+0	
1	+25	+333	+0	+0	+0	+25	
2	+10	+0	+0	+0	+0	+0	
2	+10	+8	+158	+8	+0	+0	
4	+27.25	+12.5	+0	-10	+0	+0	
5	+250	+325	+10	+0	+98	+0	
6	+258	-248	+15	+0	+0	+0	
7	+1200	+0	+8	+9	+0	+0	+
						ок	Cancel
BEGIN	END	PAGE	PAGE		1	1	ENI



Defining Machining Operations

Unit 10 Rotation (FCL 2 function)

With unit 10, rotation, you define an angle of rotation by which smarT.NC rotates the subsequently defined operations in the active working plane.

Before Cycle 10, at least one tool call must be programmed including definition of the tool axis so that smarT.NC can find the plane of rotation.

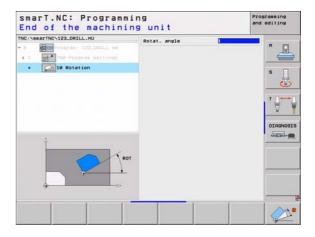
Reset rotation: Define unit 10 with rotation 0.

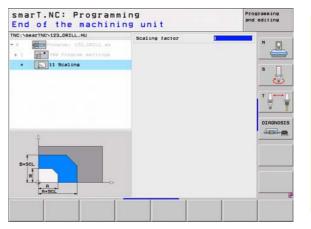
Unit 11 Scaling (FCL 2 function)

With unit 11 you define a scaling factor with which you can run the subsequently defined operations with enlarged or reduced dimensions.

With machine parameter MP7411 you specify whether the scaling factor is to be effective only in the active working plane or also in the tool axis.

Reset scaling factor: Define unit 11 with scaling factor 1.





Unit 140 Tilt plane (FCL 2 function)

- (Ÿ)

The machine manufacturer must enable the functions for tilting the working plane!

You can only use the PLANE function on machines that have at least two tilting axes (head and/or table). Exception: The **PLANE AXIAL** function (FCL 3 function) can also be used if only a single rotary axis is present or active on your machine

With unit 140 you can define tilted working planes in various ways. You can set the plane definition and positioning behavior separately.

TNC:\SearTNC\123_DRILL.HU	Plane definition plane definition plane sparse Spatial angle R Spatial angle C	* _
X1- A.	Positioning behavior prove Setup clearance r positioning Selection tild direction * Rutomatic C Positive C Negative	
	Selection type of transformation # Automatic C Rotate table/head C Rotate coordinate system	

114

The following plane definitions are available:

Type of plane definition	Soft key
Plane defined by spatial angle	SPATIAL
Plane defined by projection angle	PROJECTED
Plane defined by Euler angle	EULER
Plane defined by vector	VECTOR
Plane defined by three points	POINTS
Define an incremental spatial angle	REL. SPA.
Define axis angles (FCL 3 function)	AXIAL
Reset the tilted plane function	RESET

You can use soft keys to switch the positioning behavior, the selection of the tilt direction and the type of transformation.

_	ų	
Г		Γ

The type of transformation is effective only in transformations with a C axis (rotary table).

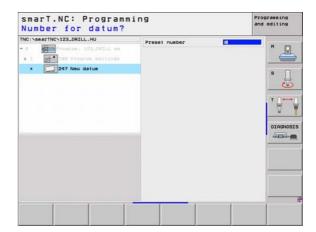


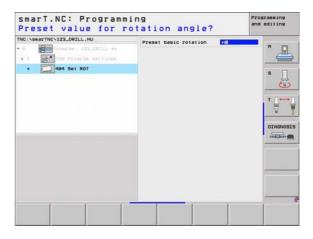
Unit 247 New Datum

With 247 you define a reference point from the active preset table.

Unit 404 Set Basic Rotation

With unit 404 you can set any basic rotation. Use this unit primarily to reset basic rotations that you have specified through probing functions.





Ť

Special functions main group

The following varied functions are available in the special functions main group:

Function	Soft key	Page
UNIT 151: Program call	UNIT 151	Page 118
UNIT 50: Separate tool call	UNIT 58	Page 119
UNIT 40: Conversational unit	LINIT 40	Page 120
UNIT 700 (2nd soft-key row): Program settings	LINIT 700	Page 35

Preset F Define the preset is Global data Setup Clearance 2nd set-up Clearance 7 positioning F retraction	2	
	B Global data Setup clearance 2nd set-up clearanc F positioning	je Global data Setup clearance 2 2nd set-up clearance (50 F positioning 750



Defining Machining Operations

Unit 151 Program Call

From smarT.NC you use this unit to call any program of the following file type:

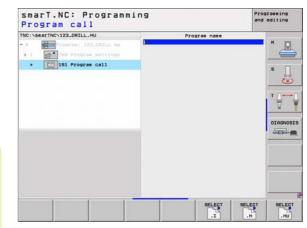
- smarT.NC unit program (file type .HU)
- Conversational dialog program (file type .H)
- ISO program (file type .I)

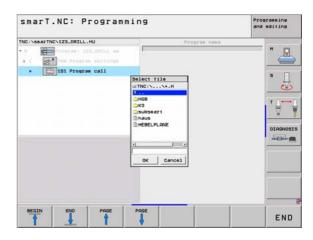
Parameters in the overview form:

Program Name: Enter the path and name of the program to be called.



- If you want to call the desired program via soft key (pop-up) window, see figure at bottom right), then the program must be saved in the TNC:\smarTNC directory.
- If the desired program is not stored in the TNC:\smarTNC directory, you must enter the complete path.





Defining Machining Operations

Unit 50 Separate Tool Call

You can define a separate tool call with this unit.

Parameters in the overview form:

- ▶ T: Tool number or name (switchable via soft key)
- **S:** Spindle speed [rpm] or cutting speed [m/min or ipm]
- **DL:** Delta length for tool T.
- **DR:** Delta radius for tool T.
- **DR2:** Delta radius 2 (corner radius) for tool T.
- **Define M function:** If desired, enter any miscellaneous functions M.
- Define pre-positioning: If desired, enter a position to be approached after the tool change. Positioning sequence: First the machining plane (X/Y), then tool axis (Z).
- Tool preselect: If needed, this is the number of the next tool for faster tool change (machine-dependent).

INC:\smarTNC\123_DRILL.HU O O Totrarae 122_DRILL es 1 Totrarae 122_DRILL es 5 Totrarae 5 Totrae 5 Totrae 5 Totrae 5 Totrarae 5	T 20 5		
	DRZ	-	- 🕹
	F Define M function M Function: M Function:	8	= ' 0 0
Nusber	Coordinate X Coordinate X Feed rate Coordinate Z Feed rate	FHRX	
	Tool preselect		
		SELE	TOOL NAME



Defining Machining Operations

Unit 40 Conversational Unit

Use this unit to insert conversational dialog sequences between machining blocks. It can always be used in the following cases:

You require TNC functions for which form entry is not yet availableYou want to define OEM cycles

You want to insert any positioning movements between units

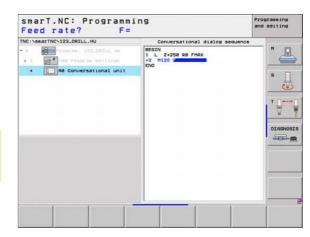
Vou want to define machine-specific M functions

빤

There is no limit to the number of conversational programming blocks that can be entered per conversational dialog sequence.

The following conversational functions, for which no form input is possible, can be inserted:

- Path functions L, CHF, CC, C, CR, CT, and RND via the gray path function keys
- STOP block via the STOP key
- Separate M-function block via ASCII key M
- Tool call with the TOOL CALL key
- Cycle definitions
- Touch-probe cycle definitions
- Program section repeats/subprogram technique
- Q parameter programming



Defining Machining Positions

Fundamentals

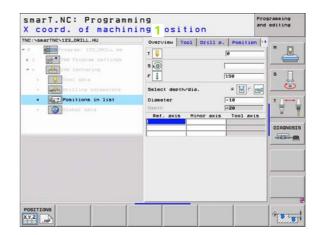
On the **overview** form (1) you can directly define the machining positions of the current machining step in Cartesian coordinates (see figure at top right). If machining is to be performed at more than three positions, you can define up to six more positions—for a total of nine—on the **Positions** detail form (2).

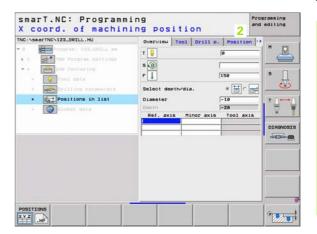
Incremental input is allowed beginning with the 2nd machining position. You can use the I key or soft key to switch over. The first machining position must be absolute.

The fastest, easiest and most accurate way of defining machining positions is with the pattern generator. The pattern generator immediately displays the entered machining positions graphically after the required parameters have been entered and saved.

smarT.NC automatically saves in a point table (.HP file) the machining positions you defined using the pattern generator. This point table can be used as often as you like. A very convenient feature is the possibility of hiding or disabling any machining positions by graphically selecting them.

Point tables that you may have used on older controls can also be used with smarT.NC.





121

Starting the pattern generator

The pattern generator for smarT.NC can be started two different ways:

- Directly from the third soft-key row of the smarT.NC main menu, if you want to directly define several point files in a row
- From the form during the machining definition, if you want to enter machining positions.

Starting the pattern generator from the main row of the editing menu



- Select the smarT.NC operating mode.
- POSITIONS
- Select the third soft-key row.
- Start the pattern generator: smarT.NC switches to the file manager (see figure at right) and shows any existing point files.
- Select an existing point file (*.HP) and open it with the ENT key, or



Create a new point file: Enter the file name (without the file type), and confirm with the MM or INCH key. smarT.NC creates a point file with the units of measurement you selected, and then starts the pattern generator.

	C: File managemen			and editing
NC:\searTNC	FR1.HP			
	TNC: \searTNC			8 13
320 30088F	File name	+ Size Ch	anged 5	tat -
DOGRAF	Scap_eso_2005	17078 25	.09.05 05:33	
	CONT1	812 25	. 10.05 10:16	
DEMO	ECOORD	839 92	.12.05 07:46	s 🗆
DUMPPEN	IECO0RD1	1390 02	.12.05 09:47	
dxf	IECPOC1	1130 00	.12.05 00:22	🙂
		1020 26	.07.05 13:53	
CH1	CPOCKLINKS	130 29	.04.05 06:29	
CHGB	CSTUDLINKS	124 29	.04.05 06:29	1 2 2 7
	DREIECKRECHTS	150 29	.04.05 05:29	¥
> ONEWDEMO	m flansch	478K 17	.11.05 09:50	
PENDELN	EFR1	2888 11	.04.05 15:15	DIRENOS
SCHULE	BEARUHEEL	16573 05	. 84.84 14:23	
- DsearTNC	GEARNHEEL	3588 22	.09.05 08:30	120,000
and the second s	E GEARUHEEL	815 12	.10.05 14:37	
+++++	I AREN	876 89	.07.05 07:32	
4 the	P Ibhaus	2622 21	.03.05 14:05	
	HEBEL	512 09	.07.05 15:46	
2 1 1	IEHEBEL	772 18	.05.05 07:07	
X * 1	THEBELPLANE	914 27	.84.85 87:56	
44	IBHEBELPOC		.07.05 09:04	
4 T+44+	HEBELSTUD	150 29	.04.05 05:29	···· •
++++++	+ 79 Objects / 6570.0KBytes /	21214.9HBytes		
PAGE	PAGE SELECT COPY	SELECT	NEW LAS	
A		CPPD .	FILE FIL	END

Starting the pattern generator from a form



- Select the smarT.NC operating mode.
- Select any machining step in which machining positions can be defined.
- Select an input field in which a machining position is to be defined (see figure at top right).
- Switch to **Define machining positions in point table**.



- To create a new file: Enter the file name (without the file type), and confirm with the NEW .HP soft key.
- Specify the units of measurement for the new point file with the MM or INCH button in the pop-up window. smarT.NC then starts the pattern generator.



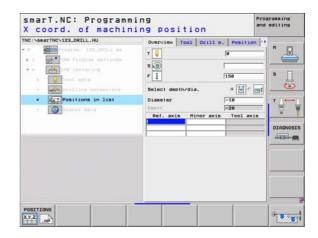
To select an existing HP file: Press the SELECT .HP soft key. smarT.NC opens a pop-up window with available point files. Select one of the displayed files, and open it with the ENT key or OK screen button.

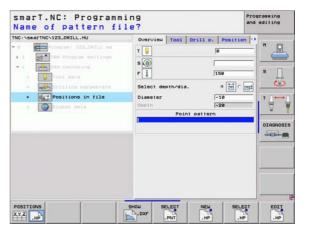


- To edit an existing HP file: Press the EDIT .HP soft key. smarT.NC starts the pattern generator.
 To select an existing PNT file: Press the SELECT .PNT soft key.
- key. smarT.NC opens a pop-up window with available point files. Select one of the displayed files, and open it with the ENT key or OK screen button.



If you want to edit a .PNT file, smarT.NC converts it to an .HP file! Answer the dialog prompt with OK.





i

Exiting the pattern generator



- Press the END key or soft key. smarT.NC opens a pop-up window (see figure at right).
- Press the ENT key or YES button to save all changes—or to save a newly created file—and to exit the pattern generator.
- Press the NO ENT key or NO button to discard all changes, and to exit the pattern generator.
- ▶ Press the ESC key to return to the pattern generator.



If you started the pattern generator from a form, then you automatically return to that form after exiting the generator.

If you started the pattern generator from the main menu, then you automatically return to the last selected .HU program after exiting the generator.

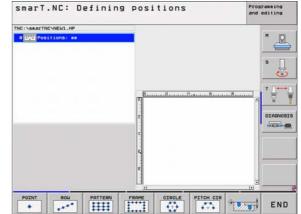
			nd editing
0 Postions an	Pps. Ref. acts	ninor axis	
• A1 III Film circle	Retraction height	1.000	
1.1 + Position			s
1.0 • Pasifier			
	End the pattern generator		. ` ₽
	Save the changes?		- <u> </u>
	Yes No Cancel		DIAGNOS
		_	-
	1		1
. 1.	1 +	-0-	
+			
1. A.	Li		J
			1

Working with the pattern generator

Overview

The following possibilities are available in the pattern generator for defining machining positions:

Function	Soft key	Page
Single point, Cartesian	POINT	Page 130
Single row, straight or arced	ROU	Page 130
Pattern straight, arced or distorted	PATTERN	Page 131
Frame straight, arced or distorted	FRAME	Page 132
Full circle	CIRCLE	Page 133
Circular arc	PITCH CIR	Page 134
Change starting height		Page 135





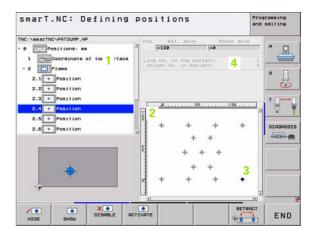
Defining a pattern

- Select via soft key the pattern to be defined.
- ▶ Define the required entry parameters in the form. Select the next input field with the ENT key or the arrow down key.
- Press the END key to save the entered parameters.

After you have entered a pattern via a form, smarT.NC displays this pattern symbolically as an icon on the left side of the screen in the tree view (1).

The pattern is displayed graphically in the bottom right quarter of the screen (2) immediately after the entry parameters have been saved.

After opening the tree view with the right arrow key, you can select any point within the pattern you've created with the arrow down key. The point selected on the left is shown in blue in the graphic on the right by smarT.NC (3). For informational purposes, the Cartesian coordinates of the currently selected point are shown in the top right quarter of the screen (4).



Functions of the pattern generator

Function

Hide the pattern or position selected in the tree view for machining. Hidden patterns or positions are marked in the tree view with a red slash, and in the preview graphics with a bright red dot.

Reactivate a hidden pattern or position.

Disable the position selected in the tree view for machining. Disabled positions are marked in the tree view with a red x. smarT.NC does not display disabled positions in the graphic at all. These positions are not saved in the .HP file that smarT.NC creates as soon as you exit the pattern generator.

Reactivate disabled positions

Export the defined machining positions to a PNT file. Necessary only if you want to use the machining pattern on older software levels of the iTNC 530

Show only the pattern selected in the tree view, or show all defined patterns. smarT.NC shows the pattern selected in the tree view in blue.



Soft key

1.

HIDE

SHOU

X .

ACTIVATE

PREVIEW

COMPLETE

smarT.NC: Defining	positions		Programming and editing
TNC:\searTNC\PATDUMP.HP - 0 [[]]]Positions: me	Pos. Ref. axis	Minor axis	
1 Coordinate of top surface 2 1 rate 2.1 Position 2.2 Position	Line no. In the patt Column no. in patte		1 4 5
2.3 + Position 2.4 + Position 2.5 + Position	1		- - - -
2.6 Position		* * *	DIAGNOS
•	* * * *	+ +	
		RETRAC	END

-

Function	Soft key
Display or hide rulers	
Go to previous page	PAGE
Go to next page	PAGE
Go to beginning of file	BEGIN
Go to end of file	
Zoom function: Shift zoom area up (last soft-key row)	t
Zoom function: Shift zoom area down (last soft-key row)	ţ
Zoom function: Shift zoom area to the left (last soft-key row)	÷
Zoom function: Shift zoom area to the right (last soft-key row)	
Zoom function: Magnify workpiece. The TNC always magnifies the center of the view currently being displayed. Use the scroll bars to position the drawing in the window so that the desired section appears after the soft key has been pressed (last soft-key row).	*

Function

Soft key

-

Zoom function: Reduce the workpiece (last soft-key row)

Zoom function: Show workpiece in original size (last soft-key row)





Single point, Cartesian



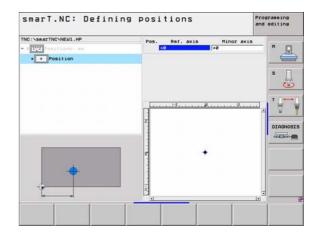
- ► X: Coordinate in the reference axis of the working plane
 - > Y: Coordinate in the minor axis of the working plane

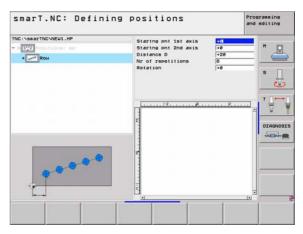
Single row, straight or arced



+

- Starting point 1st axis: Coordinate of the starting point of the row in the reference axis of the working plane.
- Starting point 2nd axis: Coordinate of the starting point of the row in the minor axis of the working plane.
- **Distance:** Distance between the machining positions. You can enter a positive or negative value.
- Number of repetitions: Total number of machining positions.
- Rotation: Angle of rotation around the entered starting point. Reference axis: Major axis of the active machining plane (e.g. X for tool axis Z). You can enter a positive or negative value.





Pattern, straight, rotated or distorted



- Starting point 1st axis: Coordinate of the starting point of the pattern (1) in the major axis of the working plane.
- Starting point 2nd axis: Coordinate of the starting point of the pattern (2) in the minor axis of the working plane.
- Distance in 1st axis: Distance of the machining positions in the major axis of the working plane. You can enter a positive or negative value.
- Distance in 2nd axis: Distance of the machining positions in the minor axis of the working plane. You can enter a positive or negative value.
- **Number of lines:** Total number of rows in the pattern.
- **Number of columns:** Total number of columns in the pattern.
- Rotation: Angle of rotation by which the entire pattern is rotated around the entered starting point. Reference axis: Major axis of the active machining plane (e.g. X for tool axis Z). You can enter a positive or negative value.
- Rotary pos. ref. ax.: Angle of rotation around which only the major axis of the machining plane is distorted around the entered starting point. You can enter a positive or negative value.
- Rotary pos. minor ax.: Angle of rotation around which only the minor axis of the machining plane is distorted around the entered starting point. You can enter a positive or negative value.



The **Rotary pos. ref. ax.** and **Rotary pos. minor ax.** parameters are added to a previously performed **rotation** of the entire pattern.

C VARANTACVNEUL IPP	Startno pni tist axis Startno pni žnd axis Boacing in 1st axis Boacing in 1st axis Number of lines Number of columns Rotation Rotation Rotatiy pos.ref. ax. Rotaty pos.ref. ax.	+6 +0 +20 +10 6 4 +0 +0 +0 +0 +0 +0 +0	
$\bullet \bullet \bullet \bullet \bullet$	diam'r		



Frame straight, rotated or distorted



- Starting point 1st axis: Coordinate of the starting point of the frame (1) in the major axis of the working plane.
- Starting point 2nd axis: Coordinate of the starting point of the frame (2) in the minor axis of the working plane.
- Distance in 1st axis: Distance of the machining positions in the major axis of the working plane. You can enter a positive or negative value.
- Distance in 2nd axis: Distance of the machining positions in the minor axis of the working plane. You can enter a positive or negative value.
- Number of lines: Total number of rows in the frame.
- **Number of columns:** Total number of columns in the frame.
- Rotation: Angle of rotation by which the entire frame is rotated around the entered starting point. Reference axis: Major axis of the active machining plane (e.g. X for tool axis Z). You can enter a positive or negative value.
- Rotary pos. ref. ax.: Angle of rotation around which only the major axis of the machining plane is distorted around the entered starting point. You can enter a positive or negative value.
- Rotary pos. minor ax.: Angle of rotation around which only the minor axis of the machining plane is distorted around the entered starting point. You can enter a positive or negative value.



The **Rotary pos. ref. ax.** and **Rotary pos. minor ax.** parameters are added to a previously performed **rotation** of the entire frame.

smarT.NC: Defining	Staring pht 1st axis Staring pht 2nd axis Specing in 1st axis Specing in 2nd axis Number of lines Number of columns Relation Relaty pos. ref. ax. Relaty pos.sinc ax.	+08 +20 +120 6 6 4 +0 +0 +0 +0	and editing
		[

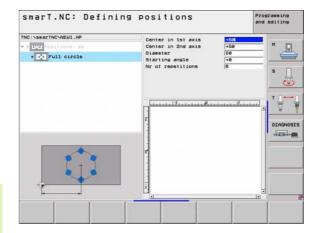
Full circle



ᇞ

- Center in 1st axis: Coordinate of the circle center point (1) in the major axis of the working plane.
- Center in 2nd axis: Coordinate of the circle center point
 (2) in the minor axis of the working plane.
- **Diameter:** Circle diameter.
- Starting angle: Polar angle of the first machining position. Reference axis: Major axis of the active machining plane (e.g. X for tool axis Z). You can enter a positive or negative value.
- Number of repetitions: Total number of machining positions on the circle.

smarT.NC always calculates the angle increment between two machining positions by dividing 360° by the number of machining operations.

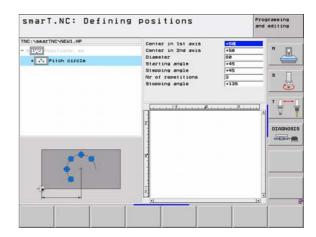




Circular arc



- **Center in 1st axis:** Coordinate of the circle center point (1) in the major axis of the working plane.
- Center in 2nd axis: Coordinate of the circle center point
 (2) in the minor axis of the working plane.
- **Diameter:** Circle diameter.
- Starting angle: Polar angle of the first machining position. Reference axis: Major axis of the active machining plane (e.g. X for tool axis Z). You can enter a positive or negative value.
- Stepping angle: Incremental polar angle between two machining positions. You can enter a positive or negative value. Changing the stepping angle automatically changes the defined stopping angle.
- Number of repetitions: Total number of machining positions on the circle.
- Stopping angle: Polar angle of the last bore hole. Reference axis: Major axis of the active machining plane (e.g. X for tool axis Z). You can enter a positive or negative value. Changing the stopping angle automatically changes the stepping angle if one was defined.



Change starting height



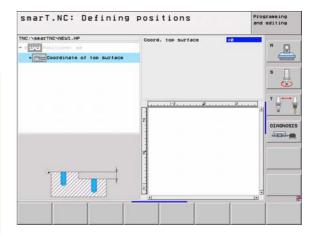
Top surface coordinate: Coordinate of the top surface of the workpiece.

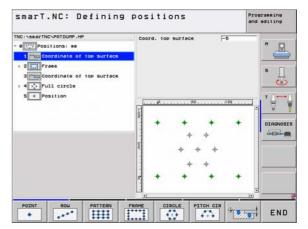


If you do not define a starting height in the definition of the machining positions, smarT.NC always sets the coordinate of the workpiece surface to 0.

If you change the starting height, then the new starting height is valid for all subsequently programmed machining positions.

If you select the symbol for the top surface coordinate in the tree view, then all machining positions for which this starting height is valid turn green in the preview graphic (see figure at bottom right).





i

Defining a retraction height for positioning (FCL 3 Function)

Use the arrow keys to select any single position that is to be approached at a height you define.



Retraction height: Enter the absolute coordinate at which the TNC is to approach this position. The TNC marks the position with an additional circle.

The retraction height you define is always referenced to the active datum.

positions	Program	
[+5a [+9a	H	
		1
	-	0
		IAGNOS
+	+ -	
	Pos Bef. exis Pinor F58 (+98 Retraction height Side +	and es

ĺ

Defining Contours

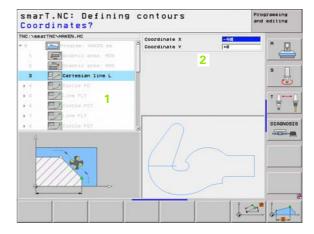
Fundamentals

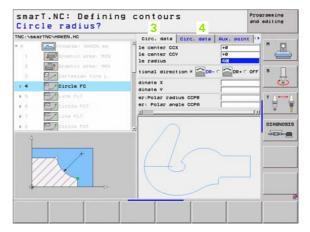
Contours are defined in separate files (file type **.HC**). Since .HC files contain pure descriptions of contours—only geometry data, no technology data—they can be used flexibly: as contour trains, as pockets or as islands.

You can create HC files either with the path functions or by using the DXF converter (software option) to import it from existing DXF files.

Existing contour descriptions in older plain-language programs (.H files) can easily be converted into smarT.NC contour descriptions (see Page 146).

Just as with unit programs and the pattern generator, smarT.NC displays each contour element in the tree view (1) with an appropriate icon. Enter the data for each contour element in the form (2). In the FK free contour programming, along with the overview form (3) there are up to three additional detail forms (4) in which you can enter data (see figure at bottom right).





i

Starting the contour programming

The contour programming for smarT.NC can be started two different ways:

- Directly from the main row of the editing menu, if you want to define several separate contours in a row.
- From the form during the machining definition, if you want to enter the names of the contours to be edited.

Starting the contour programming from the main row of the editing menu



Defining Contours

- Select the smarT.NC operating mode.
- ▷ ▷ Selec
- Select the third soft-key row.



- Start contour programming: smarT.NC switches to the file manager (see figure at right) and shows any existing contour programs.
- Select an existing contour program (*.HC) and open it with the ENT key, or



- Create a new contour program: Enter the file name (without file type), and confirm with the MM or INCH key. smarT.NC opens a contour program with the units of measurement you selected.
- smarT.NC automatically inserts two lines for defining the drawing surface. If required, adjust the dimensions

smarT	. NC :	File ma	nageme	nt			ogramming d editing
TNC:\searTN	c					_	-
1220		=TNC:\searTNC	N. HC				8 13
D 320		File name		. 5	ize Changed	Stat	-
CIQUIT		1		<	Dir>		1
CIBHB		CHGB			Dir> 05.03.05	15:18	
DEMO		C)K3		<	Dir> 03.05.05	08:54	S
DUMPPO		Subseart		<	Dir> 02.12.05	11:29	+
Odxf		EC1			556 20.02.06		\sim
> CFK	1	C125			388 29.04.05		1
CH1		C2			558 20.02.06		T
CHGB		C3			558 20.02.06		8
OMHL		CONT1			812 25.10.05		5 M 3
> ONEVDEM	5	CPOCKLINKS			130 29.04.05		-
. DPENDEL	()	CSTUDLINKS				e6:29	DIAGNOS
SCHULE		DREIECKRECHT	5		150 29.04.05		
" issarTN		GEARNHEEL			580 22.09.05		THE REAL
10000		MAPHAKEN			876 09.07.05 512 09.07.05		1
		HEBEL			158 29.84.85		-
		HOUSING1				09:58	
		HOUSING2			352 17.08.05		
		KONTUR			578 29.04.05		
		RETSLINKS			124 29.04.05		
		KREISRECHTS			124 18.07.05		
						10.1	-
		28 Objects / 1	5956 Bytes /	28004.48	Bytes free		
PAGE	PAGE	SELECT	COPY	SELECT	NEU	LAST	
4			D.D.	CEPT	FILE	FILES	END
			ABC XYZ	TYPE		TAC.	

İ 138

Starting contour programming from a form



HC NEW

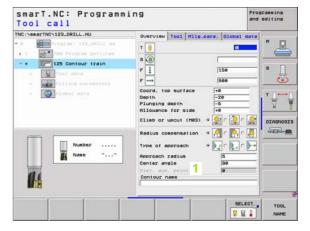
- Select the smarT.NC operating mode.
- Select any machining step for which contour programs are required (Unit 122, Unit 125).
- Select the input field in which the name of the contour program is to be defined (1, see figure).
- To create a new file: Enter the file name (without file type), and confirm with the NEW soft key.
- Specify the units of measurement for the new contour program with the MM or INCH button in the pop-up window: smarT.NC creates a contour program with the units of measurement you selected, opens contour programming, and automatically takes over the workpiece blank definition specified in the unit program (definition of the drawing surface)

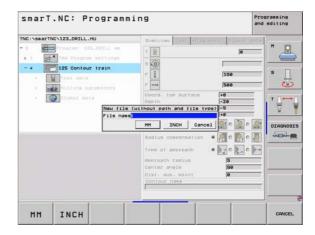


▶ To select an existing HC file: Press the SELECT HC soft key. smarT.NC opens a pop-up window with available contour programs. Select one of the displayed contour programs, and open it with the ENT key or OK button.



- ► To edit an existing HC file: Press the EDIT soft key. smarT.NC starts contour programming.
- To select an HC file with the DXF converter: Press the SHOW DXF soft key. smarT.NC opens a pop-up window with available DXF files. Select one of the displayed DXF files and confirm your selection with the ENT key or the OK button. The TNC starts the DXF converter, with which you select the desired contour and can save the contour name directly in the form (see "Processing DXF Files (Software Option)" on page 147).





i

Exiting the contour programming

- Press the END key: smarT.NC exits the contour programming and returns to the state from which you started contour programming: Either to the last active .HU program, if you started from the smarT.NC main row, or to the entry form of the machining step, if you started from the form.



If you started contour programming from a form, then you automatically return to that form after exiting the generator.

If you started contour programming from the main menu, then you automatically return to the last selected .HU program after exiting the programming.

Working with contour programming

Overview

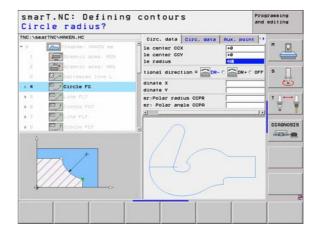
The contour elements are programmed using the familiar conversational dialog functions. Along with the gray path functions keys, the powerful FK free contour programming is also available. These forms are called via soft keys.

The support graphics, which are available for each input field and clarify which parameter is to be entered, are especially helpful for FK programming.

All familiar functions of the programming graphics are available in smarT.NC without restriction.

Dialog guidance in the forms is almost identical with that in conversational programming:

- The orange axis keys position the cursor in the desired input field
- With the orange I key you can switch between absolute and incremental programming
- With the orange P key you can switch between Cartesian and polar coordinate programming



FK Free Contour Programming

Workpiece drawings that are not dimensioned for NC often contain unconventional coordinate data that cannot be entered with the gray path function keys.

You can enter such dimensional data directly by using the FK free contour programming function. The TNC calculates the contour from the known contour information that you entered in the form. The following functions are available:

nformation that you entered in the form. The ble:		21 Circle 22 Circle 23 Circle Circle
on	Soft key	
t line with tangential connection	FLT	
t line without tangential connection		FL FLT
arc with tangential connection	FCT	
arc without tangential connection	FC	
r FK programming	FPOL	



Functio Straight

Straight

Circular

Circular

Pole for

Information about possible contour entries are in the tooltip that the TNC displays for each entry field (see "Mouse operation" on page 28). and in the Conversational User's Manual.

142

Functions

	smarT.NC: Defining contours	Programming and editing
Soft key	> 18 Line FLT	POINT P
RESET + START	10 Une FL 11e redius 40 110 110 redius 40 110 110 redius 100 redius 120 Line FL 110 redius 20 Rounding arc RND rdinate X	
START SINGLE	21 Line FL 12 Circle FC 22 Circle FC 22 Circle FC 22 Circle FC 22 Circle FC	
START		
STOP		START RESET SINGLE + START
<u>↑</u> ↓	SmarT.NC: Defining contours THG:NBBETHIC-HBKEN.HC 10 Circ. data Circ. data Aux. 1 10 Circ. data Circ. data Circ. data Aux. 1 10 Circ. data Circ	Programming and editing
	20 Conding arc RND rdinate X	
	+ 21 CLINE FL + 22 Coircle FC + 22 CLICE FC + 23 CLICE FCT - 24 CLICE FCT	
UINDOU BLK Form	45	
WINDOW DETAIL		
	RESET START START START STOP STOP STOP STOP UINOOU BLK FORM	Soft key RESET START START

Defining Contours

The different colors of the displayed contour elements indicate their validity:

Blue The contour element is fully defined.

Green The entered data describe a limited number of possible solutions: select the correct one.

Red The entered data are not sufficient to determine the contour element: enter further data.

Selecting from multiple possible solutions

If incomplete entries lead to multiple theoretically possible solutions, then (with graphic support) you can select the correct solution via soft key:

SHOU SOLUTION SELECT SOLUTION

Show the possible solutions.

Enter the displayed solution in the part program.

Enter data for subsequent contour elements.



Graphically display the next programmed block.

smarT.NC: Defining contours Programming and editing TNC:\searTNC\HAKEN.HC Circ. data Circ. data Aux. point 5 16 Line FLT the center CCX R +8 Solution FSELECT :le center CCV +0 17 :le radius 60 Line FL + 18 OR+ O OFF stional direction # COR- C Line FL > 19 志 rdinate X Rounding arc RND 20 rdinate V 1 21 Line FL ter:Polar radius CCPR н Circle FC ter: Polar angle CCPA 1 22 1 23 Circle FCT Circle FCT DIAGNOSIS 24 START SHOW SELECT END SINGLE SOLUTION SOLUTION SELECT

Defining Contours

i

Functions available for contour programming

Function	Soft key
Assume the workpiece-blank definition from the .HU program if you called the contour programming from a smarT.NC working unit.	USE WORKPIECE BLANK
Display or hide the block numbers	SHOU OHIT BLOCK NR.
Redraw the programming graphics, for example if lines were deleted by intersections	REDRAW
Clear programming graphics	CLEAR GRAPHICS
Graphically display programmed contour elements immediately after entry: Function OFF / ON	AUTO DRAU OFF ON



Converting existing conversational dialog programs into contour programs

In this procedure you must copy an existing conversational dialog program (.H file) into a contour description (.HC file). Since the two file types have a different internal data format, an ASCII file must be created as an intermediary for this copy procedure. Proceed as follows:



Select the Programming and Editing mode of operation.



- ▶ Call the file manager.
- Select the .H program to be converted



- Select the copy function: Enter *.A as the target file. The TNC creates an ASCII file from the conversational dialog program.
- Select the created ASCII file.



- Select the copy function: Enter *.HC as the target file. The TNC creates a contour description from the ASCII file.
- Select the newly created .HC file and remove all blocks except the BLK FORM workpiece blank definition—that do not describe contours.
- Remove programmed radius compensations, feed rates and M functions. The .HC file can now be used by smarT.NC.



Defining Contours

Processing DXF Files (Software Option)

Processing DXF Files (Software Option)

Function

DXF files created in a CAD system can be opened directly by the TNC, in order to extract contours or machining positions, and save them as conversational programs or as point files. Plain-language programs acquired in this manner can also be run by older TNC controls, since these contour programs contain only L and CC-/CP blocks.

The DXF files to be processed must be stored in the SMARTNC directory on the hard disk of your TNC.

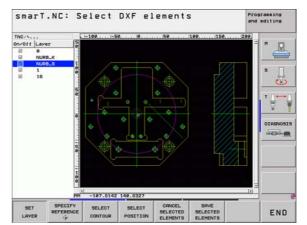
Before loading the file to the TNC, ensure that the name of the DXF file does not contain any blank spaces or impermissible special characters.

The DXF file to be opened must contain at least one layer.

The TNC supports the most common DXF format, R12 (equivalent to AC1009).

The following DXF elements are selectable as contours:

- LINE (straight line)
- CIRCLE (complete circle)
- ARC (circular arc)



i

Opening a DXF file

The DXF converter can be started in different ways:

- From the file manager if you want to extract several contour or position files consecutively.
- From the form during the definition of machining of Units 125 (contour train), 122 (contour pocket) and 130 (contour pocket on point pattern) if you are supposed to enter the names of the contours to be edited.
- While defining the machining, if you enter the machining positions via point files.

Starting the DXF converter through the file manager

- Select the smarT.NC operating mode.
- ▶ Call the file manager.
- SELECT
- In order to see the soft-key menu for selecting the file type
 - to be displayed, press the SELECT TYPE soft key.
 - ▶ To show all DXF files, press the SHOW DXF soft key.



E)

PGM MGT

TYPE

Select the desired DXF file, and load it with the ENT key. smarT.NC starts the DXF converter and shows the contents of the DXF file on the screen. The TNC shows the lavers in the left window, and the drawing in the right window.

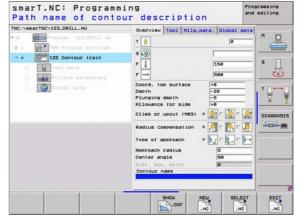
Starting the DXF converter from a form



SHOW

DXF

- Select the smarT.NC operating mode.
- Select any machining step for which contour programs or point files are necessary.
- Select the input field in which the name of the contour program or point file is to be defined.
- To start the DXF converter: Press the SHOW DXF soft key. smarT.NC opens a pop-up window with available DXF files. If necessary, select the directory in which the DXF file to be opened is saved. Select one of the displayed DXF files and confirm your selection with the ENT key or the OK button. The TNC starts the DXF converter, with which you select the desired contour or positions, and can save the contour name or the name of the point file directly in the form (see "Processing DXF Files (Software Option)" on page 147).



Basic settings

The third soft-key row has various possibilities for settings:

Setting

Show/hide rulers: The TNC shows the rulers at the left and top edges of the drawing. The values shown on the ruler are based on the drawing datum.

Show/hide status bar: The TNC shows the status bar at the bottom edge of the drawing. The following information is shown in the status bar:



Soft key

RULERS

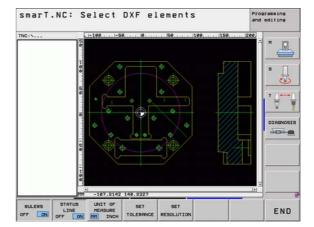
OFF ON

- Active unit of measurement (MM or INCH)
- X and Y coordinates of the current mouse position
- In the SELECT CONTOUR mode, the TNC shows whether the selected contour is open (open contour) or closed (closed contour).

Unit of Measure MM/INCH: Enter the unit of measurement of the DXF file. The TNC then outputs the contour program in this unit of measurement.



Set Tolerance: The tolerance specifies how far apart neighboring contour elements may be from each other. You can use the tolerance to compensate for inaccuracies that occurred when the drawing was made. The default setting depends on the extent of the entire DXF file.





Setting

Soft key

SET

RESOLUTION

Set Resolution: The resolution specifies how many decimal places the TNC should use when generating the contour program. Default setting: 4 decimal places (equivalent to resolution of 0.1 µm)

~	-
ÌÈ	7
4	3

Please note that you must set the correct unit of measurement, since the DXF file does not contain any such information.



Layer settings

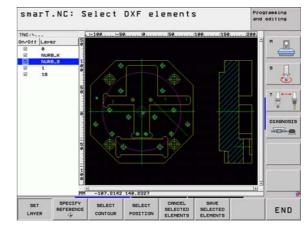
As a rule, DXF files contain multiple layers, with which the designer organizes the drawing. The designer uses the layers to create groups of various types of elements, such as the actual workpiece contour, dimensions, auxiliary and design lines, shadings, and texts.

So that as little unnecessary information as possible appears on the screen during selection of the contours, you can hide all excessive layers contained in the DXF file.

The DXF file to be processed must contain at least one layer.

You can even select a contour if the designer has saved it on different layers.

- SET LAVER
- If it has not already been activated, select the mode for the layer settings. In the left window the TNC shows all layers contained in the active DXF file.
- To hide a layer, select the layer with the left mouse button, and click its checkbox to hide it.
- To show a layer, select the layer with the left mouse button, and click its checkbox again to show it.



Processing DXF Files (Software Option)

Datum specifying

The datum of the drawing for the DXF file is not always located in a manner that lets you use it directly as reference point for the workpiece. Therefore, the TNC has a function with which you can shift the drawing datum to a suitable location by clicking an element.

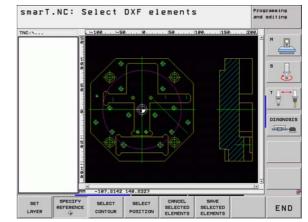
You can define a reference point at the following locations:

- At the beginning, end or center of a straight line
- At the beginning or end of a circular arc
- At the transition between quadrants or at the center of a complete circle
- At the intersection of a
 - straight line and a straight line, even if the intersection is actually on the extension of one of the lines
 - straight line and a circular arc
 - straight line and a complete circle
 - complete circle/arc and a complete circle/arc



You must use the touchpad on the TNC keyboard or a mouse attached via the USB port in order to specify a reference point.

You can also change the reference point once you have already selected the contour. The TNC does not calculate the actual contour data until you save the selected contour in a contour program.



Selecting a reference point on a single element



(B)

- Select the mode for specifying the reference point.
- Click the element on which you want to set the reference point with the left mouse button. The TNC indicates possible locations for reference points on the selected element with stars.
- Click the star you want to select as reference point. The TNC sets the reference-point symbol to the selected location. Use the zoom function if the selected element is too small.

Selecting a reference point on the intersection of two elements

SPECIFY	Select
REFERENCE	00.000

- the mode for specifying the reference point.
- Click the first element (straight line, complete circle or circular arc) with the left mouse button. The TNC indicates possible locations for reference points on the selected element with stars.
- Click the second element (straight line, complete circle or circular arc) with the left mouse button. The TNC sets the reference-point symbol on the intersection.

	-
Γ	F
5	Ľ

.

The TNC calculates the intersection of two elements even it is on the extension of one of these elements.

If the TNC calculates multiple intersections, it selects the intersection nearest the mouse-click on the second element.

If the TNC cannot calculate an intersection, it rescinds the marking of the first element.

Contour selection, saving a contour program

You must use the touchpad on the TNC keyboard or a mouse attached via the USB port in order to select a contour.

Select the first contour element such that approach without collision is possible.

If the contour elements are very close to one another, use the zoom function.

SELECT

- Select the mode for choosing a contour. The TNC hides the layers shown in the left window, and the right window becomes active for contour selection.
- To select a contour element, click the desired contour element with the left mouse button. The selected contour element turns blue. At the same time, the TNC marks the selected element with a symbol (circle or line) in the left window.
- ► To select the next contour element, click the desired contour element with the left mouse button. The selected contour element turns blue. If further contour elements in the selected machining sequence are clearly selectable, these elements turn green. Click on the last green element to assume all elements into the contour program. The TNC shows all selected contour elements in the left window. The TNC displays elements that are still green in the NC column without a check mark. This elements are not output to the contour program when it is saved.



If necessary you can also deselect elements that you already selected, by clicking the element in the right window again, but this time while pressing the CTRL key.



- To save the selected contour elements in a plain-language program, enter any file name in the pop-up window displayed by the TNC. Default setting: Name of the DXF file
- Confirm the entry: The TNC saves the contour program in the directory in which the DXF file is also saved.



ENT

If you want to select more contours, press the CANCEL SELECTED ELEMENTS soft key and select the next contour as described above.

The TNC also transfers the workpiece-blank definition **(BLK FORM)** to the contour program.

The TNC only saves elements that have actually been selected (blue elements).

If you call the DXF converter from a form, smarT.NC automatically closes the DXF converter after you have completed the SAVE SELECTED ELEMENTS function. Then smarT.NC writes the defined contour names to the input field from which you have started the DXF converter.

156

Processing DXF Files (Software Option)

Dividing, extending and shortening contour elements

If contour elements to be selected in the drawing connect bluntly, then you must first divide the contour element. This function is automatically available if you are in the mode for selecting a contour.

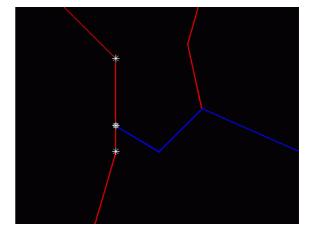
Proceed as follows:

- The bluntly connecting contour element is selected, so it is colored blue.
- Click the contour element to be divided: The TNC shows the point of intersection with a star in a circle, and the selectable end points with simple stars.
- Press the CTRL key and click the point of intersection: The TNC divides the contour element at the point of intersection and the stars disappear. If there is a gap or the elements overlap, the TNC extends or shortens the bluntly connecting contour element to the point of intersection of the two elements.
- Click the divided contour element again: The TNC shows the end points and points of intersection again.
- Click the desired end point: The TNC now colors the divided element blue.
- Select the next contour element.

ר _	Ţ
	ੜ
~	_

If the contour element to be extended or shortened is a straight line, then the TNC extends the contour element along the same line. If the contour element to be extended or shortened is a circular arc, then the TNC extends the contour element along the same arc.

In order to use this function, at least two contour elements must already be selected, so that the direction is clearly determined.



Selecting and storing machining positions



You must use the touchpad on the TNC keyboard or a mouse attached via the USB port in order to select a machining position.

If the positions to be selected are very close to one another, use the zoom function.

SELECT

Select the mode for choosing a machining position. The TNC hides the layers shown in the left window, and the right window becomes active for position selection.

- In order to select a machining position, click the desired element with the left mouse button. The TNC indicates possible locations for machining positions on the selected element with stars. Click one of the stars: The TNC loads the selected position into the left window (displays a point symbol).
- If you want to specify the machining position at the intersection of two elements, click the first element with the right mouse button: The TNC displays stars at the selectable machining positions.
- Click the second element (straight line, complete circle or circular arc) with the left mouse button. The TNC loads the intersection of the elements into the left window (displays a point symbol).

- SAVE SELECTED ELEMENTS
- To save the selected machining positions in a points file, enter any file name in the pop-up window displayed by the TNC. Default setting: Name of the DXF file
 Confirm the entry: The TNC saves the contour program in



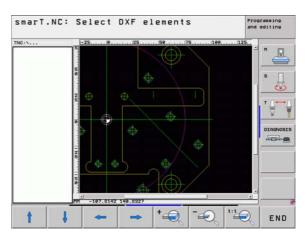
- CANCEL SELECTED ELEMENTS
- the directory in which the DXF file is also saved.
 If you want to select more machining positions in order to save them in a different file, press the CANCEL SELECTED ELEMENTS soft key and select as described above.

159

Zoom function

The TNC features a powerful zoom function for easy recognition of small details during contour or point selection.

Function	Soft key
Magnify workpiece. The TNC always magnifies the center of the view currently being displayed. Use the scroll bars to position the drawing in the window so that the desired section appears after the soft key has been pressed.	+
Reduce workpiece	-
Show workpiece at original size	1:1
Move zoomed area upward	t
Move zoomed area downward	ŧ
Move zoomed area to the left	+
Move zoomed area to the right	



160

1	

If you have a wheel mouse, you can use it to zoom in and out. The zooming center is the location of the mouse pointer.



Graphically Testing and Running a Unit Program

Interactive programming graphics



Graphically Testing and Running a Unit Program

The interactive programming graphics are only available for creation of a contour program (.HC file).

The TNC can generate a two-dimensional graphic of the contour while you are programming it:



- Generate a complete graphic
 - Generate interactive graphic blockwise
- ▶ Start and complete the graphic
- Automatic graphic generation during programming
- Erase the graphic
- Update the graphic
- Display or hide block numbers

smarT.NC: Defining	contour	2	a	nd editing
TNC:\SaarTNC\HRKEN.HC 18 Line FLT 19 Line FL 19 Line FL 20 Rounding arc RHD 21 Line FL 22 Circle FC 22 Circle FC 23 Circle FCT	tle center Co tle center Co tle radius	ction # COR	+0	
	CT FPOL	-0 51981	START	RESET

Test Graphics and Execution Graphics

ᇞ

Select the GRAPHICS or PROGRAM+GRAPHICS layout.

The TNC can graphically display a machining operation in the Execute and Test submodes of operation. The following functions are available via soft key:



Plan view



Projection in 3 planes



- - ▶ High-resolution 3-D view
 - Run a program test up to a certain block



STOP AT

▶ Test the entire program



START

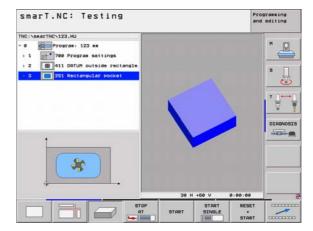
- ▶ Test the program unit-by-unit
- Reset the blank form and test the entire program
- Functions for section magnification



▶ Functions for the sectional planes



Functions for rotating and magnifying/reducing



Unit Program Testing and Graphically Running a



- Selecting the stopwatch functions
- Setting the simulation speed
- ▶ Function for measuring the machining time
- Consider or ignore program blocks preceded by a slash



i

Status displays

Select the PROGRAM+STATUS layout.

In the program run modes a window in the lower part of the screen shows information on

- Tool position
- Feed rate

ᇞ

Active miscellaneous functions

By pressing the soft keys or clicking the tabs you can have further status information be displayed in a screen window:

- STATUS UBERSICHT STATUS
- Activate **0verview** tab: Display of the most important information
- Activate POS tab: Display of positions
- POS.
- STATUS TOOL STATUS

COORD.

TRANSF .

- Activate TOOL tab: Display of tool data
- Activate TRANS tab: Display of active coordinate transformations
 Shift tabs to the left
- Shift tabs to the right

TNC:\searTN	C\123.HU			Overview	PG	H 1	BL	CYC	н	POS	0	
→ 1 🛃	Program: 123 700 Program 1411 DATUM o 251 Rectang	settings utside recta	ingle	V +0. Z +0 T : 5	0000 0000 .000 +0.	ee22	R DR-				10	s
	8% S-1	ST 22:46			LBL LBL		*		REP 00:0	00:00		DIAGNO
X	+0.000		+35	5.349				-3	COLUMN TWO	-		ĺ
+a +C	+0.00		_		-	+B			+0	. 01	00	-
NOML .		T 5		Z 5 100	1	S 1		.0		5 /	9	
STATUS OF	STATUS POS.	STATUS TOOL	STA COO TRAF	RD.						-]	

i

Running a unit program

Unit Program

Graphically Testing and Running a Unit Program

You can run UNIT programs (*.HU) in the smarT.NC operating mode, or in the usual Program Run, Single Block or Program Run, Full Sequence operating modes.

When the smarT.NC Program Run operating mode is selected, the TNC automatically deactivates all global program-run settings that you have activated in the conventional Single Block and Full Sequence program run modes. Refer to the Conversational User's Manual for more information about this.

You can run a unit program in the Execute submode in the following ways:

- Run the unit program unit by unit
- Run the entire unit program
- Run individual, active units



Please note the instructions on running programs in the machine manual and the user's manual.

smarT.NC: Progr	am ru	In					camming editing
TNC:\searTNC\123.HU		Overview	PGH	LBL CY		5 (0)	
- 0 Program: 123 mm + 1 Program setting		X +0.0 Y +0.0 Z +0.	000	•4	+0.000 DIST.		" 🖉
> 2 # 411 DATUM outside		TIS	+0.000	TAPH	+5.00		s 🗍
3 251 Rectangular po	cket	DL-TAB DL-PGM		DR-TAB DR-PGM			
				7# @ @			• ∂
8% S-IST 22:	20.00		8L	1	REP	0	DIAGNOS
		55.349	0 Z	-	306.8	29	
+a +0.000			+B	-	+0.0	00	
+C +0.000					1000 0000	-	
NOHL. @:MANCO> T :	LON .	Z 5 100	S1		000 H 5 /	. 9	
RUN RUN RUN SINGLE ALL ACTI UNITS UNITS UNIT	VE SC		BERT	DATUH	TOD TABL		~

Procedure



Select the smarT.NC operating mode



SINGLE

RUN ALL UNITS

RUN ACTIVE UNIT

- Select the Execute submode
- Press the RUN SINGLE UNIT soft key, or
 - Press the RUN ALL UNITS soft key, or
 - ▶ Press the RUN ACTIVE UNIT soft key



Mid-program startup (block scan, FCL 2 function)

With the mid-program startup function (block scan) you can run a part program from any desired line number. The TNC scans the program blocks up to that line number and displays the contour (select the PROGRAM + GRAPHICS screen layout).

If the start-up point lies on a machining step in which you have defined two or more machining positions, you can select the desired start-up point by entering a point index. The point index contains the position of the point in the input form.

You can select the point index very conveniently if you have defined the machining position in a point table. Then smarT.NC automatically shows the defined machining pattern in a preview window in which you can select a start-up point by soft key.

smarT.NC: Program run	Programming and editing
NC:VSeaTNCV122.HU Program: 122 m Defining the startup point	
Point index Point sole Point table Point tabl	**************************************
+C +0.000 S1 0.00 NOML. ⊕:MARVED I 5 Z 5 100 F 0	0
	END

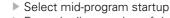
Mid-program startup in a point table (FCL 2 function)



Select the smarT.NC operating mode



Select the Execute submode



Enter the line number of the machining unit in which you want to start the program run. Confirm with the ENT key. Then smarT.NC displays the content of the point table in the preview window.



- Select the machining position where you want to resume machining
- Press the NC Start key: smarT.NC calculates all factors required for program entry



- Select the function for approaching the starting position: In a pop-up window, smarT.NC displays the machine status required at the startup position
- Press the NC Start key: smarT.NC reestablishes the machine status (e.g. insert the required tool)
- Press the NC Start key again: smarT.NC moves to the starting position in the sequence shown in the pop-up window. As an alternative, you can move separately in each axis to the starting position
- ▶ Press the NC Start key. smarT.NC resumes program run.



In addition, the following functions are available in the pop-up window:

- Show/hide preview window
- Show/hide the program interruption point last saved
- Load the program interruption point last saved



i

HEIDENHAIN

DR. JOHANNES HEIDENHAIN GmbH

Dr.-Johannes-Heidenhain-Straße 5

83301 Traunreut, Germany

2 +49 (8669) 31-0
FAX +49 (8669) 5061
E-Mail: info@heidenhain.de

HEIDENHAIN (G.B.) Limited

200 London Road, Burgess Hill West Sussex RH15 9RD, United Kingdom (01444) 247711 (01444) 870024

www.heidenhain.de

