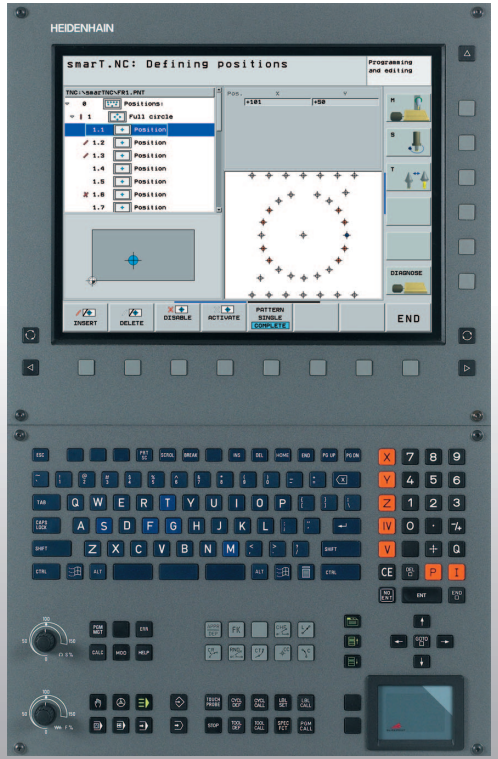
HEIDENHAIN

Pilot
smarT.NC

iTNC 530

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



English (en)
10/2005

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

Warning: danger for the user or machine!

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

Control	NC software number
iTNC 530	340 490-02
iTNC 530, export version	340 491-02
iTNC 530 with Windows 2000	340 492-02
iTNC 530 with Windows 2000, export version	340 493-02
iTNC 530 programming station	340 494-02

Contents

The smarT.NC Pilot	3
Fundamentals	5
Defining Machining Operations	26
Defining Machining Positions	111
Defining Contours	125
Generating Contour Programs from DXF Data	133
Graphically Testing and Running a Unit Program	144

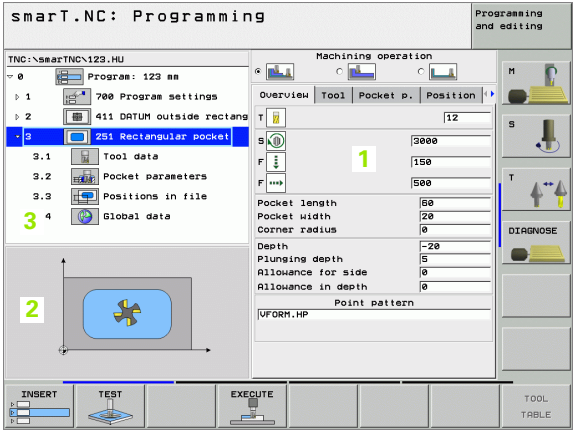
Fundamentals

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 (see area 1 in the figure at upper right) make it easier to define the required machining parameters, which are also displayed in a help graphic (2). The structured program layout in a tree diagram (see area 3) 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.



Features available with smarT.NC

In the software version available at present, not all TNC functions definable in conversational dialog are also definable with the forms in smarT.NC. We are developing the software to include as many functions as quickly as possible in smarT.NC.

To ensure flexibility, smarT.NC features a conversational unit in which almost any conversational function can be inserted between the working units defined in smarT.NC. Blocks inserted in this way will be shown exactly as they are in the conversational editor.

Available functions:

- Programming and execution of drilling cycles (201, 202, 204, 205, 240)
- Programming and execution of tapping cycles (Cycles 206 and 209)
- Programming and execution of thread milling cycles (Cycles 26x)
- Programming and execution of pocket milling cycles (Cycles 25x, Cycle 209)
- Programming and execution of simple operations (Cycle 232)
- Programming and execution of contouring cycles (Cycles 20, 22, 23, 24, 25)
- Programming and execution of touch probe cycles (all Cycles 4xx except 440 and 441)
- Programming and execution of coordinate transformations (datum shift, mirroring, rotation, scaling, tilting the working plane with the PLANE function; FCL 2 functions)
- Graphically supported definition of machining positions (pattern generator)
- Graphically supported hiding and locking of machining positions
- Graphically supported and help-graphic supported definition of contours for use in the contour cycles
- Extracting Contour from DXF files (software option)

- Contour pocket with of easy linking of pocket and island contours (EasyMode contour formula) and definition of separate depths for every subcontour (FCL 2 function)
- Selection of contour programs (.HC files) and machining positions (.HP files) from the form in a file selection dialog box
- Standard file management in the smarT.NC directory
- Graphic simulation of machining (test run)
- Graphically aided mid-program startup with possibility of selecting the entry point if it is within a point file (smarT.NC Program Run, FCL 2 function).
- Mouse support (also on the single-processor version)

Programs/Files

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

PROG20	.HU
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.

smarT.NC automatically saves all files in the **TNC:\smarTNC** directory only.

If you want to call an existing program in ISO or conversational dialog format, the program must be saved in the **TNC:\smarTNC** directory. If necessary, copy the program to this location.

Files in the TNC	Type
Programs	
In HEIDENHAIN format	.H
In ISO format	.I
smarT.NC files	
Structured unit program	.HU
Contour descriptions	.HC
Point tables for machining positions	.HP
Tables for	
Tools	.T
Tool changers	.TCH
Pallets	.P
Datums	.D
Presets (reference points)	.PR
Cutting data	.CDT
Cutting materials, workpiece materials	.TAB
Texts as	
ASCII files	.A
Drawing data as	
DXF files	.DXF

Selecting the new operating mode the first time

- 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 the file name without the file type, and 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

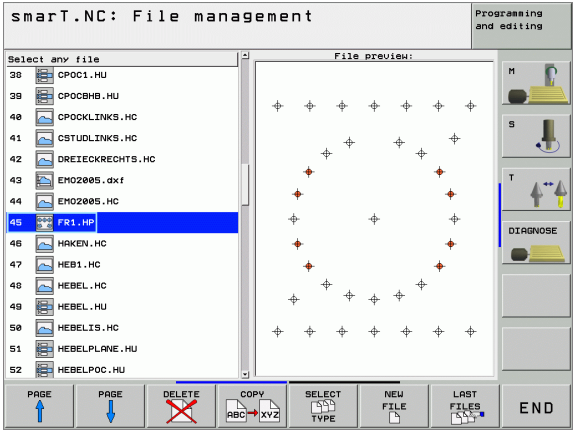
As mentioned previously, smarT.NC differentiates between three file types: unit programs (.HU), contour descriptions (.HC) and point tables (.HP). These three file types can be selected and edited in the file manager in the smarT.NC operating mode. Contour descriptions and point tables can also be edited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

You can also open DXF files from within smarT.NC in order to extract contour description (.HC files) from them (software option).

In the current software version, smarT.NC automatically saves all files in the **TNC:\smarTNC** directory only.

Select a file type

- Press the PGM MGT key to select the file manager.
- Press the SELECT TYPE soft key.
- To display all file types, press the SHOW ALL soft key, or
- To display only unit programs, press the SHOW .HU soft key, or
- To display only contour descriptions, press the SHOW .HC soft key, or
- To display only point tables, press the SHOW .HP soft key.
- To show only DXF files, press the SHOW DXF soft key.



If you have selected the file type .HP, then in the right half of the file-manager screen smarT.NC shows a preview of the point table's contents.

In order to enhance the optical distinction between the various file types, smarT.NC displays an icon in front of each file name. These symbols also appear in the tree views of each file type, as well as in the pop-up windows for file selection.

File type	Icon
Unit program	
Contour program	
Point table for machining positions	
DXF file	

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
- Press the NEW FILE soft key. smarT.NC opens a pop-up window.
- Enter the file name without the file type, and 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 screen button.

Copying 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 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 screen 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 screen button.

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
- 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 screen button. In order to cancel the delete procedure, press the ESC key or the No screen 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
- Press the RENAME soft key (second soft-key row). smarT.NC opens a pop-up window.
- Enter the new file name and confirm your entry with the ENT key or OK screen button. In order to cancel the procedure, press the ESC key or the Cancel screen button.

Selecting one of the last 15 files selected

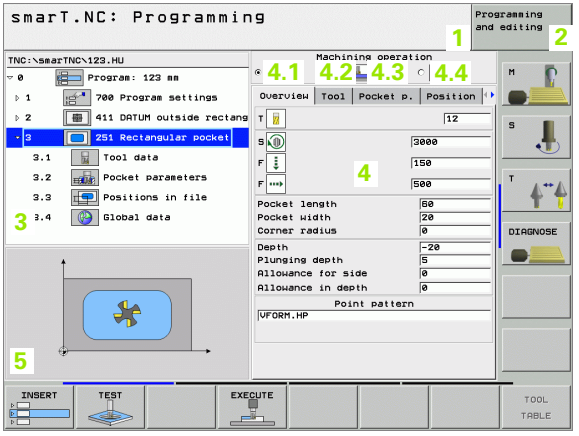
- Press the PGM MGT key to select the file manager.
- Press the LAST FILES soft key. smarT.NC displays the last 15 files that you selected in the smarT.NC operating mode.
- Use the arrow keys to place the highlight on the file you want to select
- Press the ENT key to select the file.

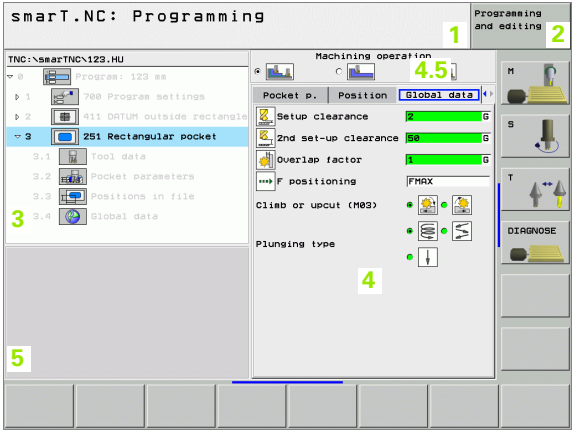
Screen layout during editing

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

Editing unit programs

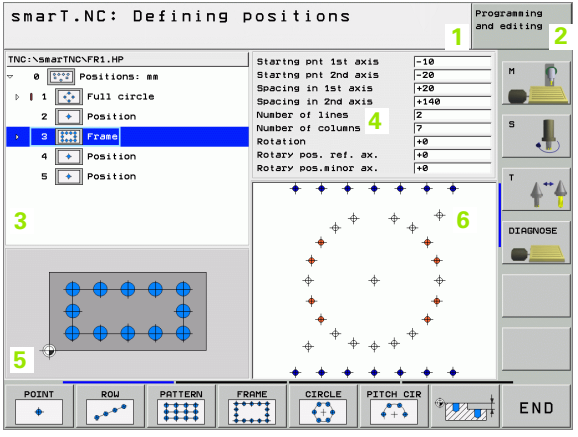
1 Header: Operating mode text, error messages
2 Active background mode of operation
3 Tree view in which the defined working units are shown in a structured format
4 Form window with the various input parameters. Depending on the machining step, there can be up to five forms.
   - 4.1: **Overview form**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





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 show immediately after being saved in the form

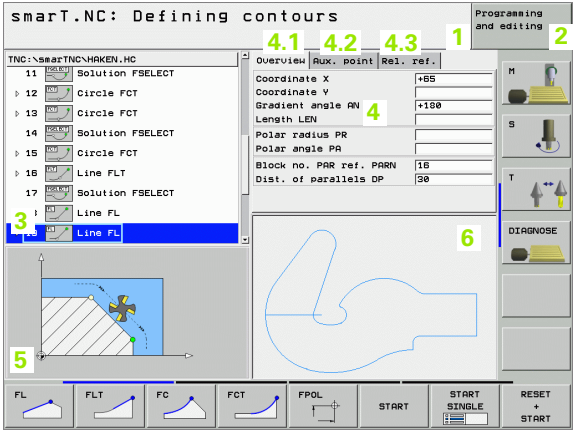


Editing contours

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.

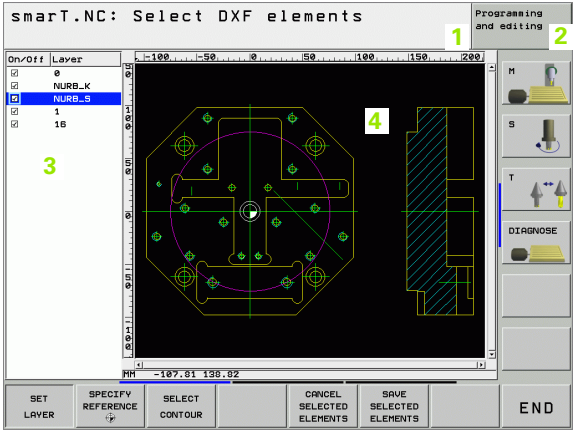
- 4.1: Overview form
  Contains the input possibilities used most often
- 4.2: Detail form 1
  Contains input possibilities for auxiliary points (FL/FLT) and for circle data (FC/FCT)
- 4.3: Detail form 2
  Contains input possibilities for auxiliary points (FL/FLT) and for auxiliary points (FC/FCT)
- 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 show immediately after being saved in the form



Displaying DXF files

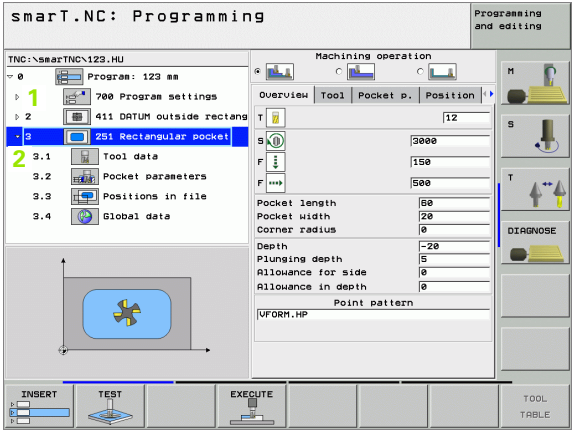
1 Header: Operating mode text, error messages
2 Active background mode of operation
3 Layers or already selected contour elements in the DXF file
4 Drawing window in which smarT.NC shows the DXF file contents



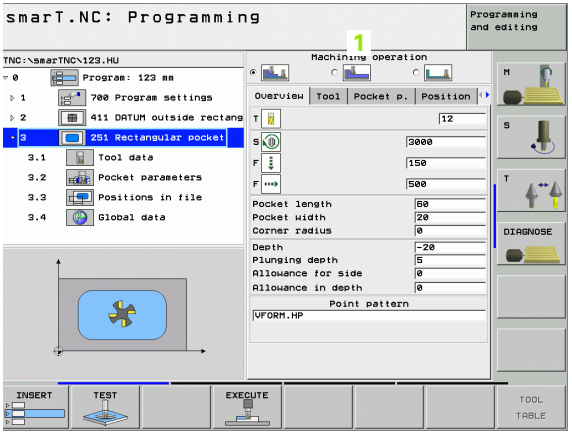
Navigating in smarT.NC

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 pointing to the right** is displayed in front of the tree view symbol (1), 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 (2)	



Function when the form is active (right side of screen)	Key
Select next input field	ENT
Conclude editing of the form: smarT.NC **saves** all changed data	END
Cancel editing of the form: smarT.NC **does not save** the changed data	DEL
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 (1, see figure): Select next/previous option	→ ←
Reset an already entered numerical value to 0	CE
Delete completely the contents of the active input field	NO ENT



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

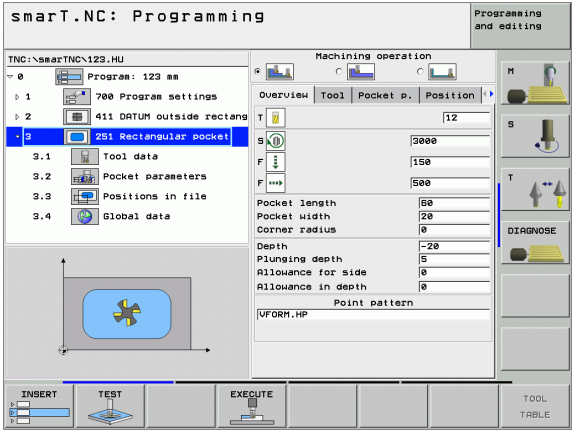
Function when the form is active (right side of screen)	Key
Select next subform	
Select first input parameter in next frame	
Select first input parameter in previous frame	

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 plain-language keys.

Function when the form is active (right side of screen)	Key
Select input field for X axis	X
Select input field for Y axis	Y
Select input field for Z axis	Z
Switching between incremental and absolute input	I
Switching between Cartesian and polar coordinate input	P

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 formula (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 behind the active one

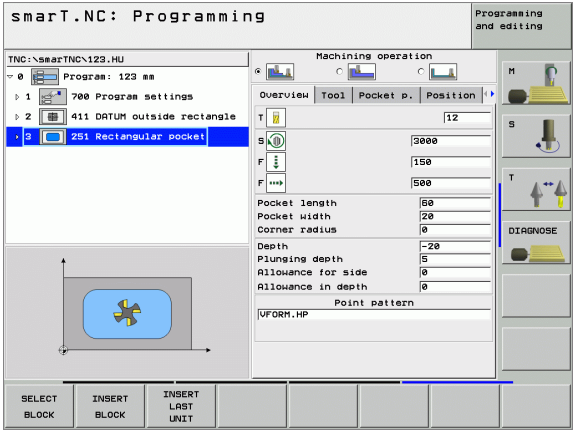
If you want to copy two or more units at the same time, proceed as follows:

- 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

- 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)



Defining Machining Operations

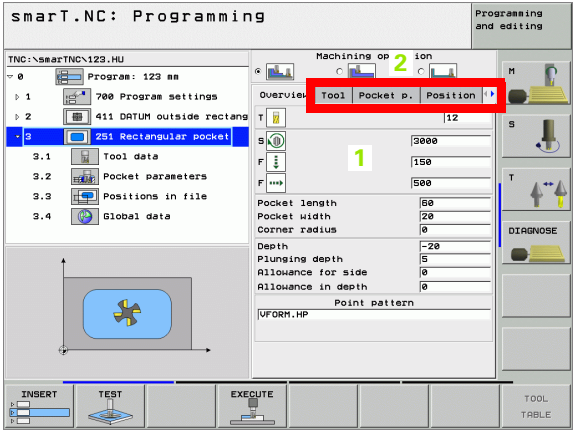
Fundamentals

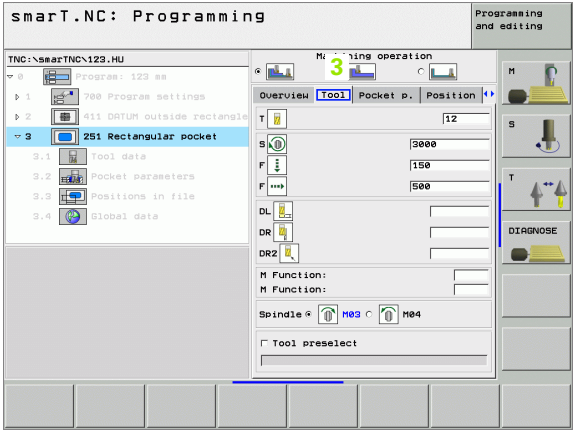
Machining operations are basically defined in smarT.NC as working steps (units), which as a rule consist of several conversational-language programming blocks. smarT.NC automatically creates the conversational blocks in the background in an .HU file (HU: **H**EIDENHAIN **U**nit program), which looks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

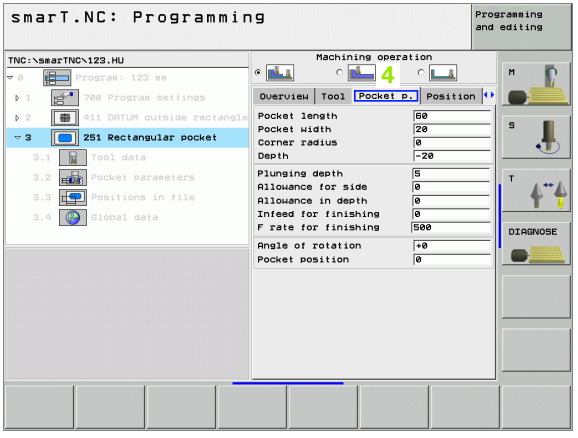
You can define a machining step with just a few entries in the overview form (1, see figure at top right). smarT.NC then performs the basic machining operation. Detail forms (2) are available for entering additional machining data. Values entered in the detail forms are automatically synchronized with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

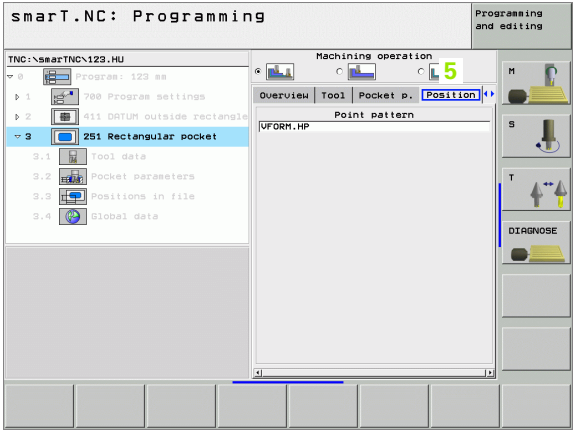
- **Tool detail form** (3)
  On the tool detail form you can enter additional tool-specific data, such as delta values for the length and radius, or M functions.





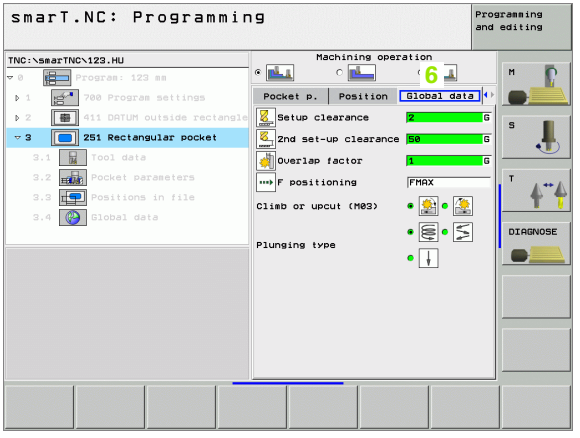
- **Optional parameters detail form** (4)
  On the optional parameters detail form you can define additional machining parameters which are not listed on the overview form, such as decrements for drilling or pocket lengths for milling.
- **Positions detail form** (5)
  On the positions detail form you 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 111.)





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.



Program settings

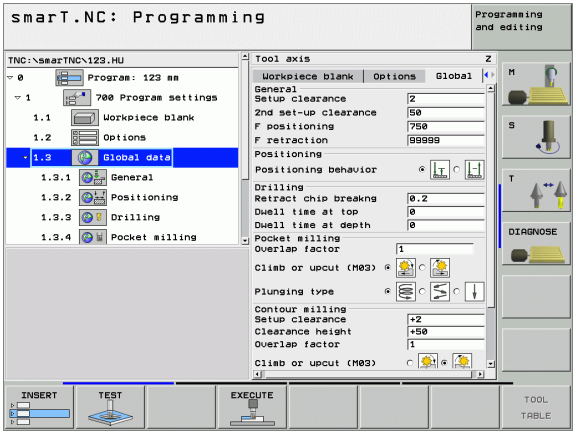
After you have created a new unit program, smarT.NC automatically inserts the **Unit 700 Program Settings.**

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.

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



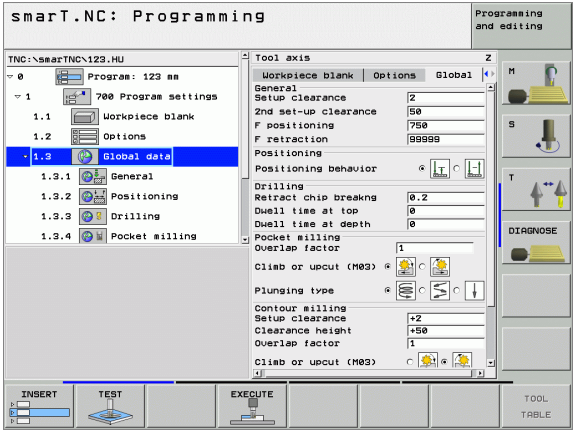
Global data

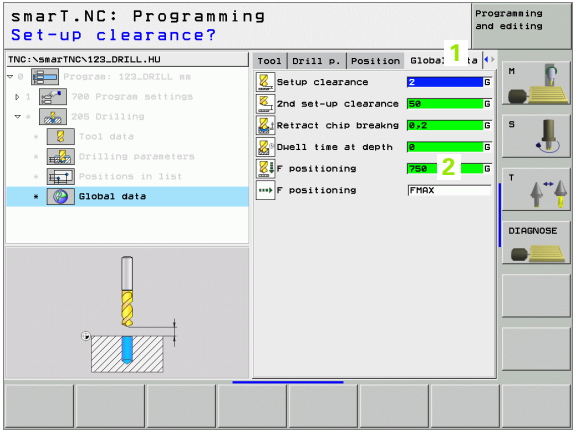
The global data are divided into six groups:

- 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** (1) for the machining step: In the form, smarT.NC shows the parameters valid for this machining step along with the currently active value (2). 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.





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.**

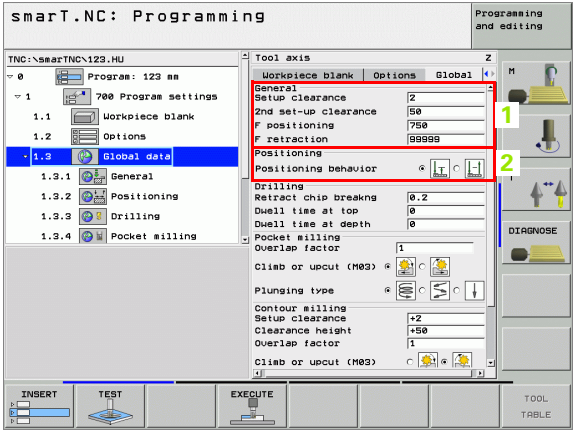
Press the SET STANDARD VALUES soft key to load and therefore activate the value of the global parameter from the program header. The input field of a global parameter whose value from the program header is in effect is displayed with a green background by smarT.NC. On the right side of the input field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

Global data valid everywhere (1)

- **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 (2)

- **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.

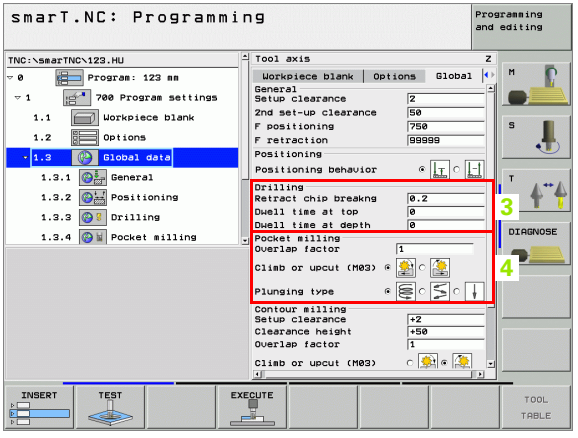


Global data for drilling operations (3)

- **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 set-up clearance.

Global data for milling operations with pocket cycles (4)

- **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.

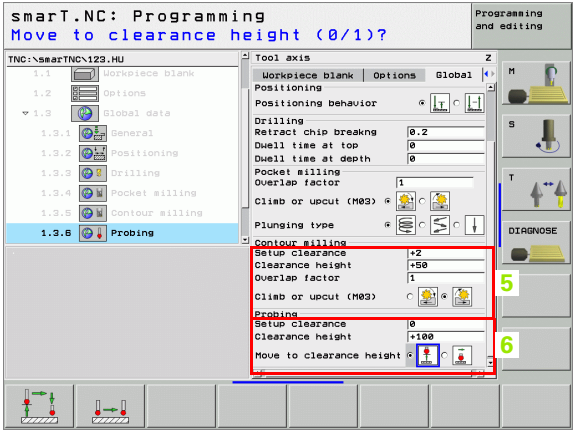


Global data for milling operations with contour cycles (5)

- **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 (6)

- **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.



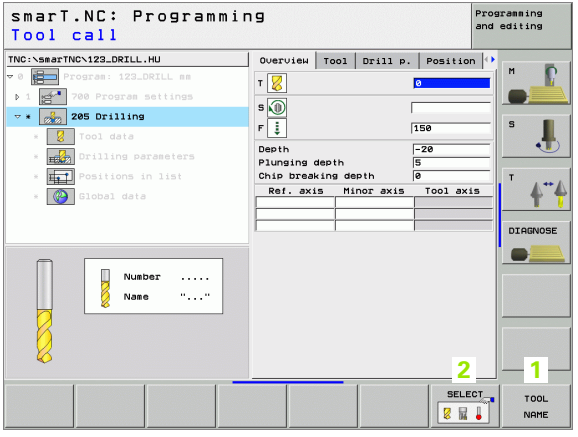
Tool selection

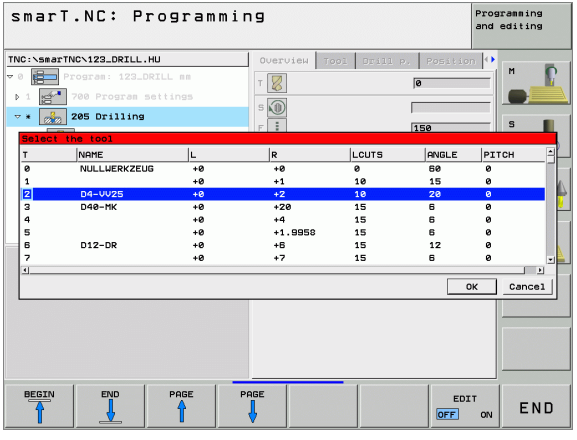
As soon as an input field in the tool selection is active, you can choose via soft key (1) if you want to enter the tool number or tool name.

There is also a soft key (2) 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





RPM/cutting-speed switchover

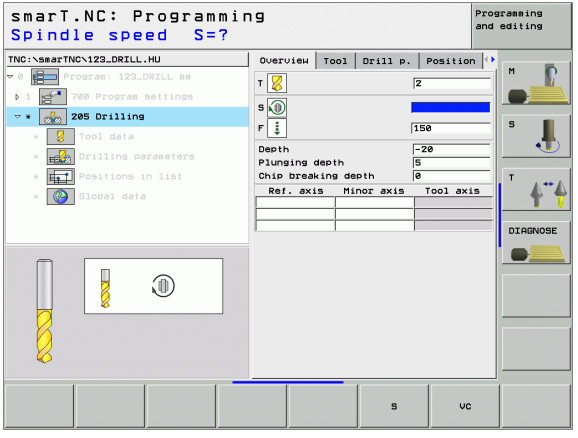
As soon as an input field for defining the spindle speed is active, you can choose whether the speed will be displayed in rpm or as cutting speed (m/min or ipm).

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

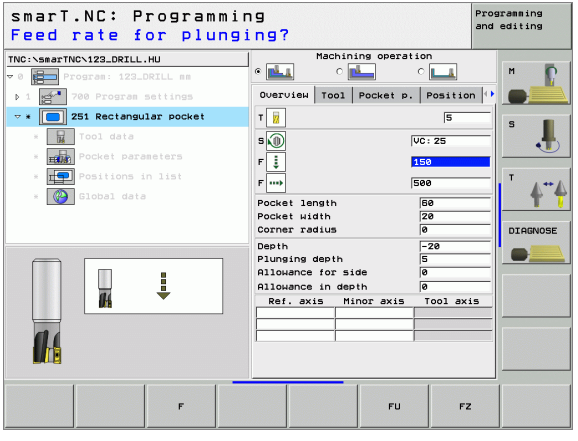


F/FZ/FU/FMAX switchover

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



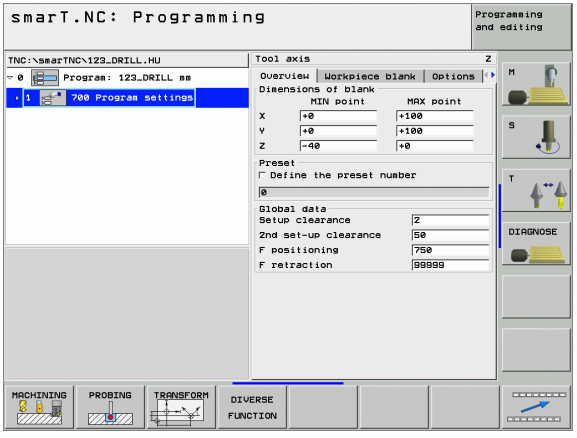
Available working steps (units)

After choosing the smarT.NC 49 operating mode,

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 38
PROBING Probe functions for 3-D touch probes	PROBING	Page 94
CONVERT Functions for coordinate transformation	TRANSFORM	Page 101
DIVERSE FUNCTIONS Program call, conversational unit	DIVERSE FUNCTION	Page 107

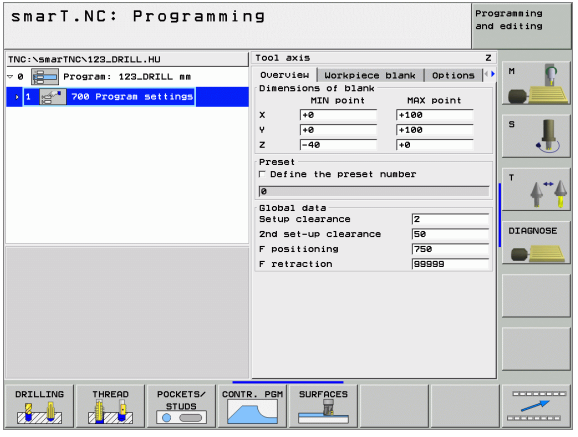
The CONTR. PGM and POSITIONS soft keys on the third soft-key row start the contour programming and pattern generator, respectively.



Machining main group

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

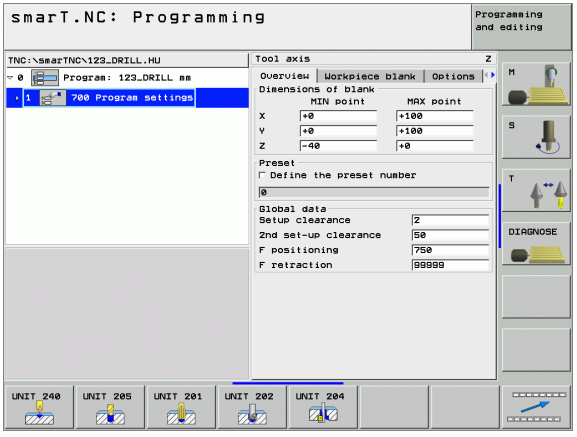
Machining group	Soft key	Page
DRILLING Centering, drilling, reaming, boring, back boring	DRILLING	Page 39
THREAD Tapping with floating tap holder, rigid tapping, thread milling	THREAD	Page 50
POCKETS/STUDS Bore milling, rectangular pockets, circular pockets, slots, circular slots	POCKETS/ STUDS	Page 65
CONTR. PGM Run contour programs: contour train, contour pocket roughing, fine roughing and finishing	CONTR. PGM	Page 77
SURFACES Face milling	SURFACES	Page 90



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 240	Page 40
Unit 205 Drilling	UNIT 205	Page 42
Unit 201 Reaming	UNIT 201	Page 44
Unit 202 Boring	UNIT 202	Page 46
Unit 204 Back Boring	UNIT 204	Page 48



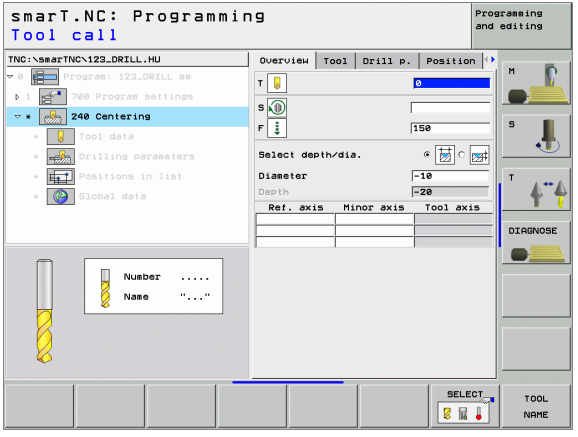
Unit 240 Centering

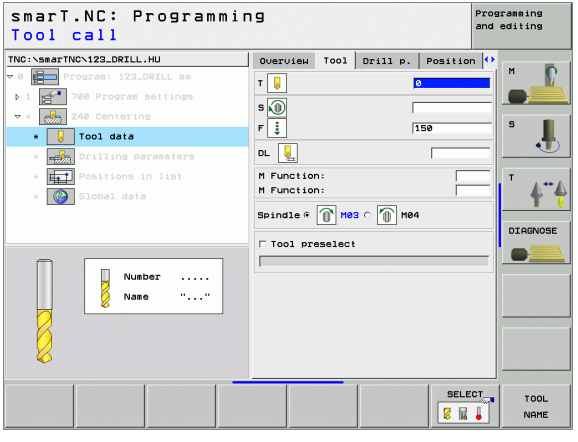
Parameters on the **overview** form:

- **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 111.)

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).

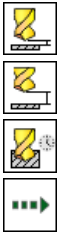




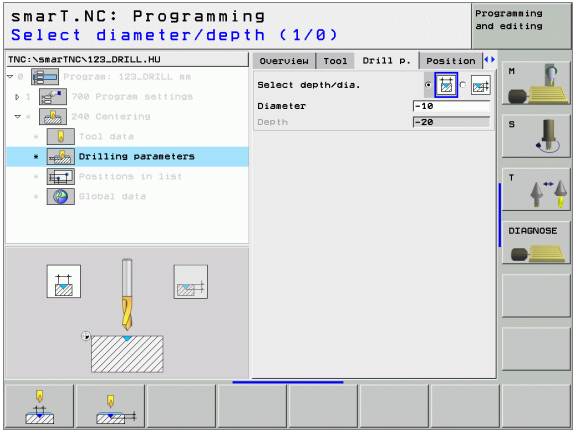
Additional parameters on the **drilling parameters** detail form:

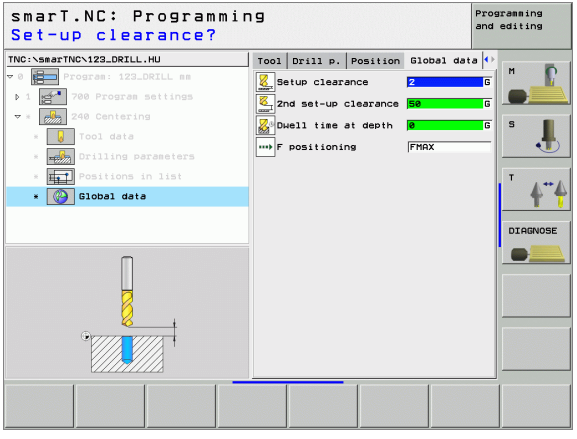
None

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





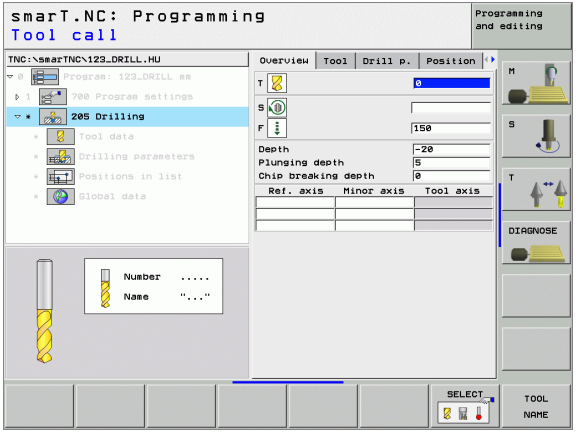
Unit 205 Drilling

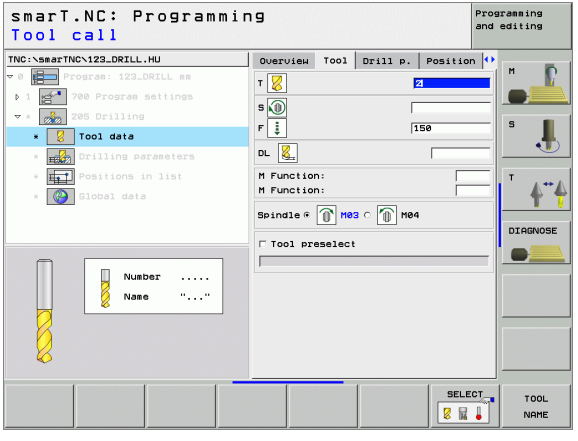
Parameters on the **overview** form:

- **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 111.)

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).



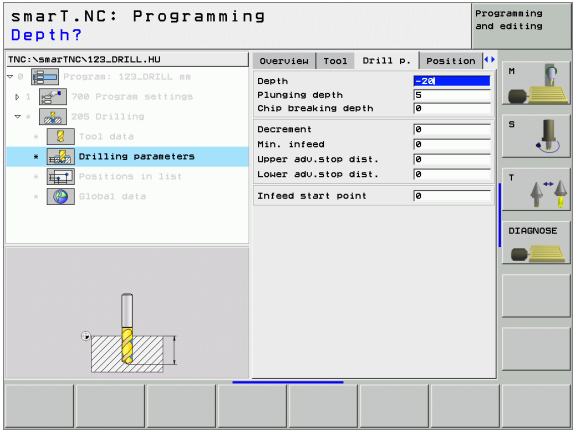


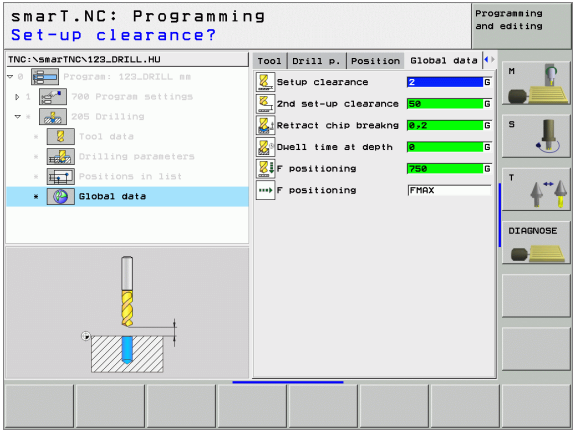
Additional parameters on the **drilling parameters** detail form:

- **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





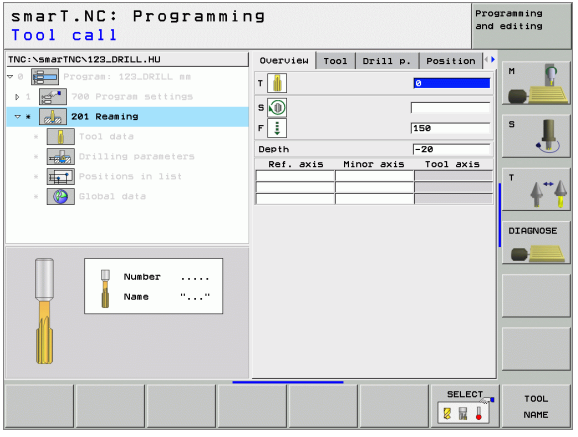
Unit 201 Reaming

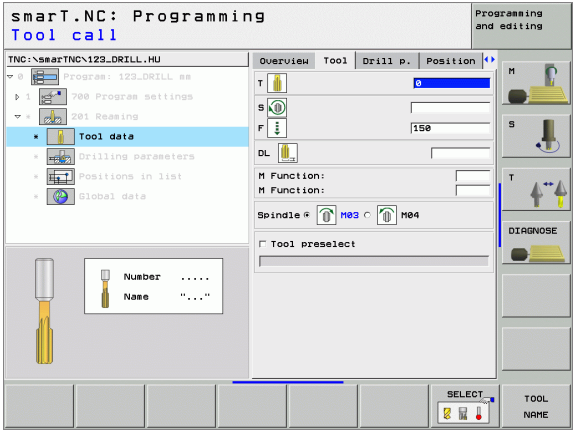
Parameters on the **overview** form:

- **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 111.)

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).



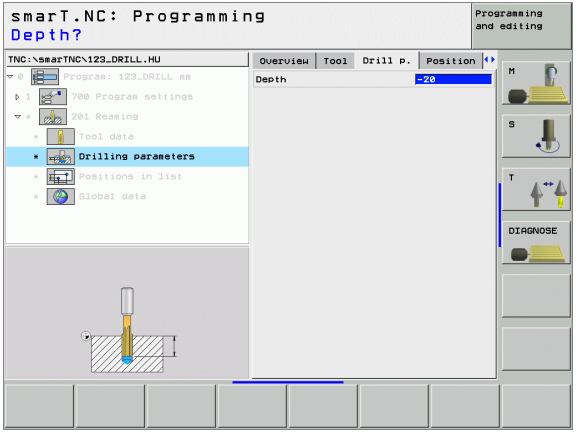


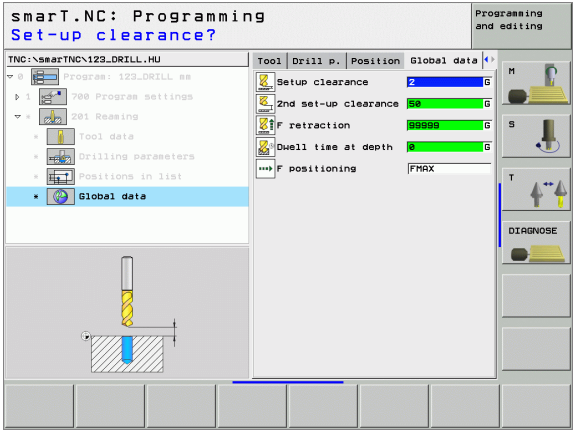
Additional parameters on the **drilling parameters** detail form:

None

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

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





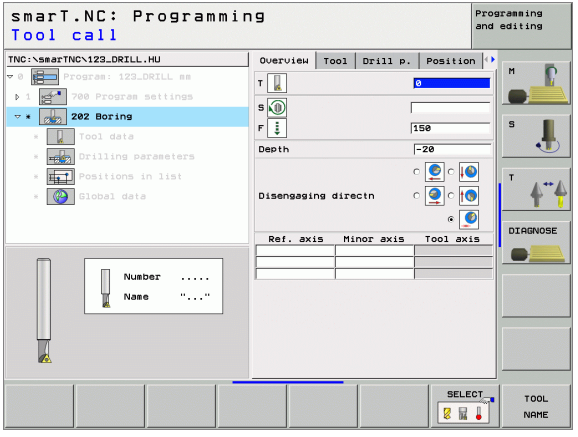
Unit 202 Boring

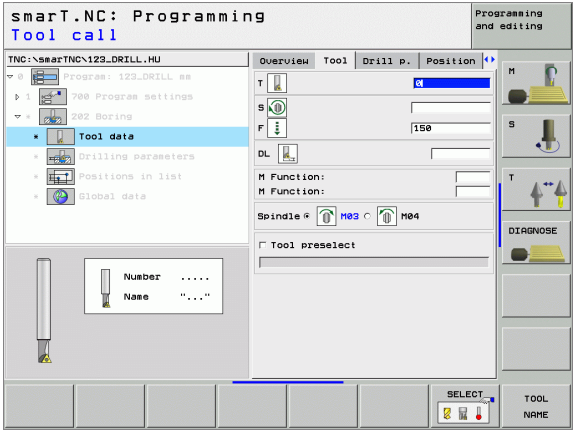
Parameters on the **overview** form:

- **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 111.)

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).



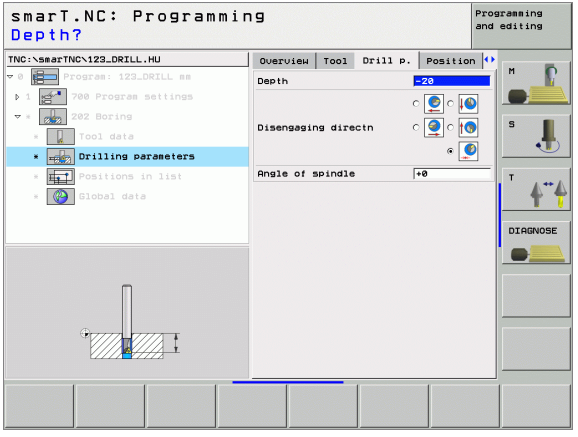


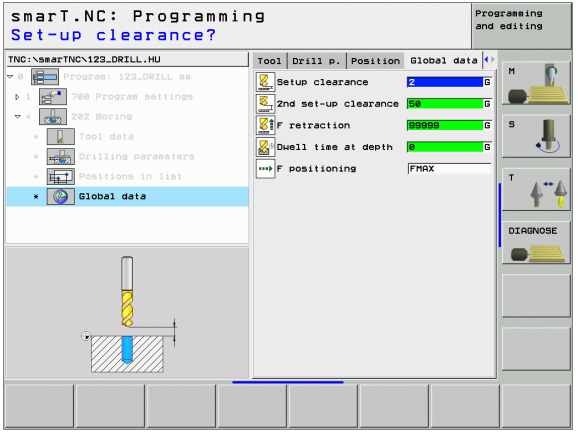
Additional parameters on the **drilling parameters** detail form:

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

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





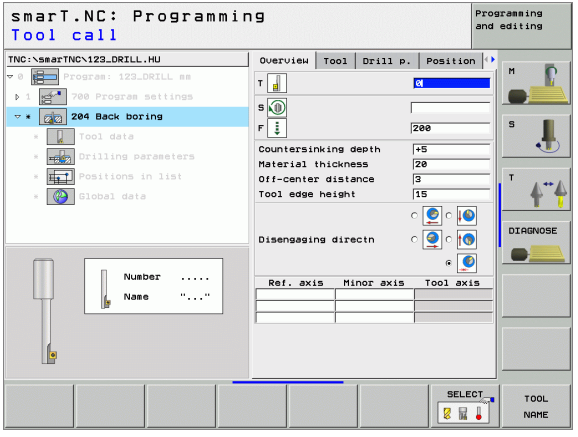
Unit 204 Back Boring

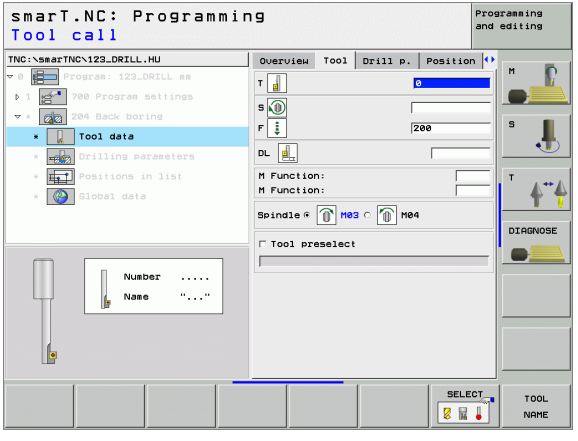
Parameters on the **overview** form:

- **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 111.)

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

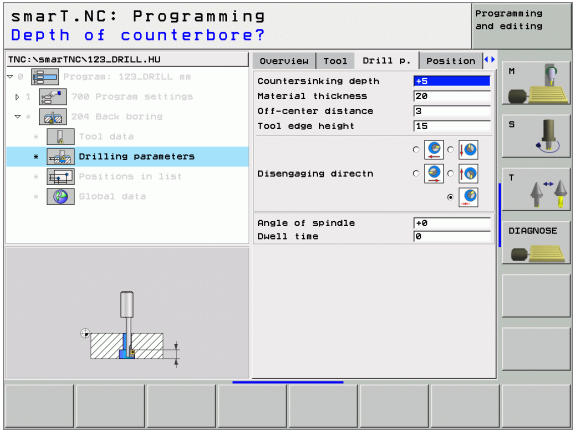
Globally effective parameters on the **global data** detail form:

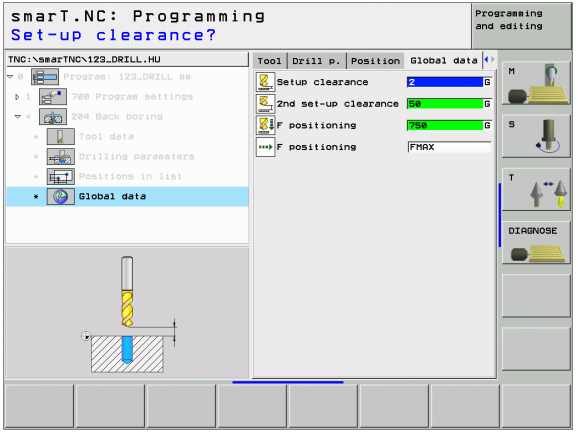
- Set-up clearance

- 2nd set-up clearance

- Feed rate for positioning

- Feed rate for traversing between machining positions

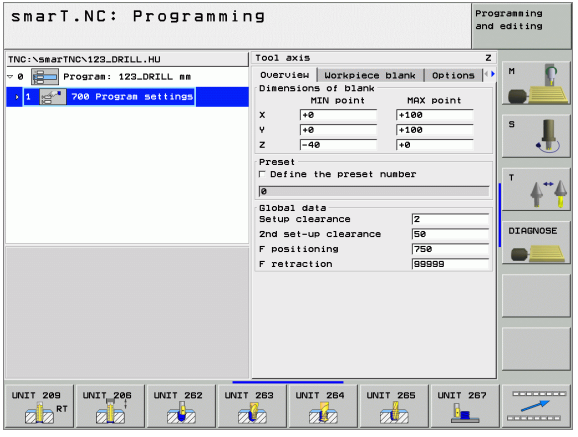




Thread machining group

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 206	Page 51
Unit 209 Rigid Tapping (also with Chip Breaking)	UNIT 209 RT	Page 53
Unit 262 Thread Milling	UNIT 262	Page 55
Unit 263 Thread Milling / Countersinking	UNIT 263	Page 57
Unit 264 Thread Drilling / Milling	UNIT 264	Page 59
Unit 265 Helical Thread Drilling / Milling	UNIT 265	Page 61
Unit 267 Outside Thread Milling	UNIT 267	Page 63



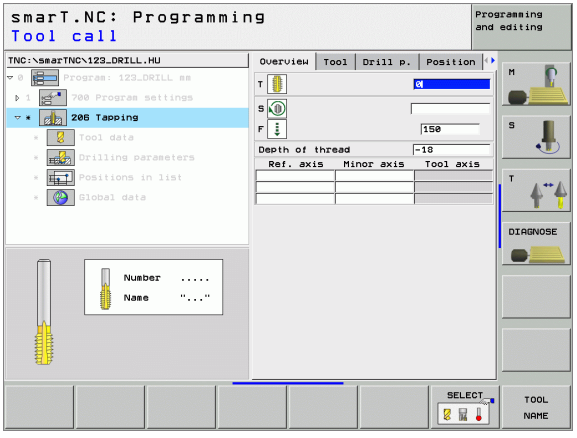
Unit 206 Tapping with a Floating Tap Holder

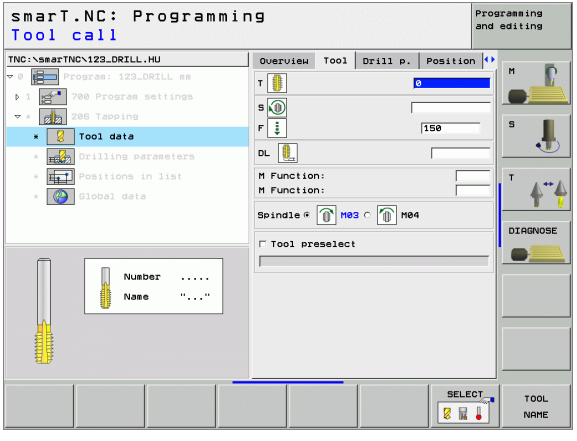
Parameters on the **overview** form:

- **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 111.)

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).

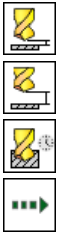




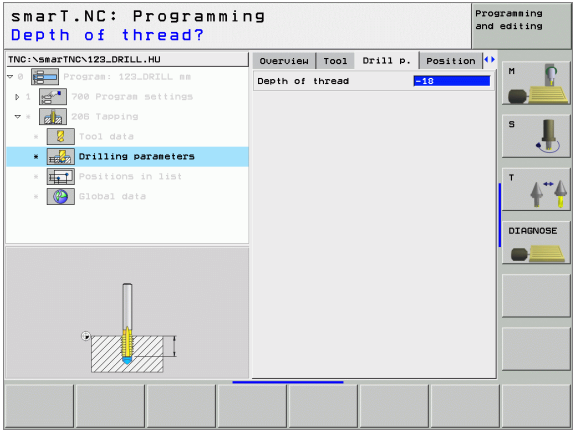
Additional parameters on the **drilling parameters** detail form:

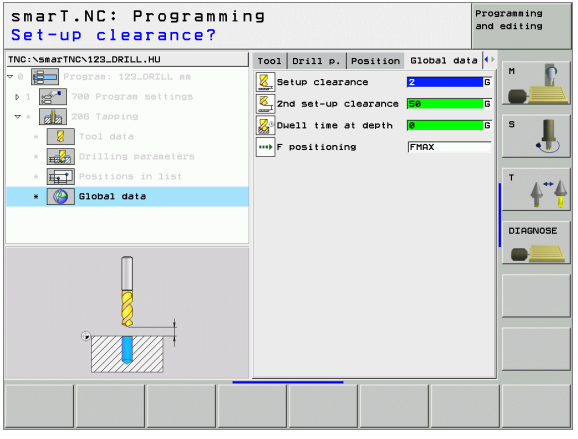
None

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





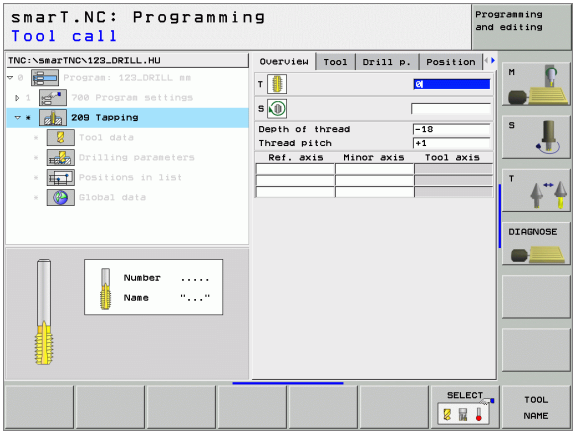
Unit 209 Rigid Tapping

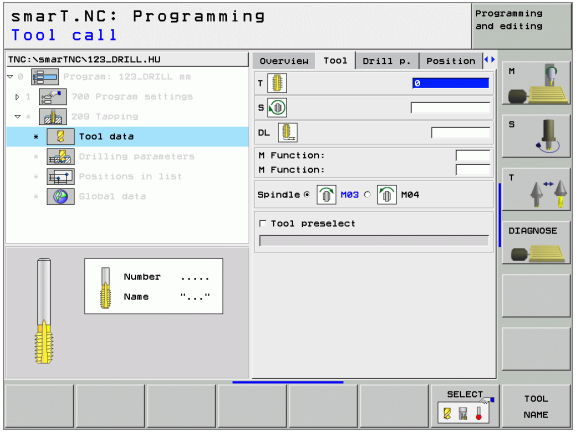
Parameters on the **overview** form:

- **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 111.)

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).

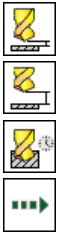




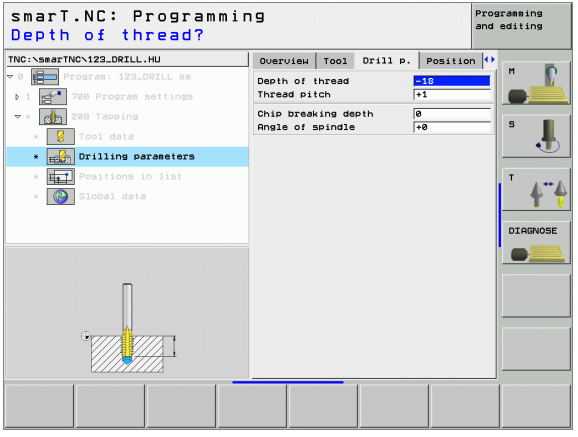
Additional parameters on the **drilling parameters** detail form:

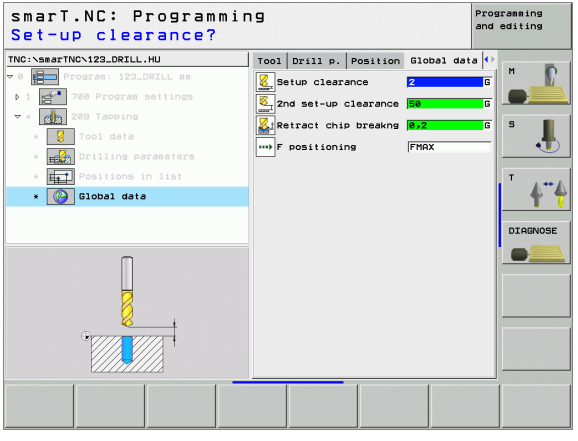
- **Chip breaking depth:** Depth at which chip breaking is to occur.
- **Angle of spindle:** Angle to which smarT.NC positions the tool before thread cutting: This permits regrooving of the thread, if needed.

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



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





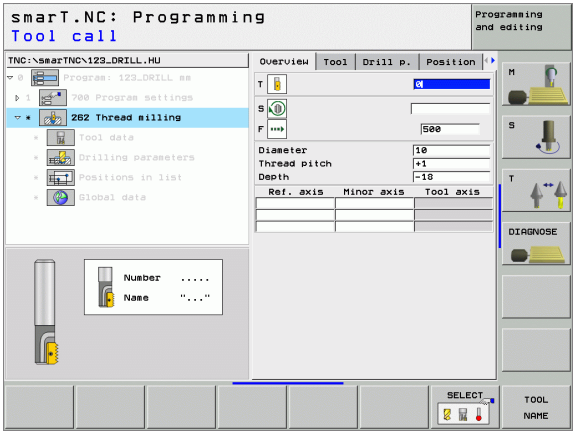
Unit 262 Thread Milling

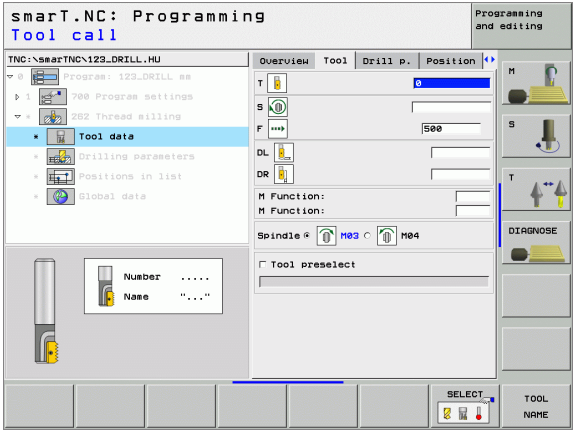
Parameters on the **overview** form:

- **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
- **Diameter:** Nominal diameter of the thread.
- **Thread pitch:** Pitch of the thread.
- **Depth:** Depth of thread.
- Machining positions (see “Defining Machining Positions” on page 111.)

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).



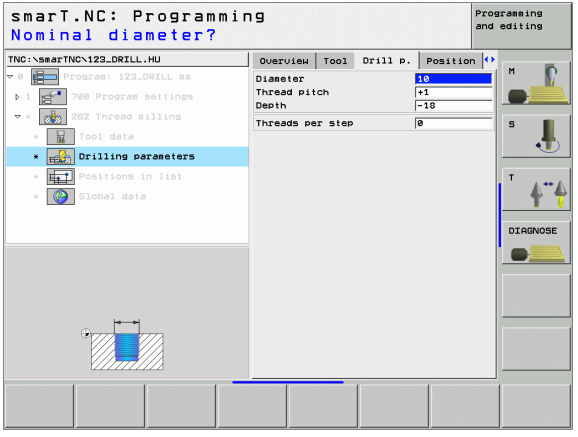


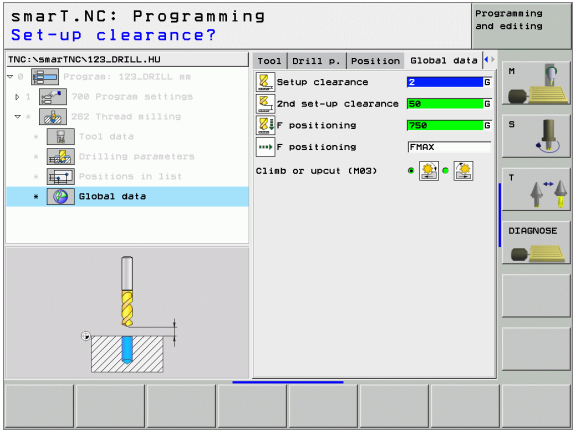
Additional parameters on the drilling parameters detail form:

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





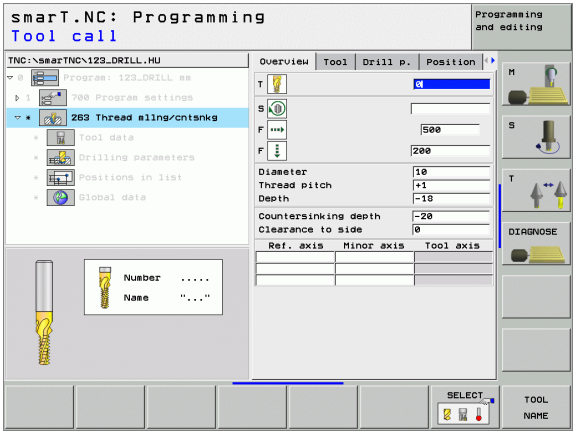
Unit 263 Thread Milling / Countersinking

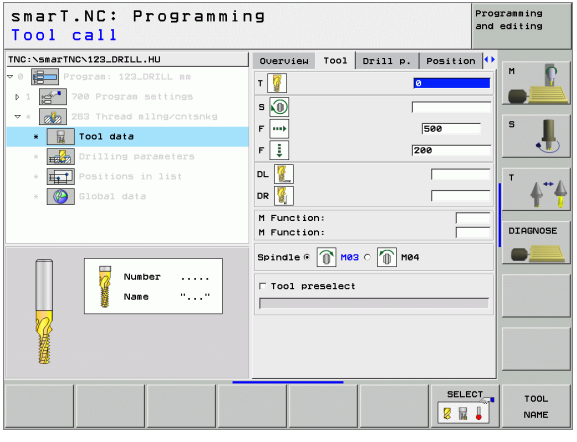
Parameters on the **overview** form:

- **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 111.)

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).





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

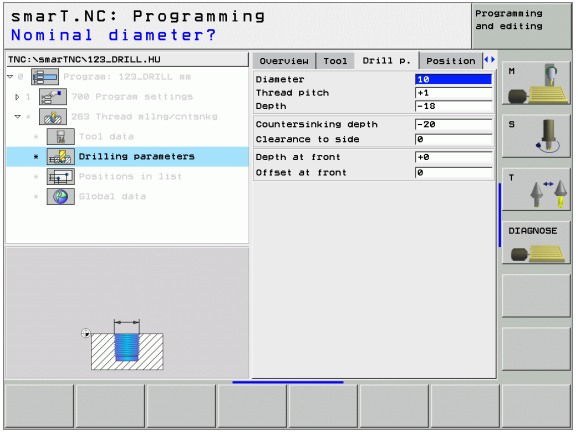
- 2nd set-up clearance

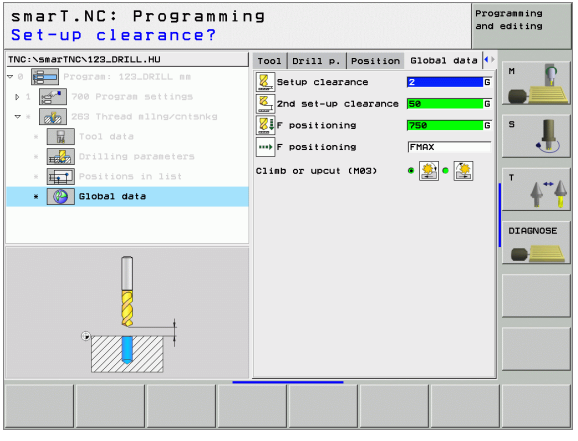
- Positioning feed rate

- Feed rate for traversing between machining positions

- Climb milling, or

- Up-cut milling





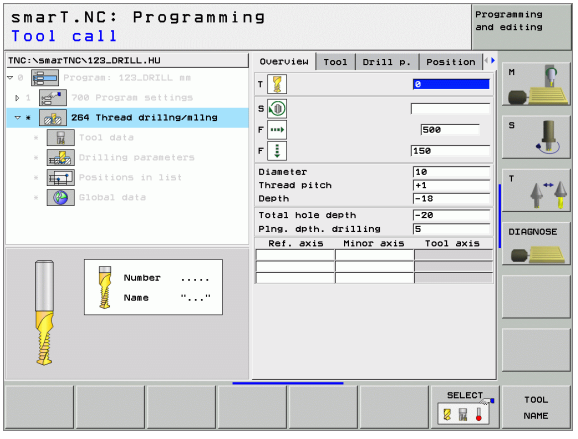
Unit 264 Thread Drilling / Milling

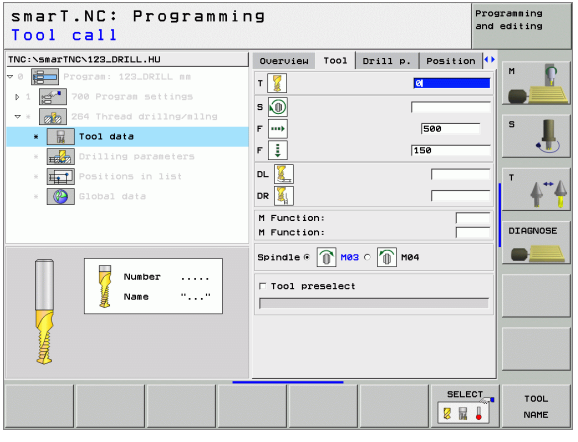
Parameters on the **overview** form:

- **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 111.)

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).



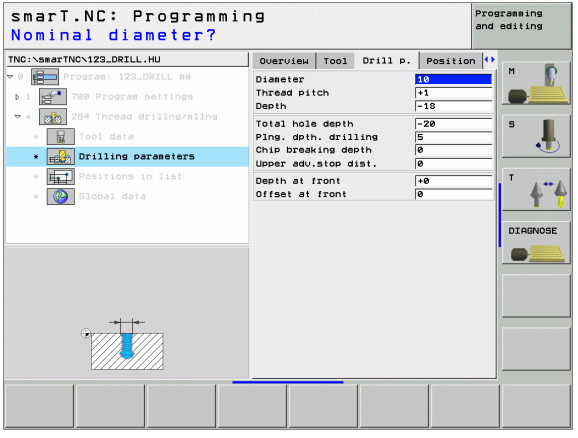


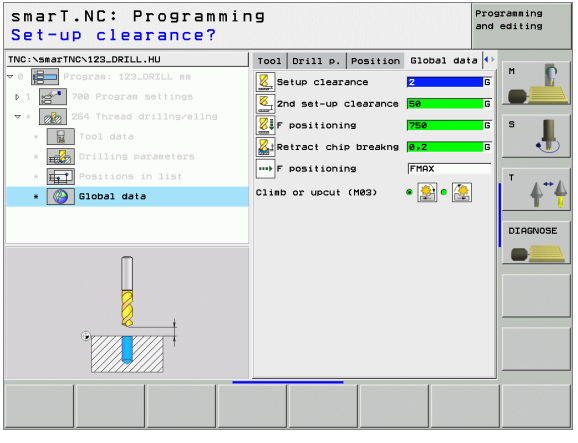
Additional parameters on the **drilling parameters** detail form:

- **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
- 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





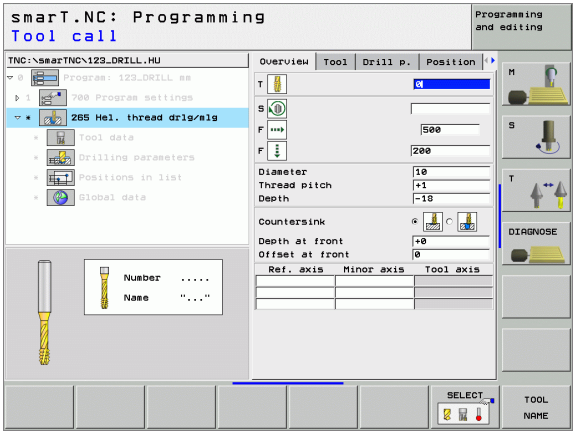
Unit 265 Helical Thread Drilling / Milling

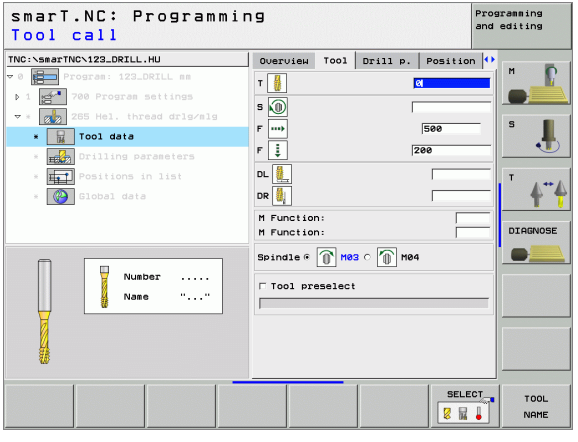
Parameters on the **overview** form:

- **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 111.)

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).

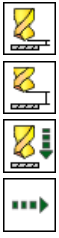




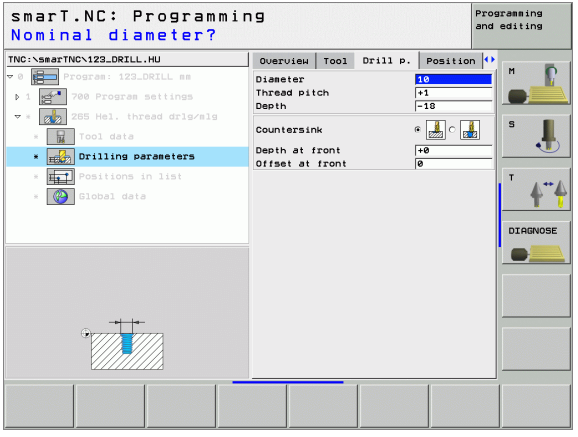
Additional parameters on the **drilling parameters** detail form:

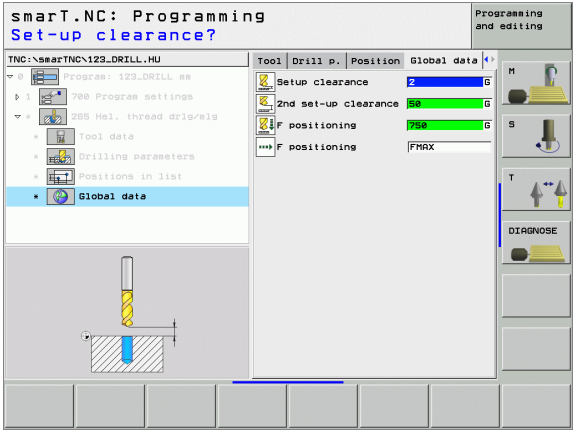
None

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



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





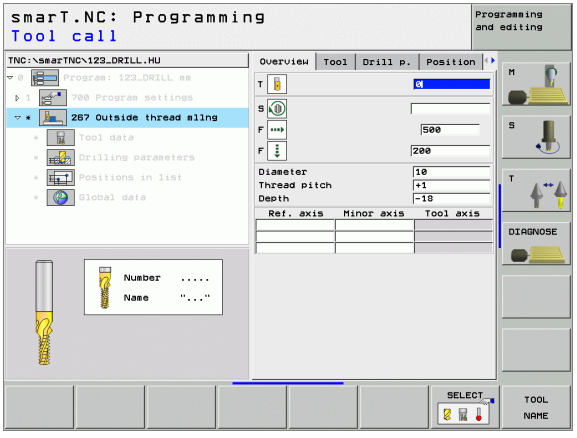
Unit 267 Thread Milling

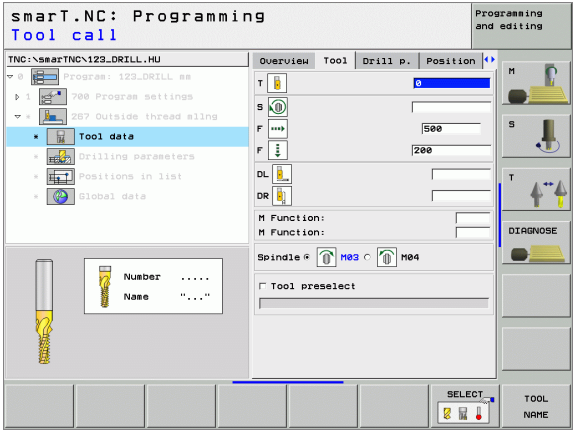
Parameters on the **overview** form:

- **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 111.)

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).





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

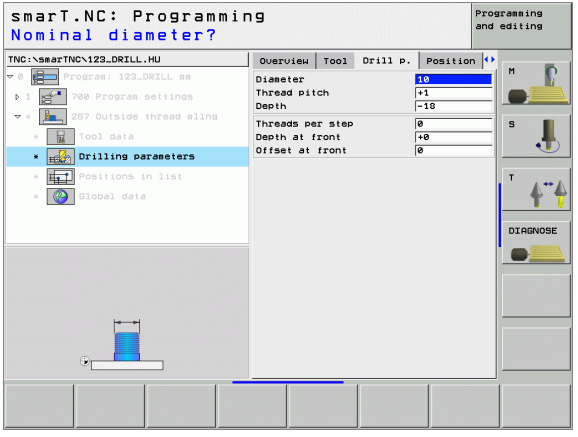
- 2nd set-up clearance

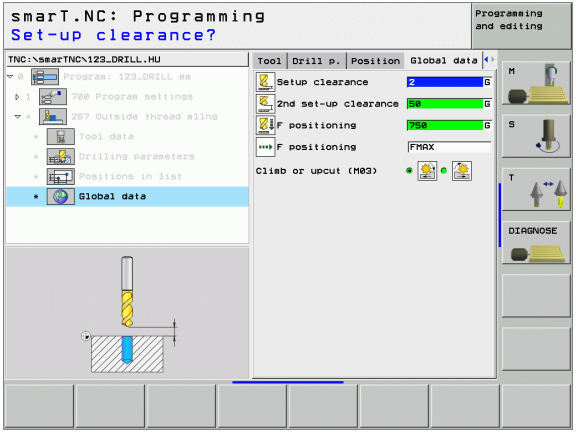
- Positioning feed rate

- Feed rate for traversing between machining positions

- Climb milling, or

- Up-cut milling

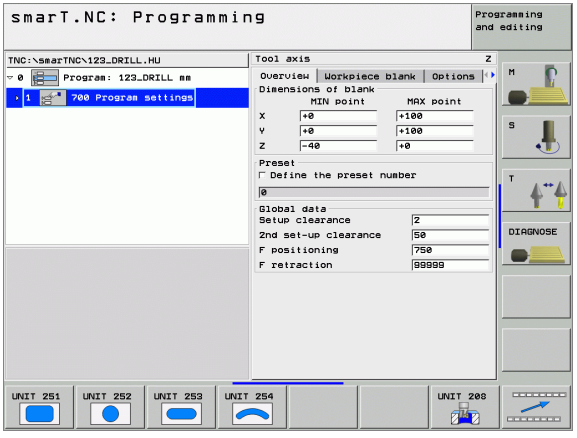




Pockets/Studs machining group

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 66
Unit 252 Circular Pocket	UNIT 252	Page 68
Unit 253 Slot	UNIT 253	Page 70
Unit 254 Circular Slot	UNIT 254	Page 72
Unit 208 Bore Milling	UNIT 208	Page 75



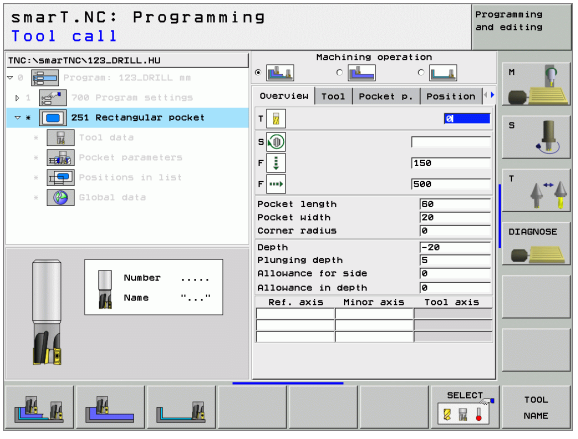
Unit 251 Rectangular Pocket

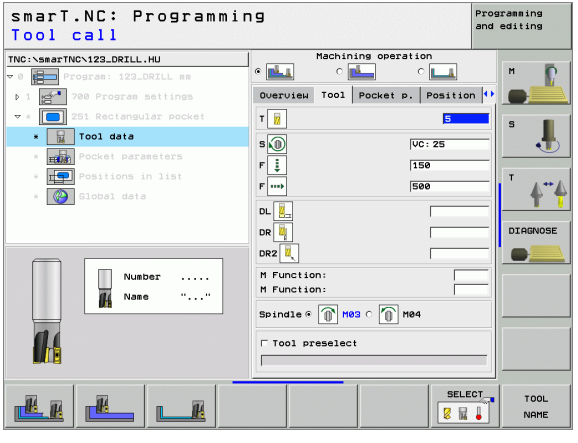
Parameters on the **overview** form:

- **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 111.)

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).



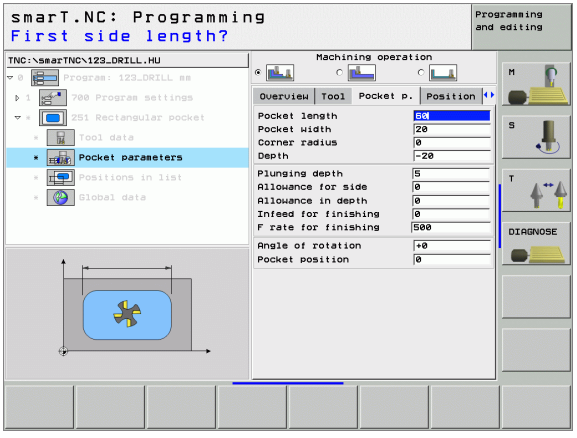


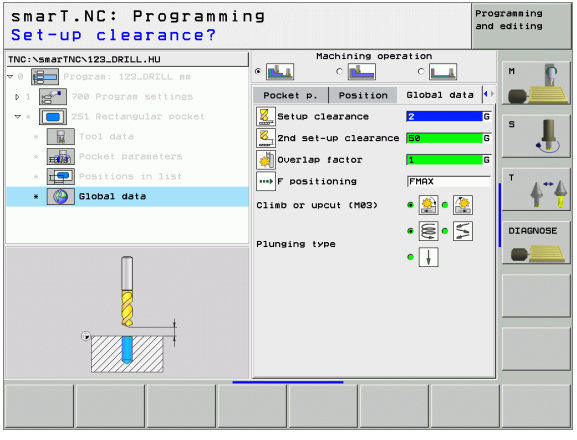
Additional parameters on the **pocket parameters** detail form:

- **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





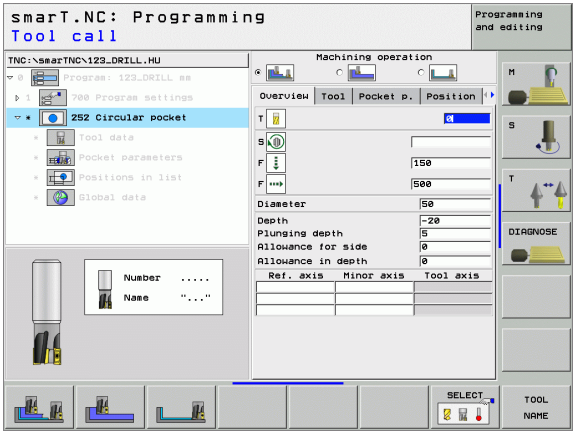
Unit 252 Circular Pocket

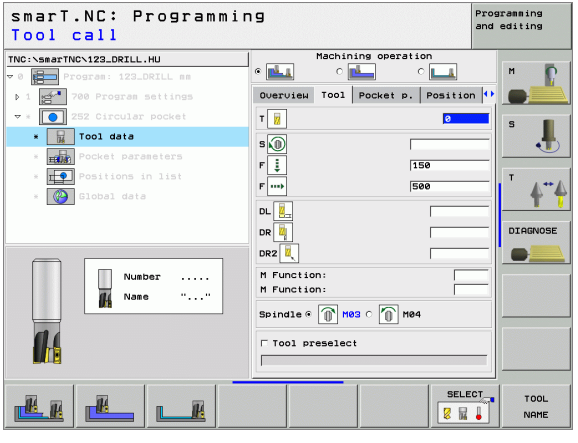
Parameters on the **overview** form:

- **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 111.)

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).



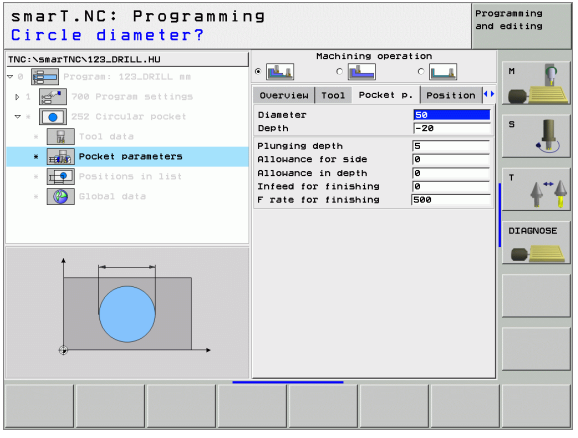


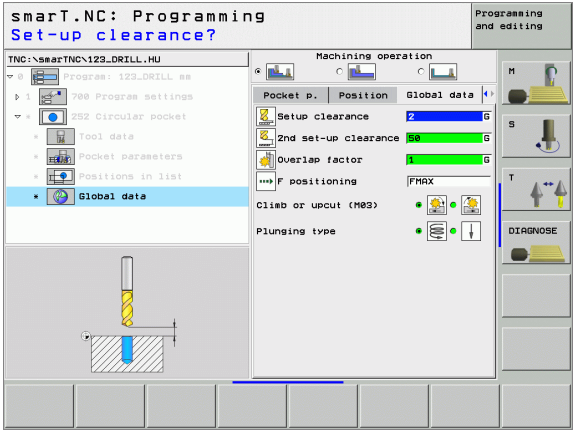
Additional parameters on the **pocket parameters** detail form:

- **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





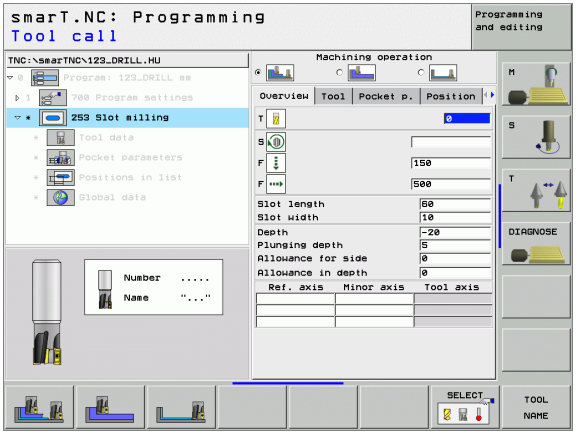
Unit 253 Slot

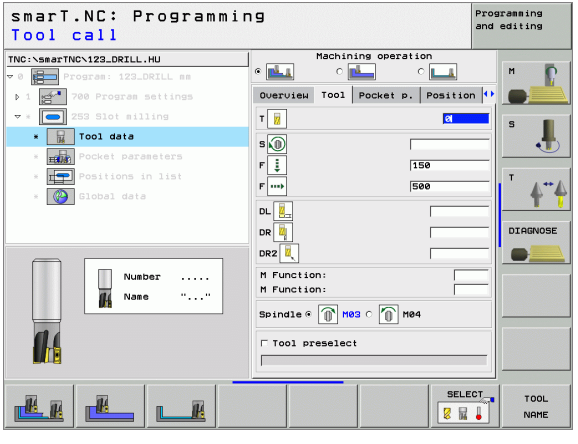
Parameters on the **overview** form:

- **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 111.)

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).



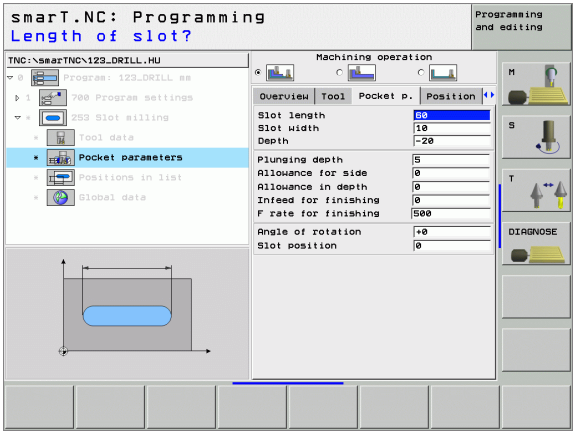


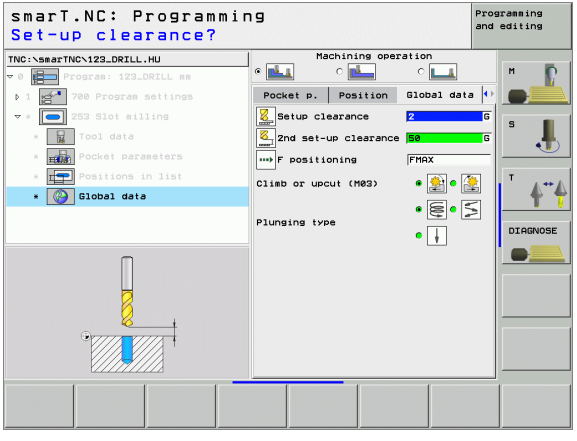
Additional parameters on the **pocket parameters** detail form:

- **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

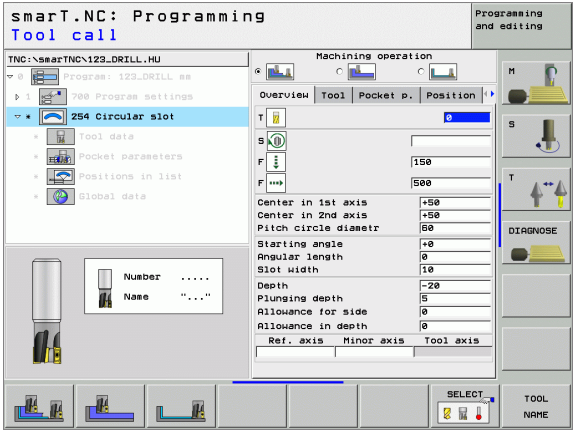




Unit 254 Circular Slot

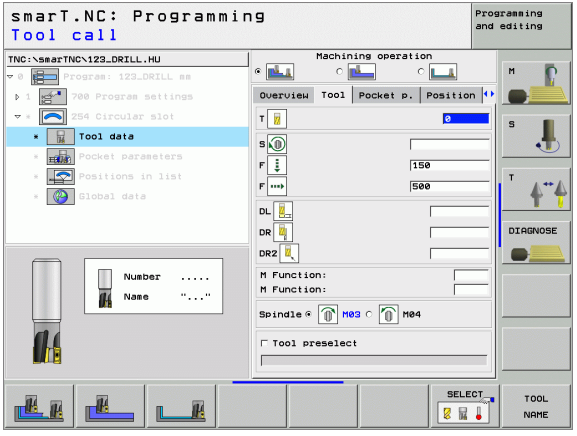
Parameters on the **overview** form:

- **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 111.)



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).

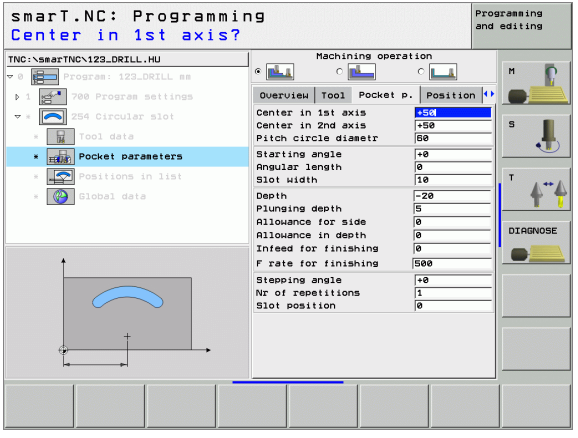


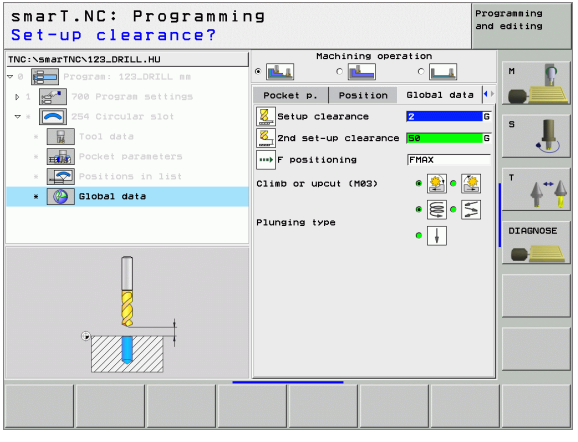
Additional parameters on the **pocket parameters** detail form:

- **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
- 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





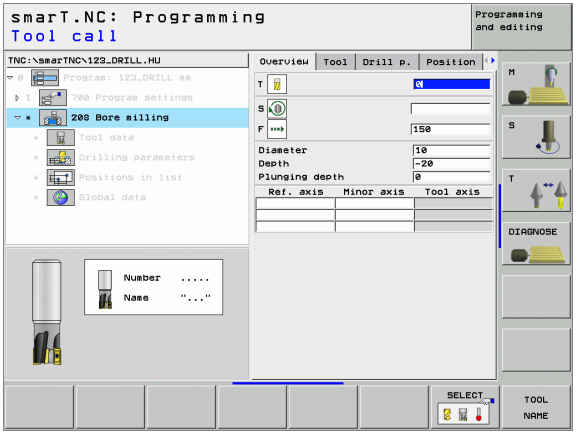
Unit 208 Bore Milling

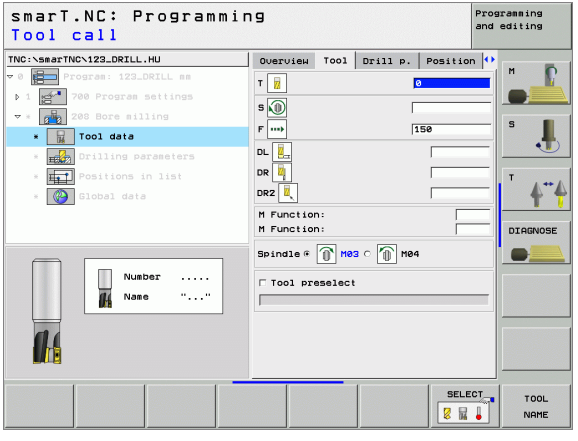
Parameters on the **overview** form:

- **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 111.)

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).



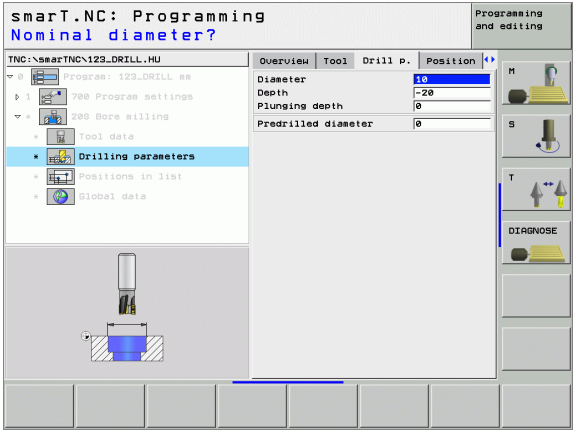


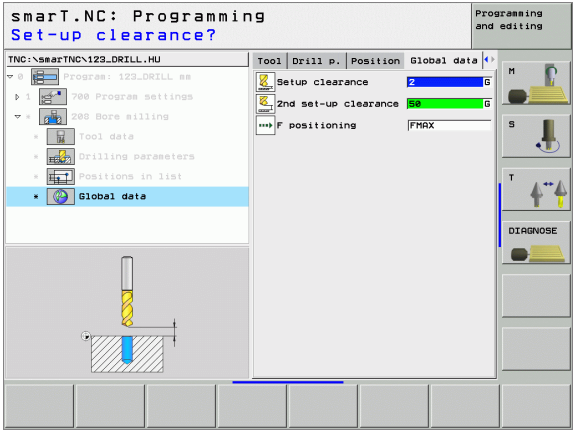
Additional parameters on the **drilling parameters** detail form:

- **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

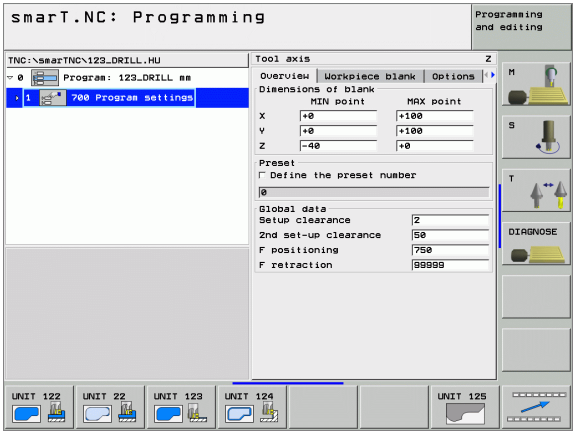




Contour Program machining group

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 78
Unit 22 Contour Pocket Fine Roughing	UNIT 22	Page 82
Unit 123 Contour Pocket Floor Finishing	UNIT 123	Page 84
Unit 124 Contour Pocket Side Finishing	UNIT 124	Page 85
Unit 125 Contour Train	UNIT 125	Page 87



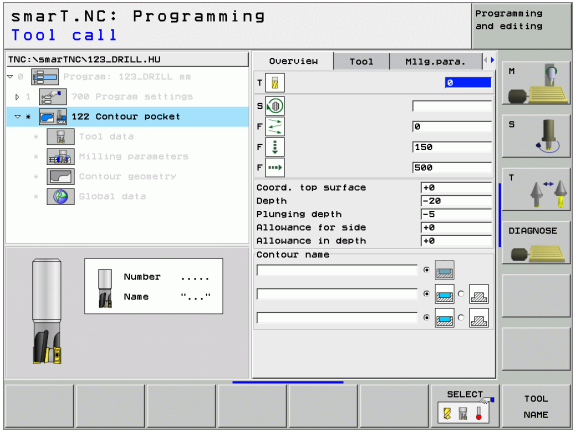
Unit 122 Contour Pocket

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.



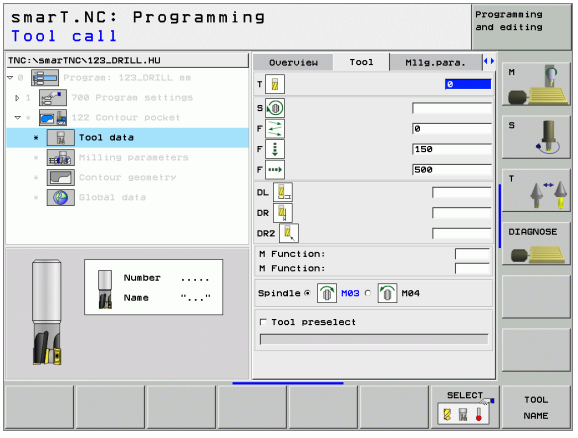
- Specify via soft key for each subcontour whether it is a pocket or an island.
- As a rule, always start the list of subcontours with a pocket (if required, the deepest pocket)!
- On the **contour** detail form you can define up to a maximum of nine subcontours (see figure at bottom right).

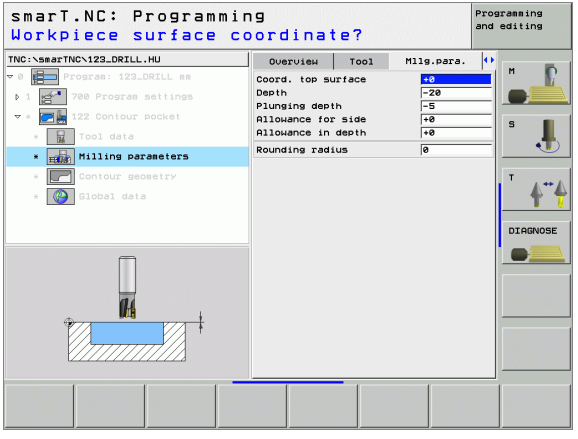
Additional parameters on the **tool** detail form:

- **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



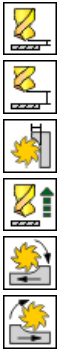


Additional parameters on the **contour** detail form:

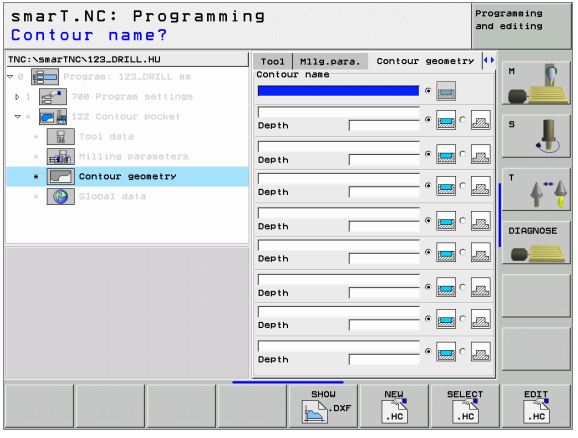
- **Depth**: Separately definable depths for each subcontour (FCL 2 function)

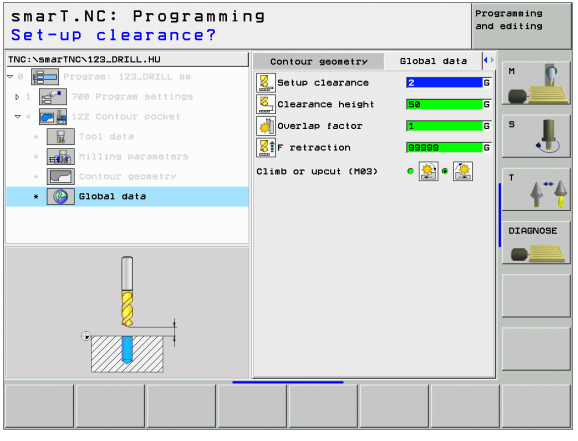
- As a rule, always start the list of subcontours with the deepest pocket!
- If the contour is defined as an island, the entered depth is the island depth (with respect to the workpiece top surface)!
- If the depth is entered as 0, the depth defined in the overview form is effective

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



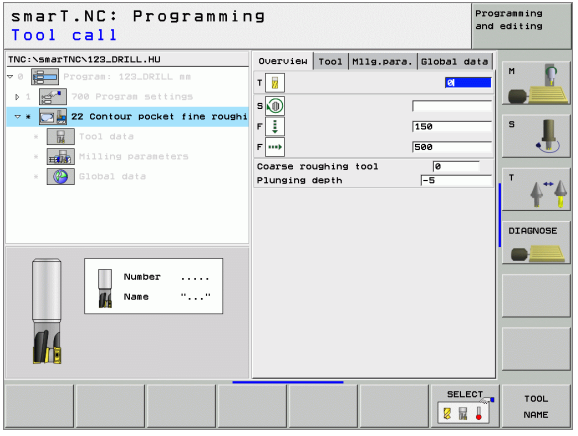


Unit 22 Fine Roughing

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 of the tool with which you have coarse-roughed the contour.
- **Plunging depth:** Infeed per cut.



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

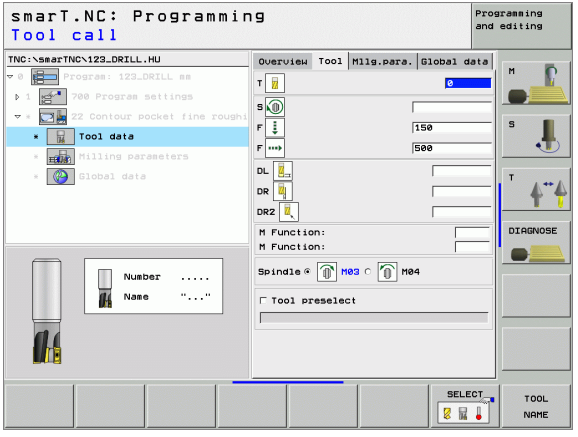
- **DL:** Delta length for tool T.
- **DR:** Delta radius for tool T.
- **DR2:** Delta radius 2 (corner radius) for tool T.
- **M function:** Any miscellaneous function M.
- **Spindle:** Direction of spindle rotation. As a default, smarT.NC sets M3.
- **Tool preselect:** If needed, this is the number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

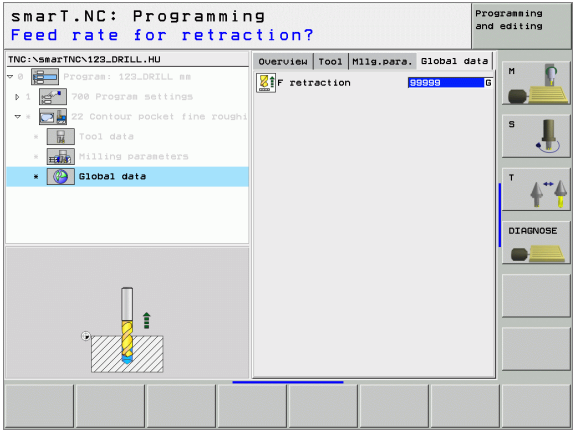
Additional parameters on the **milling parameters** detail form:

None

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

- Retraction feed rate





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:

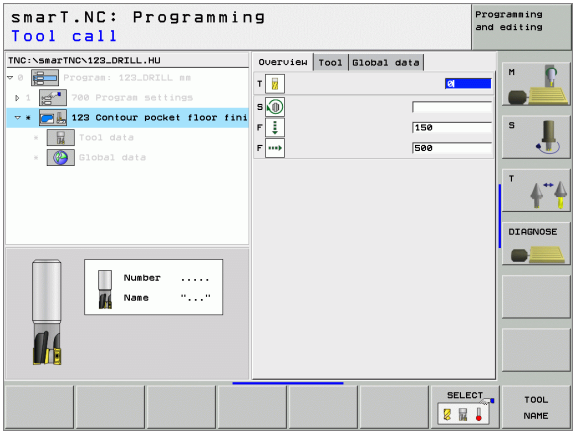
- **T:** Tool number or name (switchable via soft key)
- **S**: Spindle speed [rpm] or cutting speed [m/min or ipm]
- **F**: Feed rate for plunging [mm/min], FU [mm/rev] or FZ [mm/tooth]
- **F**: Feed rate for plunging [mm/min] or FU [mm/rev] or FZ [mm/tooth]

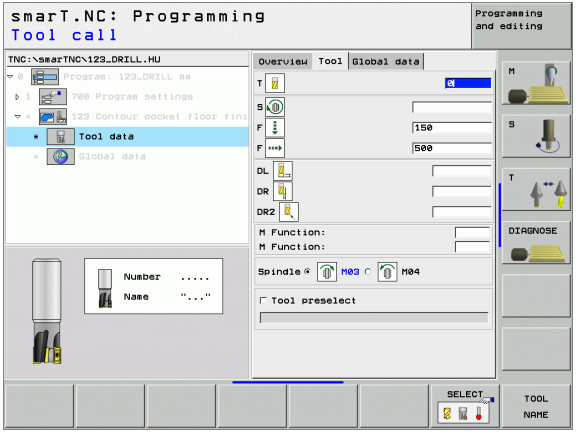
Additional parameters on the **tool** detail form:

- **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.

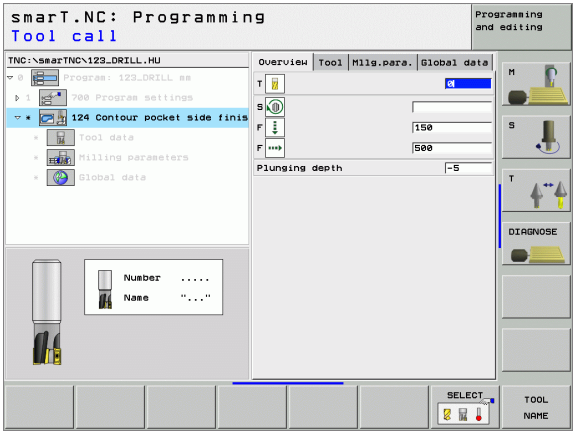
Always run the side finishing operation after floor finishing!

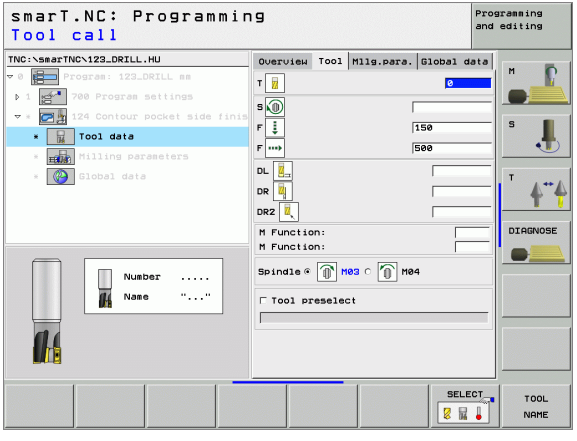
Parameters on the **overview** form:

- **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).



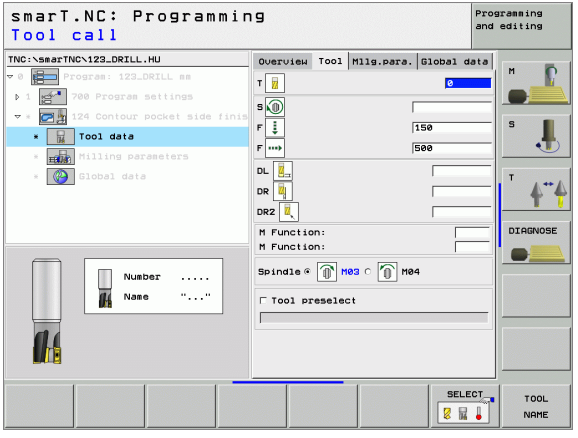


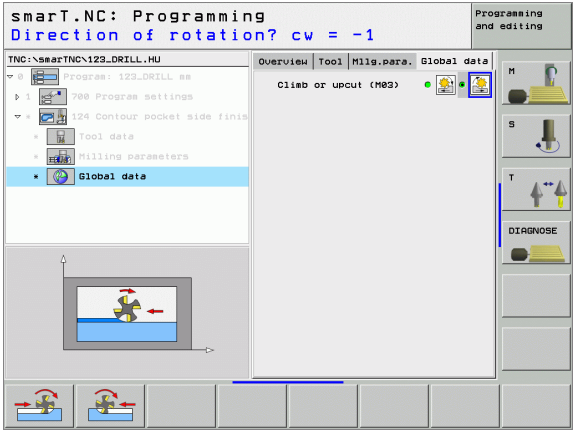
Additional parameters on the **milling parameters** detail form:

- **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

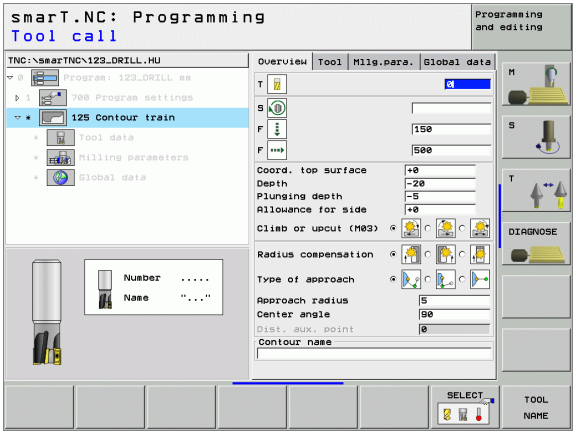




Unit 125 Contour Train

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

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.
- **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:

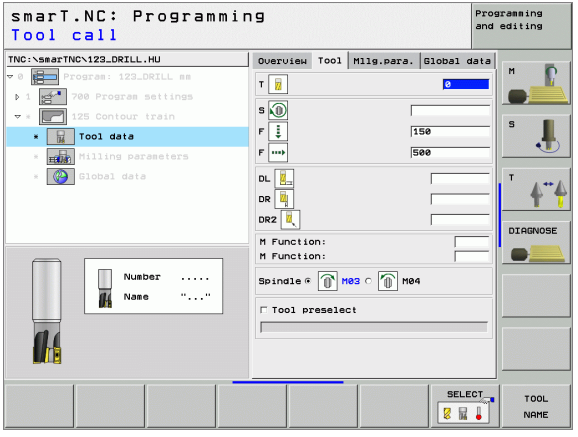
- **DL:** Delta length for tool T.
- **DR:** Delta radius for tool T.
- **DR2:** Delta radius 2 (corner radius) for tool T.
- **M function:** Any miscellaneous function M.
- **Spindle:** Direction of spindle rotation. As a default, smarT.NC sets M3.
- **Tool preselect:** If needed, this is the number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

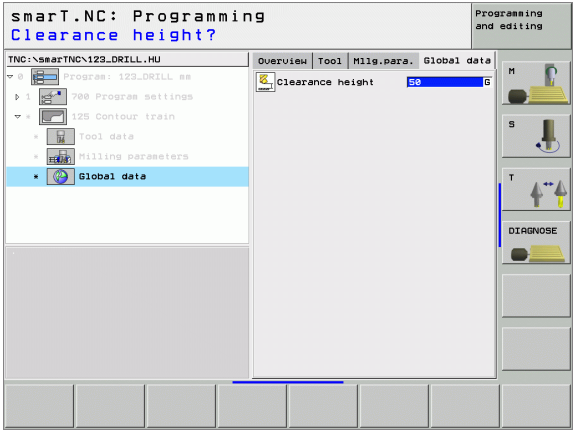
Additional parameters on the **milling parameters** detail form:

None

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

- 2nd set-up clearance

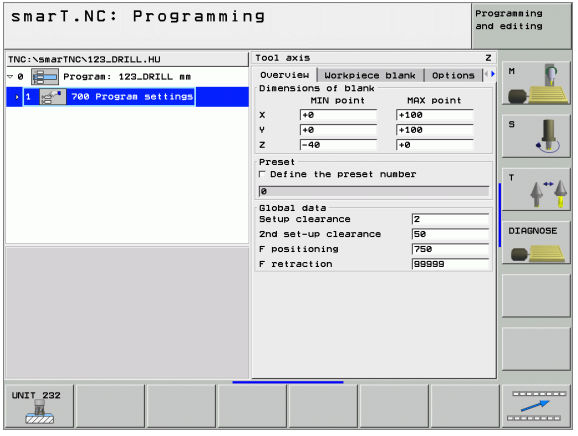




Surfaces machining group

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

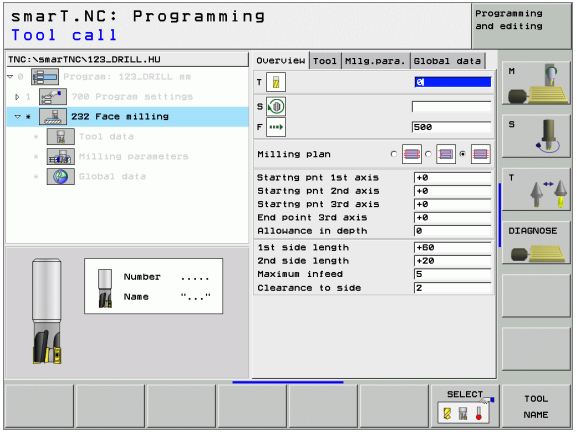
Unit	Soft key	Page
Unit 232 Face Milling	UNIT 232	Page 91



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.
- **1st 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.

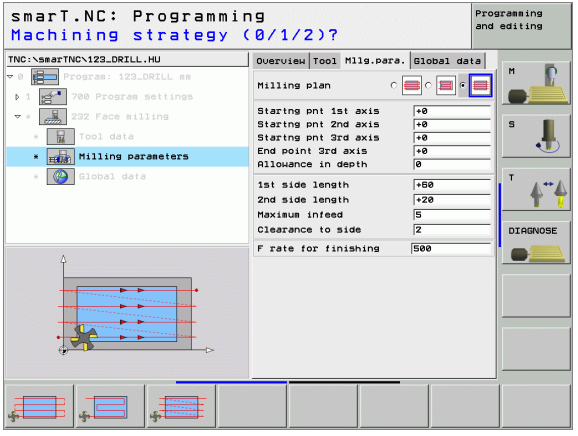


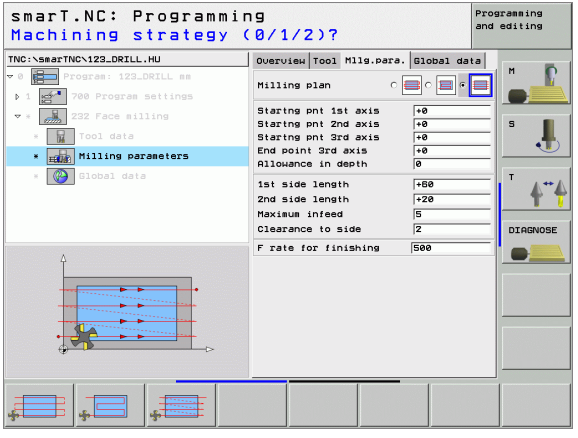
Additional parameters on the **tool** detail form:

- **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:

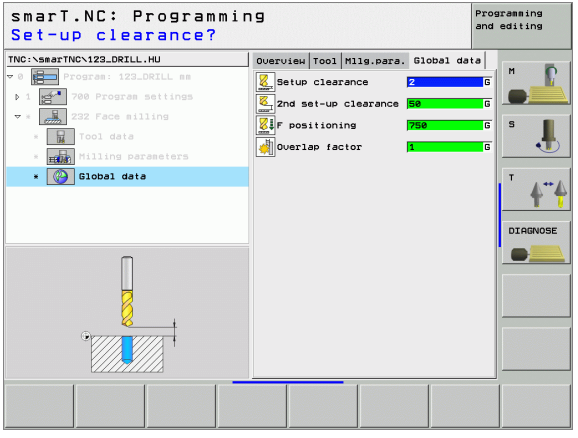
- **F rate for finishing:** Feed rate for the last finishing cut.





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

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

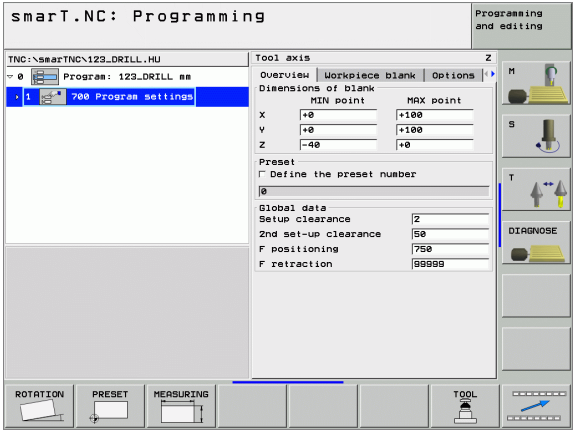


Probing main group

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

Function group	Soft key
ROTATION Touch probe functions for automatic determination of a basic rotation	ROTATION
PRESET Touch probe functions for automatic determination of a reference point	PRESET
MEASURING Touch probe functions for automatic workpiece measurement	MEASURING
TOOL Touch probe functions for automatic tool measurement	TOOL

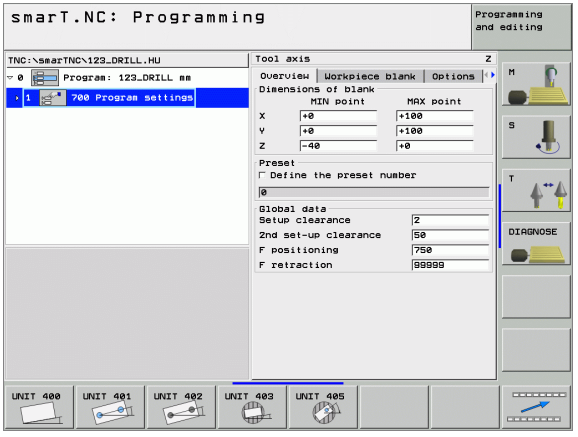
For a detailed description of the probing cycles, see the Touch Probe Cycles Manual, Chapter 4.



Rotation function group

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

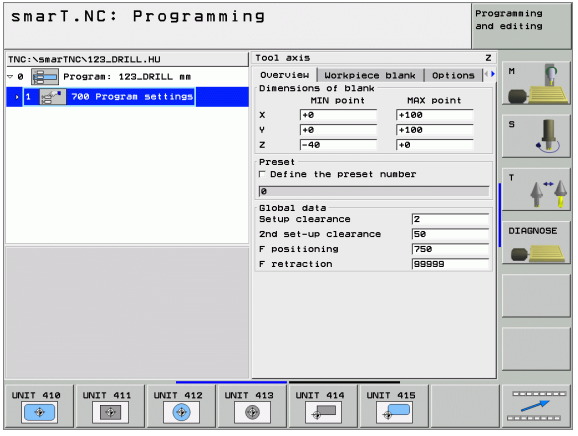
Unit	Soft key
Unit 400 Rotation around a Straight Line	
Unit 401 Rotation around two Holes	
Unit 402 Rotation around two Studs	
Unit 403 Rotation around a Rotary Axis	
Unit 405 Rotation around the C Axis	



Preset (Datum) function group

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

Unit	Soft key
Unit 410 Datum Inside Rectangle	
Unit 411 Datum Outside	
Unit 412 Datum Inside Circle	
Unit 413 Datum Outside Circle	
Unit 414 Datum Outside Corner	
Unit 415 Datum Inside Corner	
Unit 416 Datum in Circle Center	
Unit 417 Datum in Touch Probe Axis	

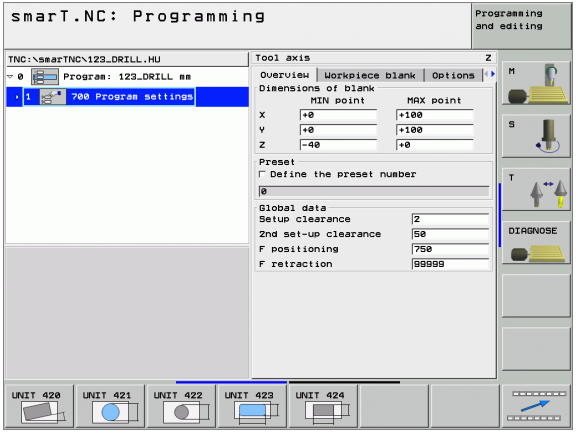


Unit	Soft key
Unit 418 Datum of four Holes	
Unit 419 Datum in one Axis	

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 420
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

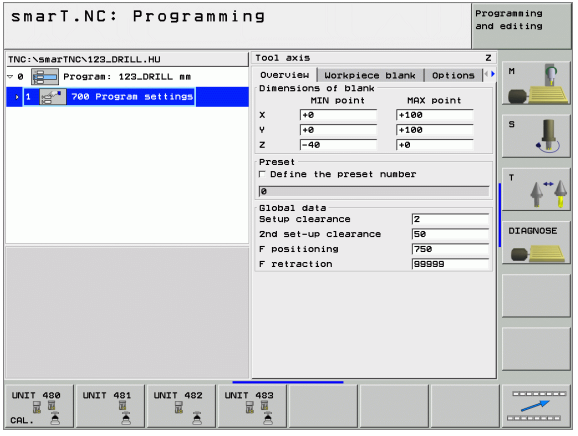


Unit	Soft key
Unit 430 Measure Bolt Hole Circle	
Unit 431 Measure Plane	

Tool function group

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

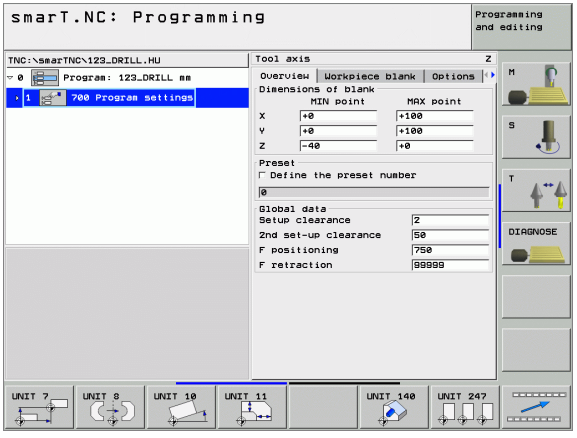
Unit	Soft key
Unit 480 TT: Calibrate TT	UNIT 480 CAL.
Unit 481 TT: Measure tool length	UNIT 481
Unit 482 TT: Measure tool radius	UNIT 482
Unit 483 TT: Measure entire tool	UNIT 483



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 102
UNIT 8 (FCL 2 function): Mirroring	UNIT 8	Page 103
UNIT 10 (FCL 2 function): Rotation	UNIT 10	Page 103
UNIT 11 (FCL 2 function): Scaling	UNIT 11	Page 104
UNIT 140 (FCL 2 function): Tilting the working plane with the PLANE function	UNIT 140	Page 104
UNIT 247: Preset number	UNIT 247	Page 106
UNIT 404 (2nd soft-key row): Set basic rotation	UNIT 404	Page 106



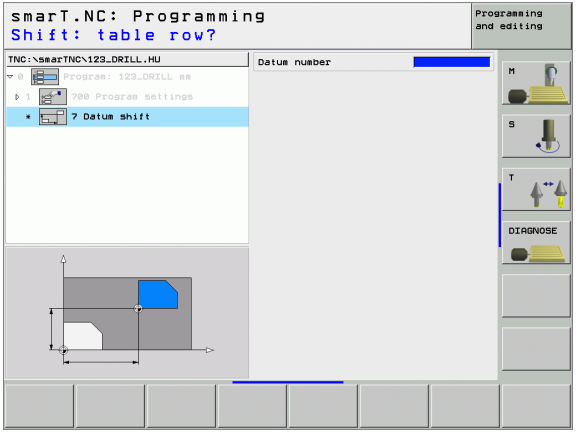
Unit 7 Datum shift (FCL 2 function)

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 29.).

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 110.).

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

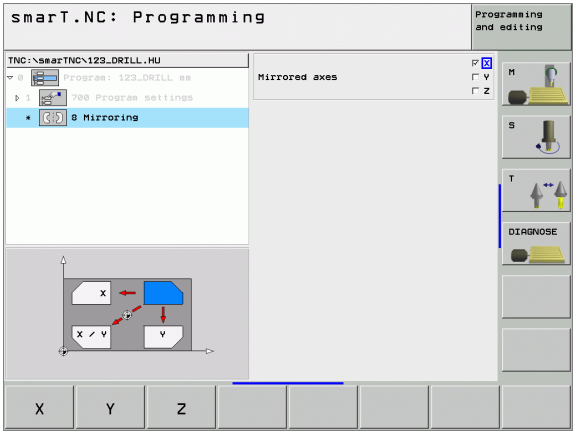


Unit 8 Mirroring (FCL 2 function)

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.

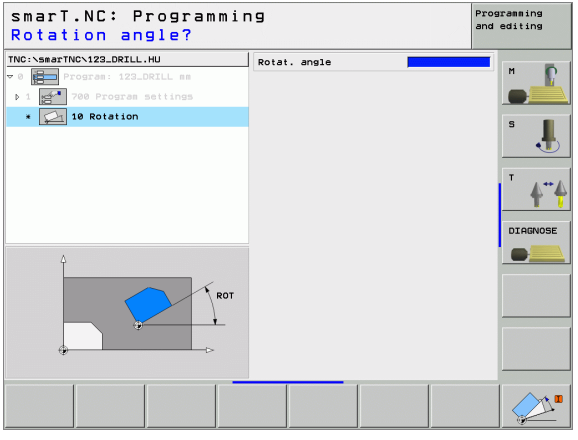


Unit 10 Rotation (FCL 2 function)

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.

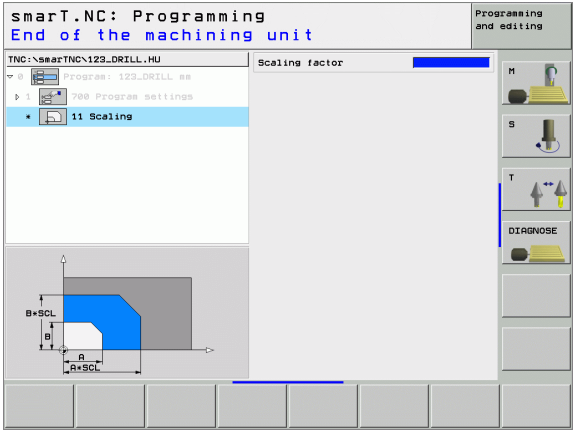
Reset scaling factor: Define unit 11 with scaling factor 1.

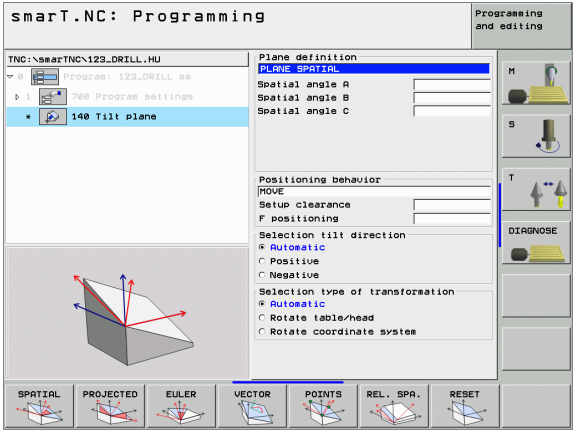
Unit 140 Tilt plane (FCL 2 function)

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).

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





The following plane definitions are available:

Type of plane definition	Soft key
Plane defined by spatial angle	
Plane defined by projection angle	
Plane defined by Euler angle	
Plane defined by vector	
Plane defined by three points	
Define an incremental spatial angle	
Reset the tilted plane function	

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

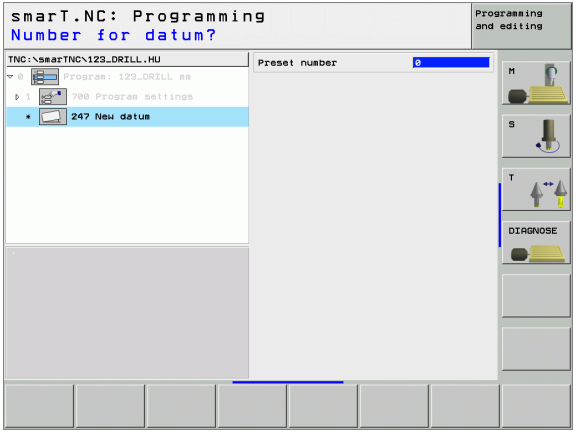
The type of transformation is effective only in transformations with a C axis (rotary table).

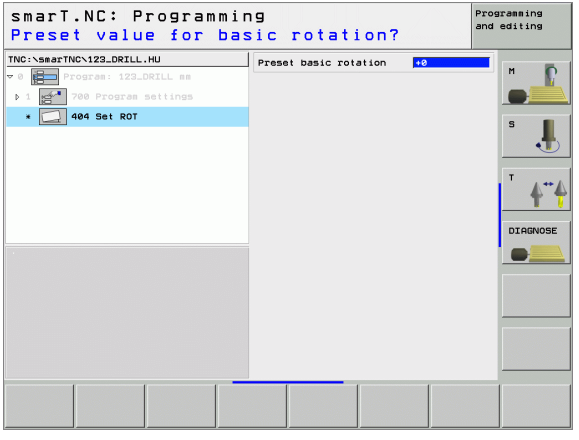
Unit 247 New Datum

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.

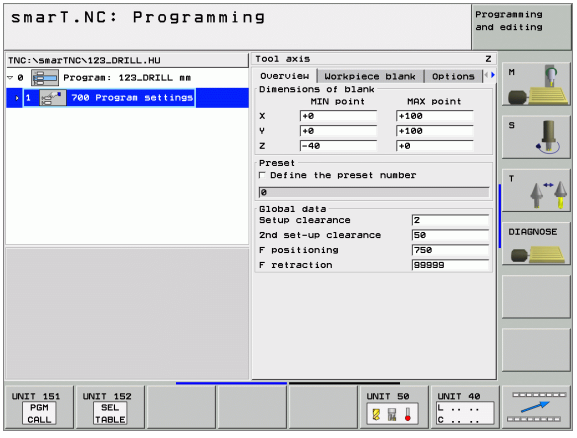




Special functions main group

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

Function	Soft key	Page
UNIT 151: Program call	UNIT 151 PGM CALL	Page 108
UNIT 50: Separate tool call	UNIT 50	Page 109
UNIT 40: Conversational unit	UNIT 40 L .. .. C	Page 110
UNIT 700 (2nd soft-key row): Program settings	UNIT 700	Page 29



Unit 151 Program Call

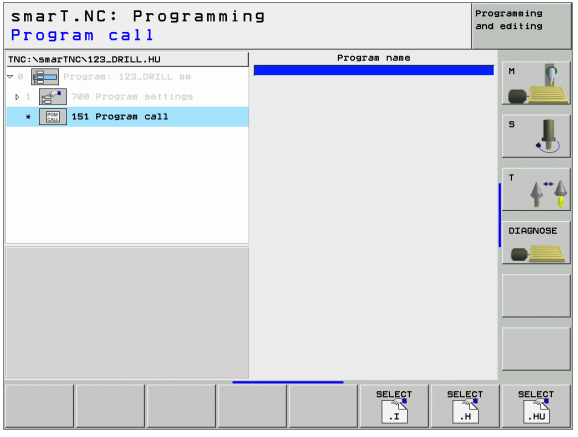
From smarT.NC you use this unit to call any program of the following file type:

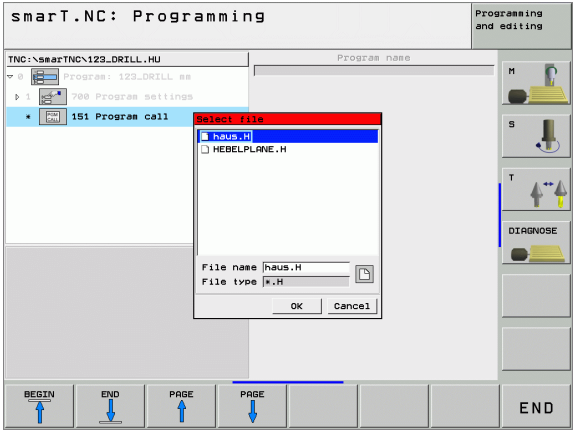
- smarT.NC unit program (file type .HU)
- Conversational dialog program (file type .H)
- ISO program (file type .I)

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.



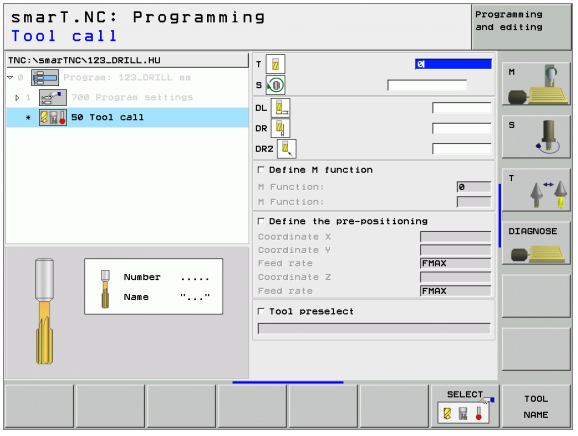


Unit 50 Separate Tool Call

You can define a separate tool call with this unit.

Parameters in the overview form:

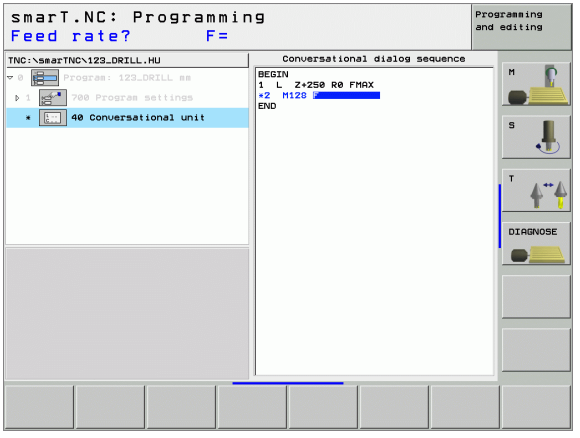
- **T:** Tool number or name (switchable via soft key)
- **S**: Spindle speed [rpm] or cutting speed [m/min or ipm]
- **DL:** Delta length for tool T.
- **DR:** Delta radius for tool T.
- **DR2:** Delta radius 2 (corner radius) for tool T.
- **Define M function:** If desired, enter any miscellaneous 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).



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 available
- You want to define OEM cycles
- You want to insert any positioning movements between units
- You want to define machine-specific M functions



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

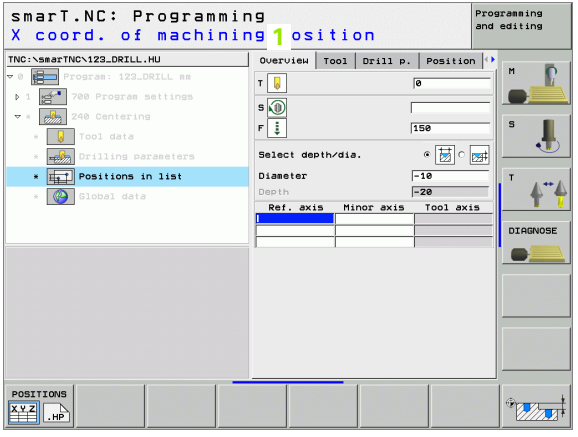
On the **overview** form (1) you can directly define the machining positions of the current machining step in Cartesian coordinates (see figure at top right). If the machining is to be performed at more than three positions, you can define up to six more positions—for a total of nine—on the **Positions**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

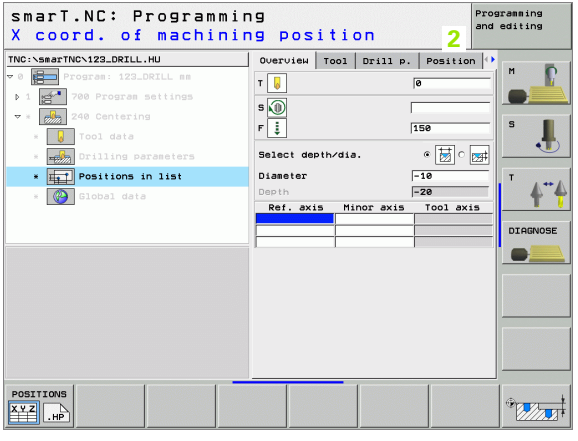
Incremental input is allowed beginning with the 2nd machining position. You can use the I key or soft key to switch over. The first machining position must be absolute.

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.





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, when you are supposed to enter machining positions.

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

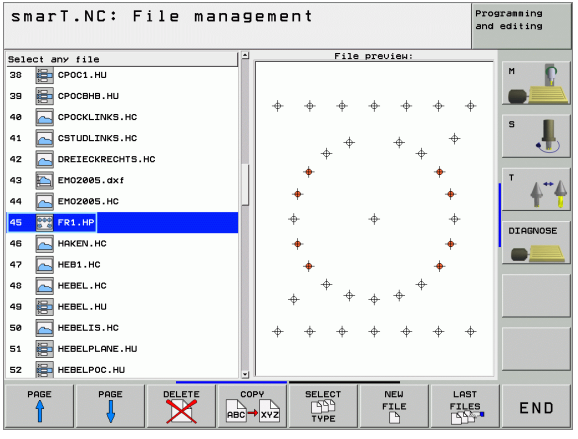
▶ Select the smarT.NC operating mode

▶ 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 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.



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.**

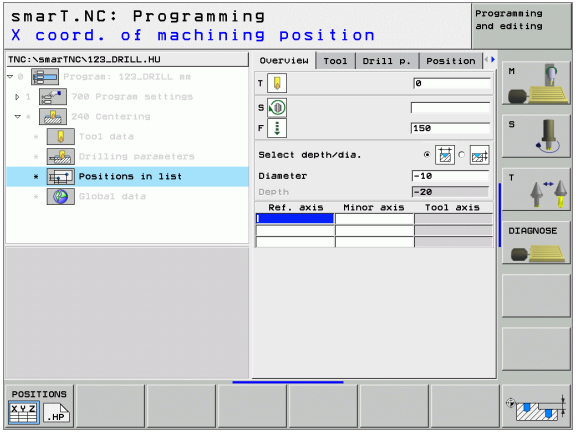
- **To create a new file:** Enter the file name (without file type), and confirm with the NEW .HP soft key.
- Specify the units of measurement for the new point file with the MM or INCH button in the pop-up window. smarT.NC then starts the pattern generator.

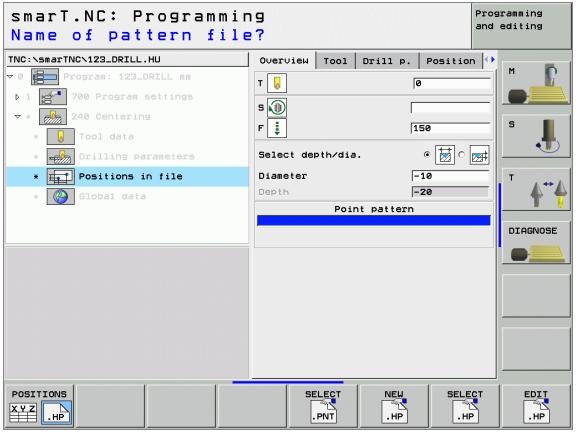
- **To select an existing HP file:** Press the SELECT .HP soft key. smarT.NC opens a pop-up window with available point files. Select one of the displayed files, and open it with the ENT key or OK screen button.

- **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. 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.





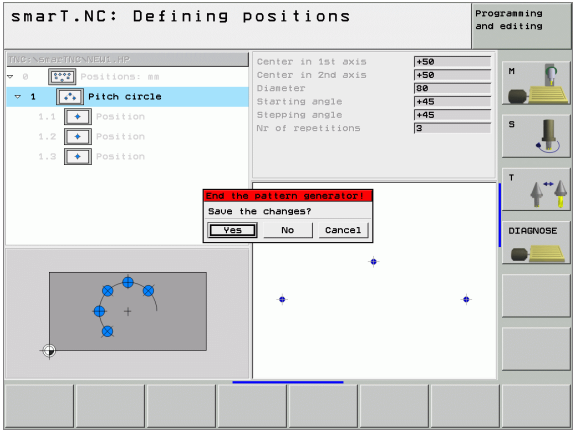
Exiting the pattern generator

END

- Press the END key or soft key. smarT.NC opens a pop-up window (see figure at right).
- Press the ENT key or YES screen 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 screen 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.

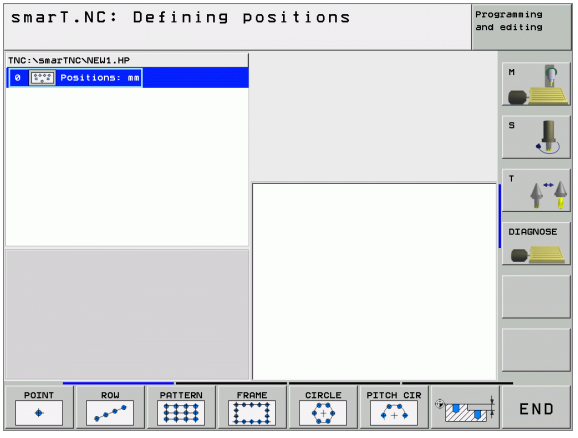


Working with the pattern generator

Overview

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

Function	Soft key	Page
Single point, Cartesian	POINT	Page 119
Single row, straight or arced	ROW	Page 119
Pattern straight, arced or distorted	PATTERN	Page 120
Frame straight, arced or distorted	FRAME	Page 121
Full circle	CIRCLE	Page 122
Circular arc	PITCH CIR	Page 123
Change starting height		Page 124



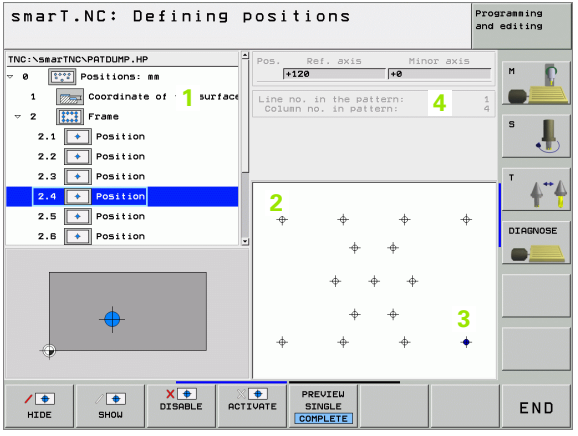
Defining a pattern

- 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).

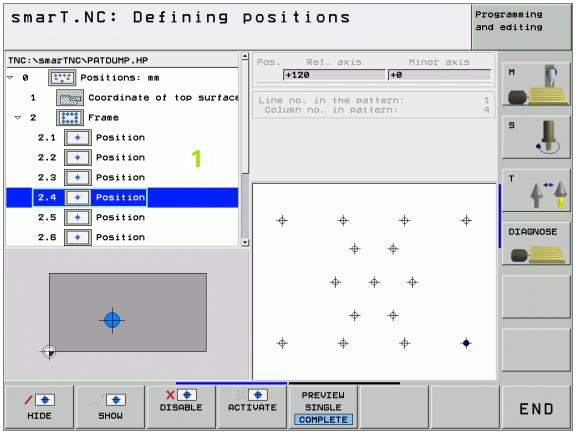
The pattern is displayed graphically in the bottom right quarter of the screen (2) immediately after the entry parameters have been saved.

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. smarT.NC marks the point selected on the left blue in the graphic on the right (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	Soft key
Hide the pattern or position selected in the tree view for machining. Hidden patterns or positions are marked in the tree view (1) with a red slash, and in the preview graphics with a bright red dot.	HIDE
Reactivate a hidden pattern or position.	SHOW
Disable the position selected in the tree view for machining. Disabled positions are marked in the tree view (1) 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.	DISABLE
Reactivate disabled positions	ACTIVATE
Exporting 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	OUTPUT .PNT
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.	PREVIEW SINGLE COMPLETE
Detail enlargement: Show and move the frame overlay. Press one of the arrow keys repeatedly to shift it (second soft-key row).	



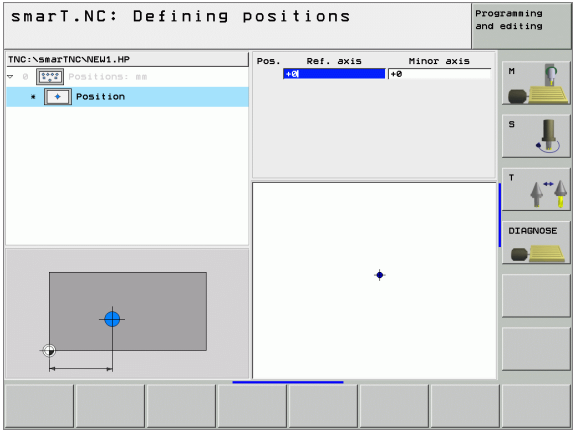
Function	Soft key
Detail enlargement: Reduce the frame overlay (second soft-key row).	
Detail enlargement: Enlarge the frame overlay (second soft-key row).	
Detail enlargement: Select marked area (second soft-key row).	TRANSFER DETAIL
Detail enlargement: Restore original section (second soft-key row).	WINDOW BLK FORM

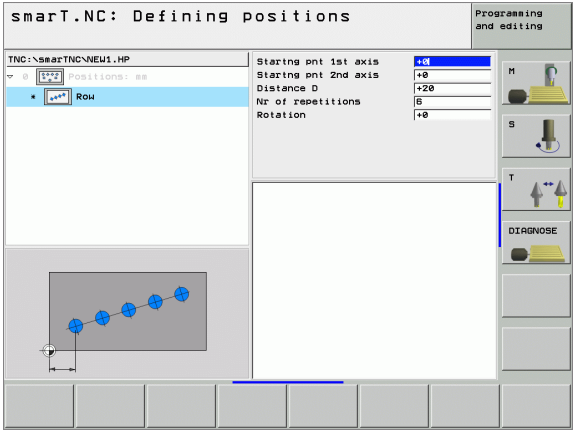
Single point, Cartesian

- **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.



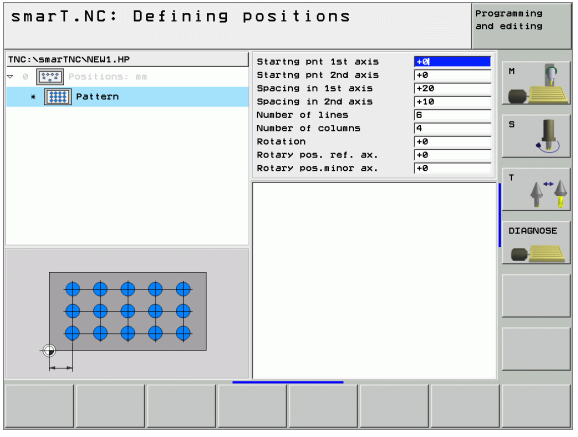


Pattern, straight, rotated or distorted

PATTERN

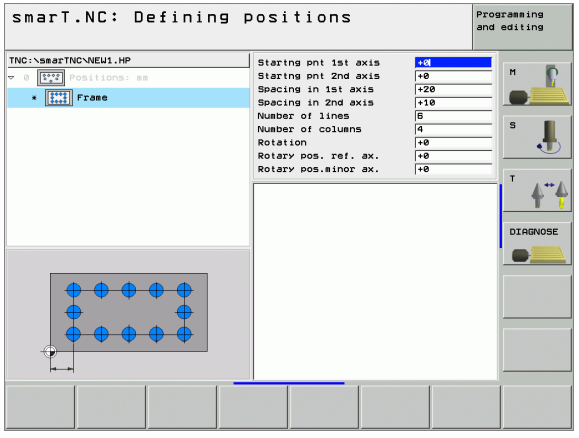
- **Starting point 1st axis:** Coordinate of the starting point of the pattern (1) in the major axis of the working plane.
- **Starting point 2nd axis:** Coordinate of the starting point of the pattern (2) in the minor axis of the working plane.
- **Distance in 1st axis:** Distance of the machining positions 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.



Frame straight, rotated or distorted

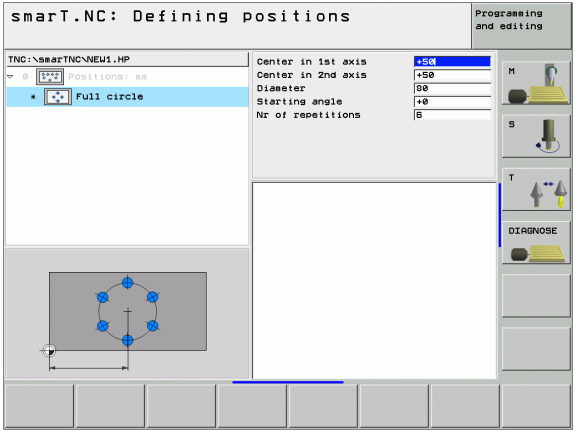
- **Starting point 1st axis:** Coordinate of the starting point of the frame (1) in the major axis of the working plane.
- **Starting point 2nd axis:** Coordinate of the starting point of the frame (2) in the minor axis of the working plane.
- **Distance in 1st axis:** Distance of the machining positions 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.

Full circle

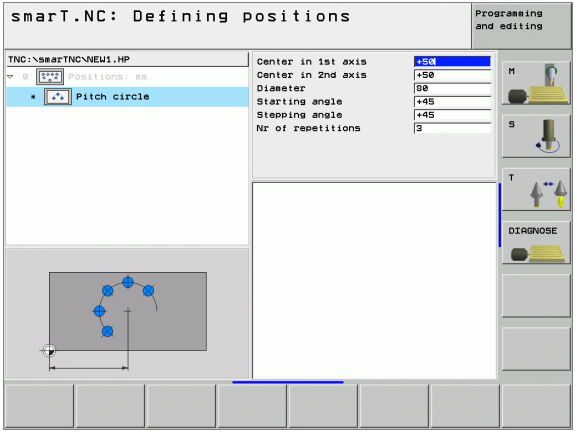
- **Center in 1st axis:** Coordinate of the circle center point (1) in the major axis of the working plane.
- **Center in 2nd axis:** Coordinate of the circle center point (2) in the minor axis of the working plane.
- **Diameter:** Circle diameter.
- **Starting angle:** Polar angle of the first machining 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.
- **Number of repetitions:** Total number of machining positions on the circle.



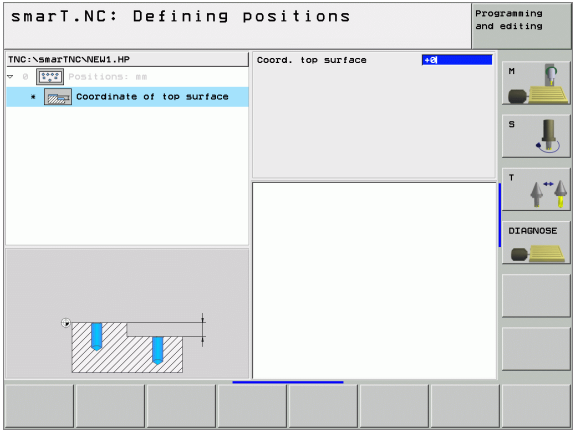
Change starting height

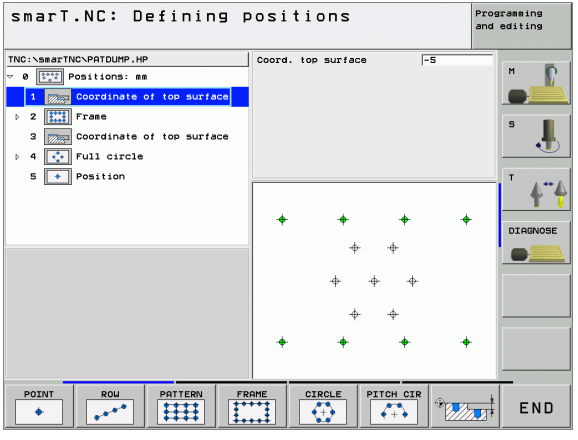
- **Top surface coordinate:** Coordinate of the top surface of the workpiece.

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).





Defining contours

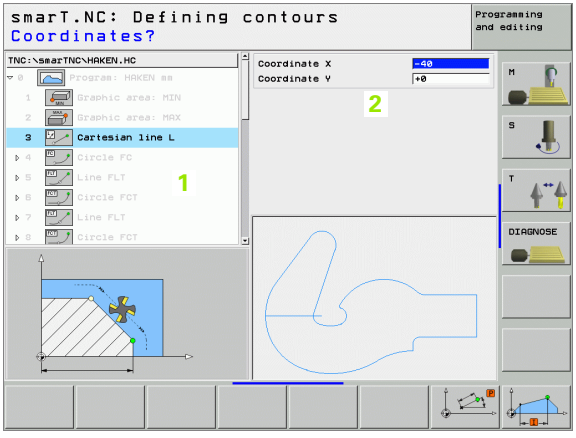
Fundamentals

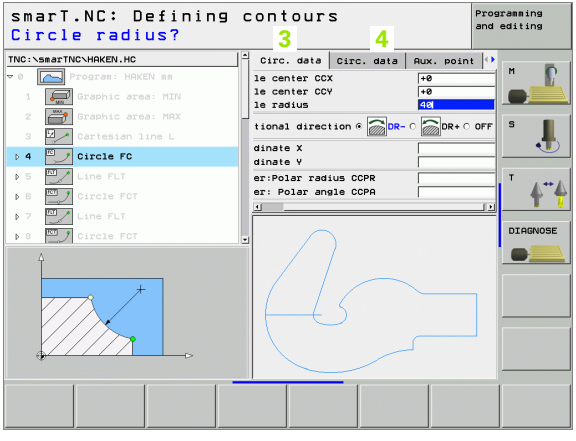
Contours are defined in separate files (file type **.HC**). Since .HC files contain pure descriptions of contours—only geometry data, no technology data—they can be used flexibly: as contour trains, as pockets or as islands.

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 132).

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).





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, when you are supposed to enter the names of the contours to be edited.

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

▶ Select the smarT.NC operating mode

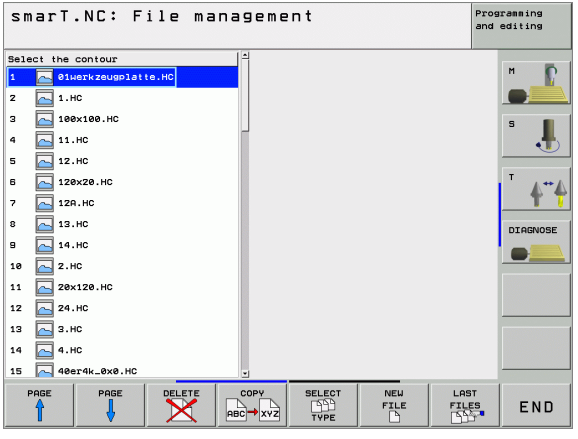
▶ Select the third soft-key row

▶ 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

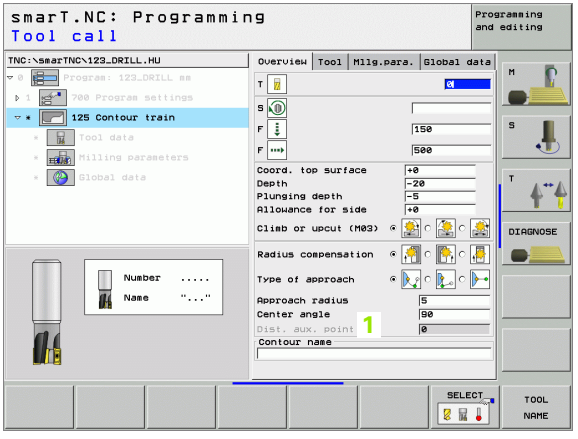
▶ Create a new contour program: Enter the file name (without file type), and confirm with the MM or INCH key. smarT.NC opens a contour program with the units of measurement you selected.

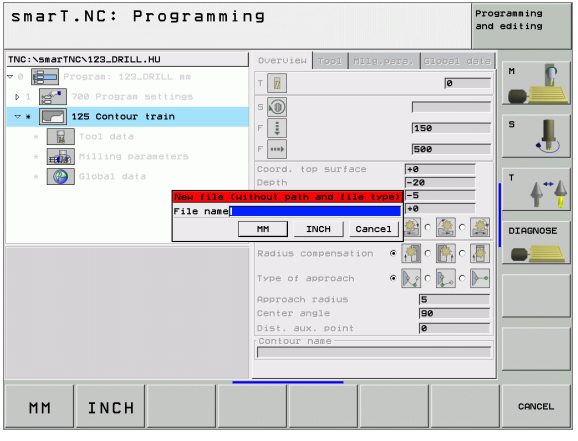
▶ smarT.NC automatically inserts two lines for defining the drawing surface. If required, adjust the dimensions



Starting contour programming from a form

- 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 screen 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 names directly in the form (see "Generating Contour Programs from DXF Data (Software Option)" on page 133).





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

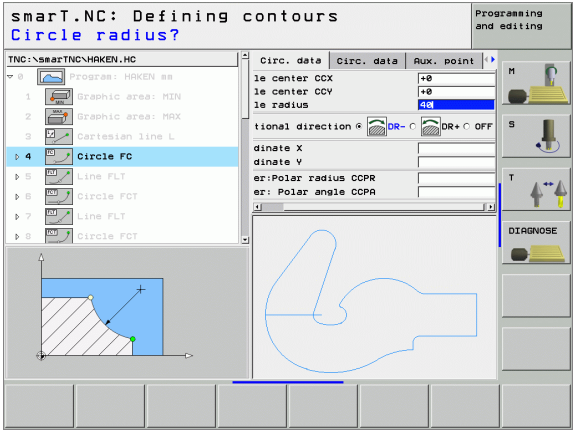
The contour elements are programmed using the familiar conversational dialog functions. Along with the gray path functions keys, the powerful FK free contour programming is also available. These forms are called via soft keys.

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.

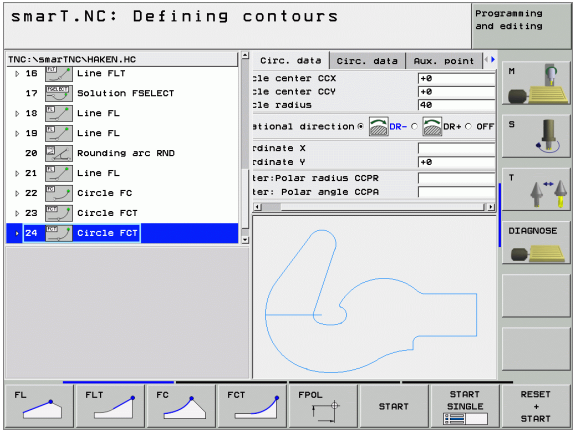
Dialog guidance in the forms is almost identical with that in conversational programming:

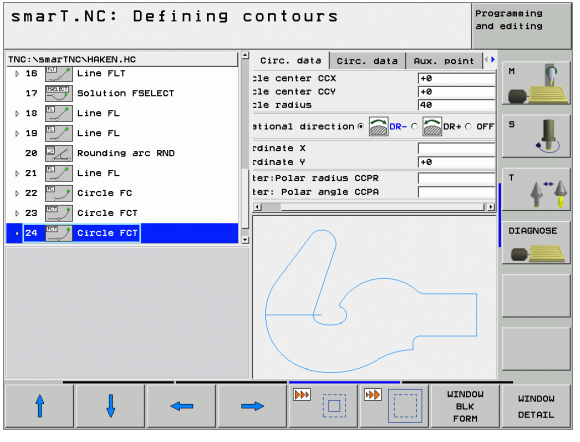
- The orange axis keys position the cursor in the desired input field
- With the orange I key you can switch between absolute and incremental programming
- With the orange P key you can switch between Cartesian and polar coordinate programming



Functions

Function	Soft key
Generate a complete graphic	RESET + START
Generate interactive graphic blockwise	START SINGLE
Generate a complete graphic or complete it after RESET + START	START
Stop the programming graphics. This soft key only appears while the TNC is generating the interactive graphics	STOP
Zoom function (third soft-key row): Reduce the section. Press the soft key repeatedly for further reduction.	
Zoom function (third soft-key row): Enlarge the section. Press the soft key repeatedly for further enlarging.	
Zoom function (third soft-key row): Show and move the frame.	





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:

SHOW SOLUTION

▶ Show the possible solutions.

SELECT SOLUTION

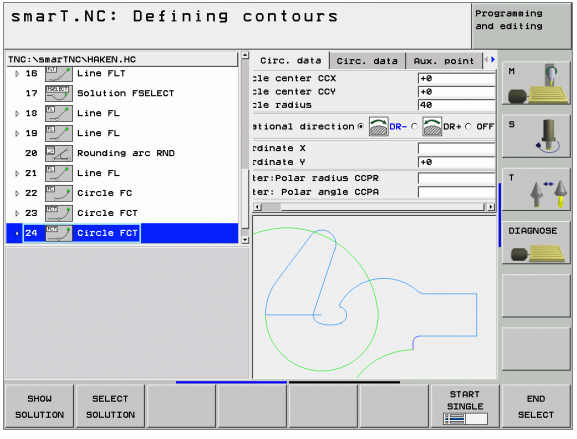
▶ Enter the displayed solution in the part program.

END SELECT

▶ Enter data for subsequent contour elements.

START SINGLE

▶ Graphically display the next programmed block.



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

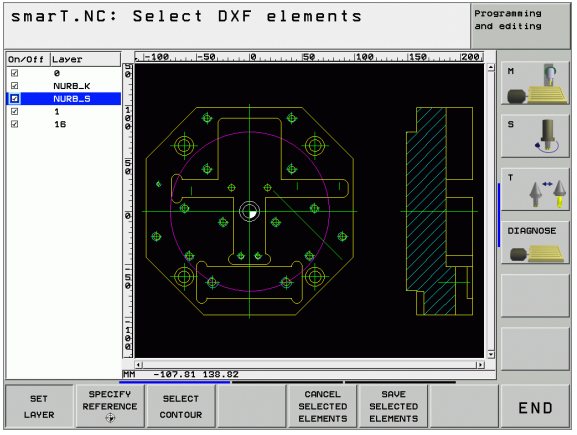
- Select the copy function: Enter ***.A** as the target file. The TNC creates an ASCII file from the conversational dialog program.
- Select the created ASCII file.

- 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.

Generating Contour Programs from DXF Data (Software Option)

Function

DXF files created in a CAD system can be opened directly in smarT.NC to extract contours and save them as contour programs (.HC files).



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

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)

Opening a DXF file

The DXF converter can be started two different ways:

- From the file management if you want to extract separate contours consecutively.
- From the form during the machining definition of units 125 (contour train) and 122 (contour pocket) if you are supposed to enter the names of the contours to be edited.

Starting the DXF converter through the file management

- Select the smarT.NC operating mode

- Call the file manager

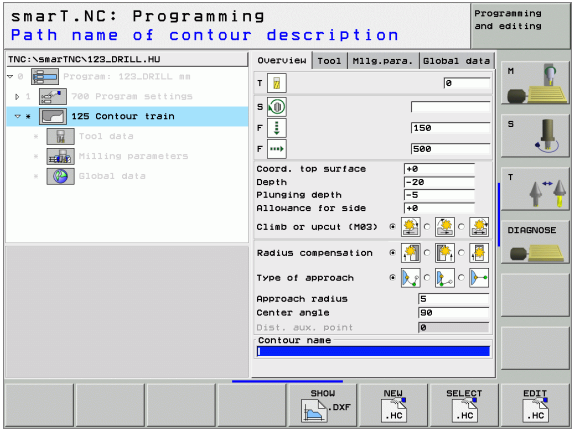
- In order to see the soft-key menu for selecting the file type to be displayed, press the SELECT TYPE soft key.

- To show all DXF files, press the SHOW DXF soft key.

- 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 layers in the left window, and the drawing in the right.

Starting the DXF converter from a form

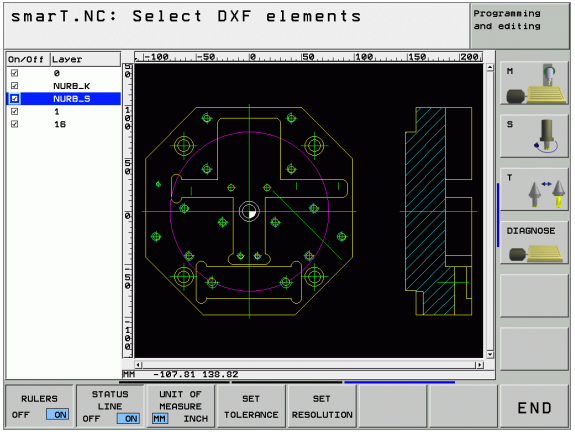
- Select the smarT.NC operating mode
- Select any machining step for which contour programs are required (Unit 122, Unit 125).
- Select the input field in which the name of the contour program is to be defined.
- **To start the DXF converter:** Press the SHOW DXF soft key. smarT.NC opens a pop-up window 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 names directly in the form (see "Generating Contour Programs from DXF Data (Software Option)" on page 133).



Basic settings

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

Setting	Soft key
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.	RULERS OFF ON
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: ■ Active unit of measurement (MM or INCH) ■ X and Y coordinates of the current mouse position	STATUS LINE OFF ON
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.	UNIT OF MEASURE MM INCH
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. Default setting: 0.1 mm	SET TOLERANCE



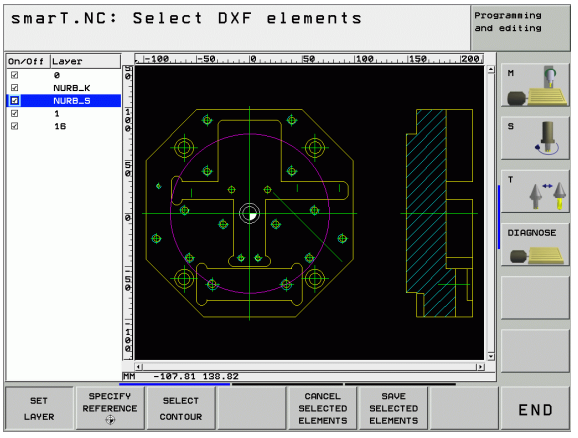
Setting	Soft key
Set Resolution: The resolution specifies how many decimal places the TNC should use when generating the contour program. Default setting: 4	SET RESOLUTION

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 LAYER

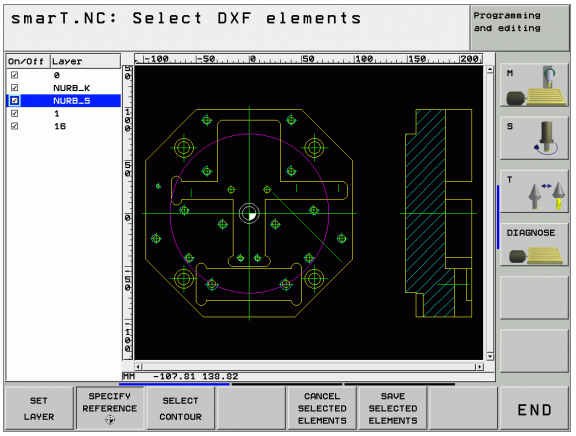
- If it has not already been activated, select the mode for the layer settings. In the left window the TNC shows all layers contained in the active DXF file.
- To hide a layer, select the layer with the left mouse button, and click its checkbox to hide it.
- To show a layer, select the layer with the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

Datum specifying

The datum of the drawing for the DXF file is not always located in 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



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

- 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

- Select 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.

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 CONTOUR

- 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.

SAVE SELECTED ELEMENTS

- 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

ENT

- Confirm the entry: The TNC saves the contour program in the directory in which the DXF file is also saved.

CANCEL SELECTED ELEMENTS

- 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.

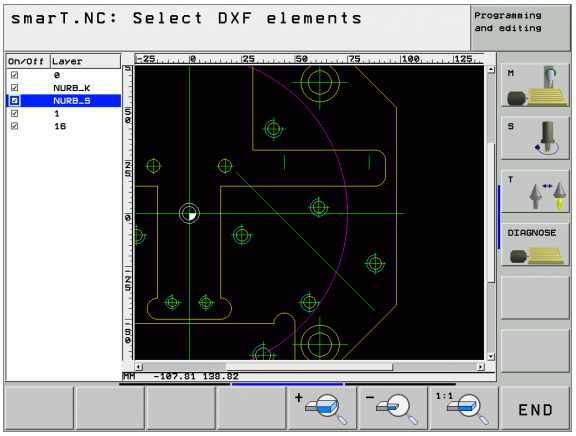
The TNC only saves elements that have actually been selected (blue elements).

If you call the DXF converter from a form, smarT.NC automatically closed 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.

Zoom function

The TNC features a powerful zoom function for easy recognition of small details during contour 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	



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.

In the **SET LAYER** and **SPECIFY REFERENCE** modes you can move the zoomed-in window with the cursor keys.

Graphically test and run a UNIT program

Interactive programming graphics

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

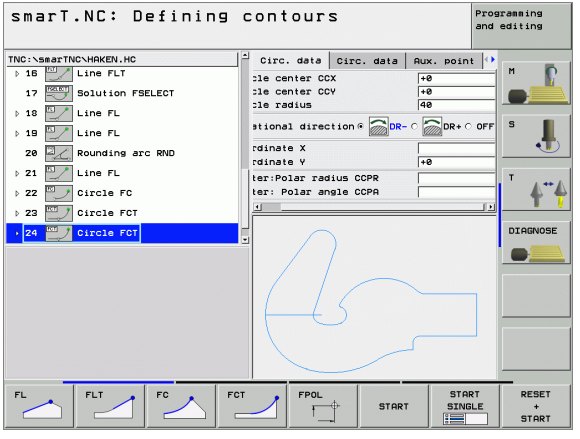
▶ Start and complete the graphic

▶ Automatic graphic generation during programming

▶ Erase the graphic

▶ Update the graphic

▶ Display or hide block numbers



Test Graphics and Execution Graphics

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

- 3-D view

- High-resolution 3-D view

- Functions for section magnification

- Functions for the sectional planes

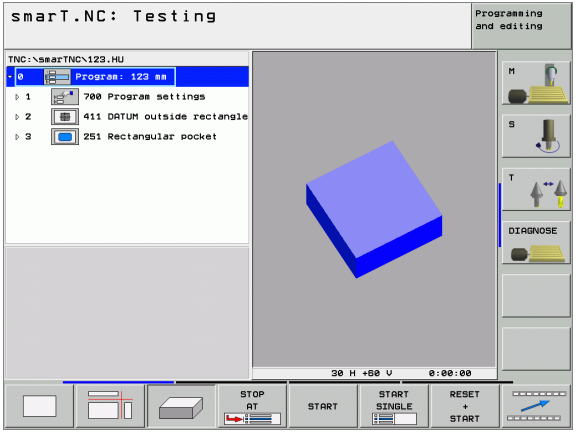
- Functions for rotating and magnifying/reducing

- Selecting the stopwatch functions

- Setting the simulation speed

- Function for measuring the machining time

- Consider or ignore program blocks preceded by a slash



Status displays

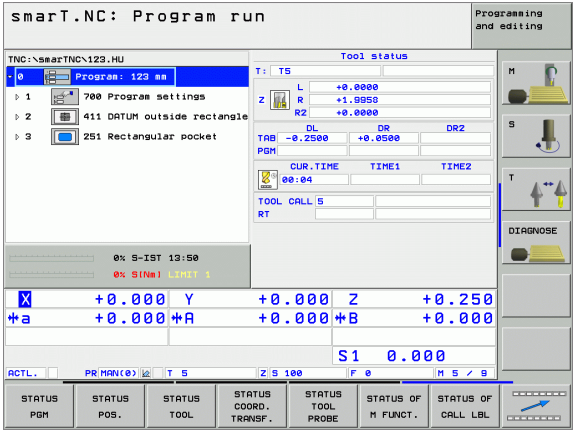
Select the PROGRAM+STATUS layout.

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

- Tool position
- Feed rate
- Active miscellaneous functions

Further status information is available via soft key for display in an additional window:

STATUS PGM	Program information
STATUS POS.	Tool positions
STATUS TOOL	Tool data
STATUS COORD. TRANSF.	Coordinate transformations
STATUS OF CALL LBL	Subprograms, program section repeats
STATUS TOOL PROBE	Tool measurement
STATUS OF M FUNCT.	Active miscellaneous functions M



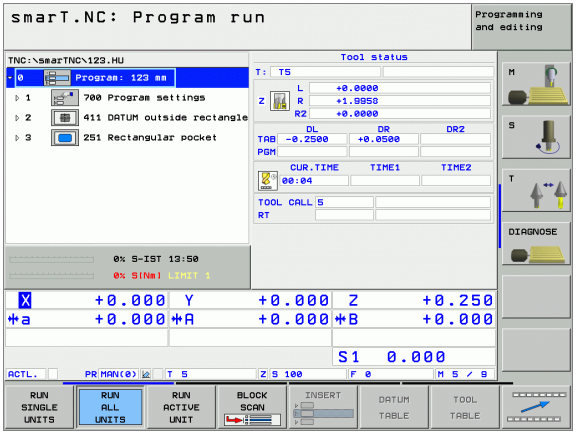
The Execute UNIT program

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

A UNIT program can be run in the Execute submode in the following manners:

- 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.



Procedure

▶ Select the smarT.NC operating mode

▶ Select the Execute submode

▶ Press the RUN SINGLE UNIT soft key, or

▶ Press the RUN ALL UNITS soft key, or

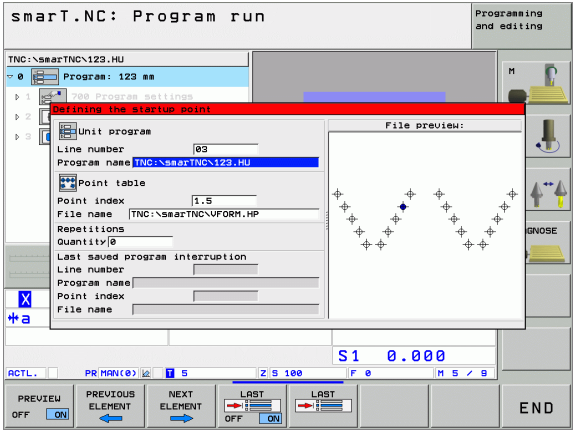
▶ Press the RUN ACTIVE UNIT soft key

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.



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

- Select the smarT.NC operating mode

- Select the Execute submode

- Select mid-program startup
- 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 button: 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 button: 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 button. 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

HEIDENHAIN

DR. JOHANNES HEIDENHAIN GmbH
Dr.-Johannes-Heidenhain-Straße 5
83301 Traunreut, Germany
☏ +49 (8669) 31-0
FAX +49 (8669) 5061
e-mail: info@heidenhain.de

Technical support FAX +49 (8669) 31-1000
e-mail: service@heidenhain.de
Measuring systems ☏ +49 (8669) 31-3104
e-mail: service.ms-support@heidenhain.de
TNC support ☏ +49 (8669) 31-3101
e-mail: service.nc-support@heidenhain.de
NC programming ☏ +49 (8669) 31-3103
e-mail: service.nc-pgm@heidenhain.de
PLC programming ☏ +49 (8669) 31-3102
e-mail: service.plc@heidenhain.de
Lathe controls ☏ +49 (711) 952803-0
e-mail: service.hsf@heidenhain.de

www.heidenhain.de

HEIDENHAIN (G.B.) Limited
200 London Road, Burgess Hill
West Sussex RH15 9RD, Great Britain
☏ (01444) 247711
FAX (01444) 870024

Ve 01
533 191-21 · SW02 · 25 · 10/2005 · Bi · Printed in Germany · Subject to change without notice